Scientific Journal Impact Factor (SJIF): 5.278
International Journal of Modern Trends in Engineering and Research
ISSN (ONLINE) : 2349 - 9745 ISSN (PRINT) : 2393 - 8161

CFD ANALYSIS OF NON PRISMATIC COMPOUND CHANNEL

Shailesh Jaiswal[1] and Dr.Bharat Jhamnani [2]
[1,2]Civil Engineering Department, College, Delhi Technological University

Abstract—Experimental results are validated with CFD results concerning the velocity profile and boundary shear stress in a non prismatic compound channel comprising of one channel which is trapezoidal in shape and two symmetrical flood plains. The flood plains were made rough so that the effect of differential roughness between the main channel and the flood plains on the lateral momentum transfer process can be analysed. The flow process in the open channel becomes more complex at over bank stages due to the various hydraulic condition prevailing in the main channel and the adjacent flood plain. As the shallow flood plains provide more friction to flow than the deep main channel, the velocity seems to be higher in deeper main channel than the shallow flood plain. This deviation of velocity between deep main channel section and the adjacent shallow flood plains increase the lateral momentum transfer, which further makes the flow process complicated. 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 comprises of a main channel and symmetric flood plains set at a fixed bed slope. Considerable agreement between the numerical results and experimental data is shown for steady uniform flow which is located at a section at a distance of 8m from the starting section.
Keywords—CFD, momentum exchange, apparent shear stress, non prismatic compound channel, roughened flood plains, Ansys.

I. INTRODUCTION

An open channel is a passage in which liquid flows with a free surface .In other words only atmospheric pressure is acting over the free surface. An open channel can be of any type natural or artificial. Depending upon the shape, a channel can be classified as either prismatic or non-prismatic. A channel is known as prismatic when the cross section is uniform and its bed slope is constant. For example rectangular, trapezoidal, circular, parabolic. A channel is considered as non-prismatic when its cross section and slope changes. Ex: River & Stream. It has been observed that the river generally exhibit a two stage geometry (deep 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 consists of a wider and rougher floodplain than the main channel. The flow process in the open channel becomes more complex at overbank stages due to the various hydraulic conditions prevailing in the main channel and the adjacent floodplains. For overbank stage, the resulting velocity distribution is normally 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 friction to flow than the deep main channel. The velocity variation increases lateral momentum transfer between the deep main channel section and the adjacent shallow floodplains, which further makes the flow process complex, leading to the irregular distribution of flow and shear stress in the main channel and floodplain areas.

In prismatic compound channels with rough floodplains the resulting interactions and momentum exchange is increased. This extra momentum exchange is an important parameter and should be taken into consideration in the overall flow modeling of a river. So research is still in process to develop methods which are physics based, have universal applicability and simple to apply. Since it is practically difficult to obtain 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 reliable method to provide the information concerning the details of the flow structure. Such information is relevant 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 movement, to prevent bank erosion in river channel, design stable channels, flood risk management, etc.). Knight and Hammed (1984)[2] extended the work of Knight and Demetrious (1983)[1], to the compound channels having rough floodplains. By adding roughness features, the floodplains were roughened. They studied the effect 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)[6] presented of an experimental results of a compound channel having fixed and movable main channel along with two rough floodplains. They inspected velocity and discharge relationships showing the complex behavior of compound channel river section. Hin and Bessaih (2004)[8] examined stage-discharge relationship, velocity distribution 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)[9] examined the reliability and performance of four different one dimensional methods of calculating the discharge capacity for compound channels by conducting a series of experiments in a compound channel comprising a smooth main channel and smooth or rough floodplains.

For the study, floodplains has been roughened in four different ways using metal meshes. The metal meshes had a width of 35.5 cm with a height of 14.5 cm 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. Most experimental efforts have been focused on homogeneous roughness (smooth) compound channels restricted to low width ratio (width of compound channel / width of main channel base).Therefore, the present study is intended 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.

II. 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.

2.1 Model setup

The model setup consists of different steps, such as creation of the geometry setup of channel model, defining and designing the grid points using mesh operator, schematic diagram of the channel and measurement locations.

A. Geometry

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

Volume: 5, Issue: 07, [July– 2018] ISSN (Online):2349–9745 ; ISSN (Print):2393-8161

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.

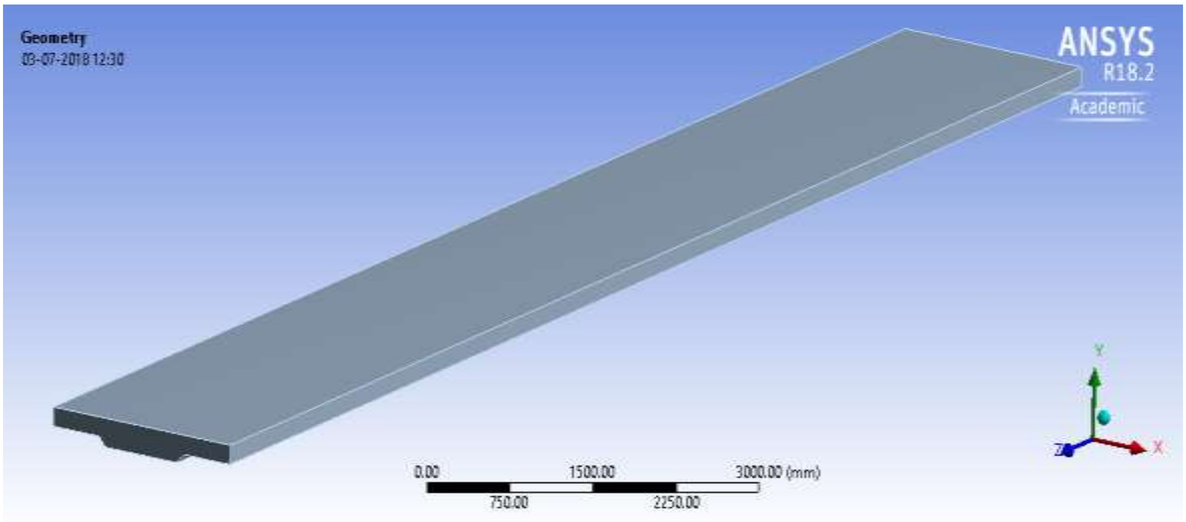

Figure 1.Geometry Of Channel

B. 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 channel is shown in the Figure 2

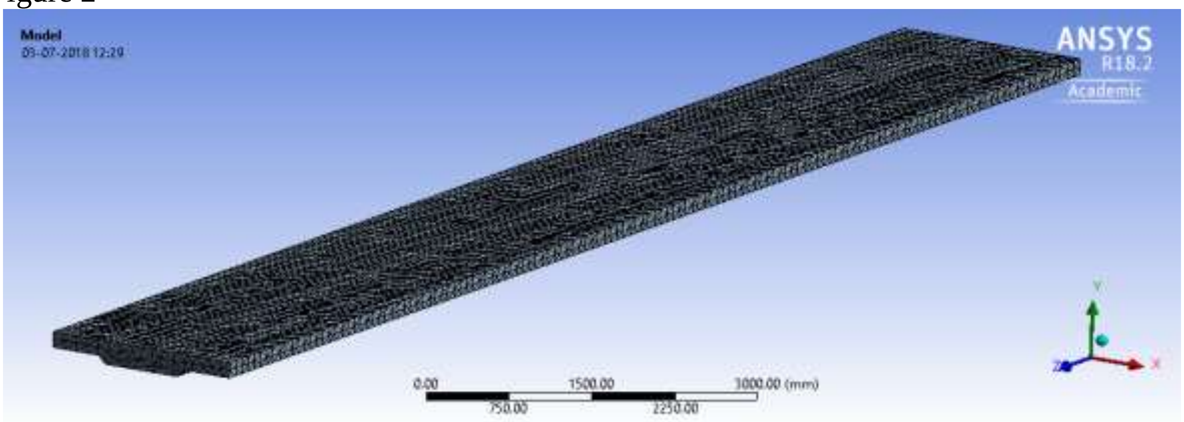

Figure 2. Meshing Of Channel

Volume: 5, Issue: 07, [July– 2018] **ISSN (Online):2349–9745 ; ISSN (Print):2393-8161**

C. 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.

D. Boundary conditions

The precision of boundary conditions on the solution domain plays a capital role for the accuracy of the results. The boundary conditions to be specified consist of flow inlet and outlet boundaries, which have to be defined with the flow properties such as turbulence parameters, velocity and pressure.”Walls and internal faces which have a direct interaction with the flow have been defined as well.

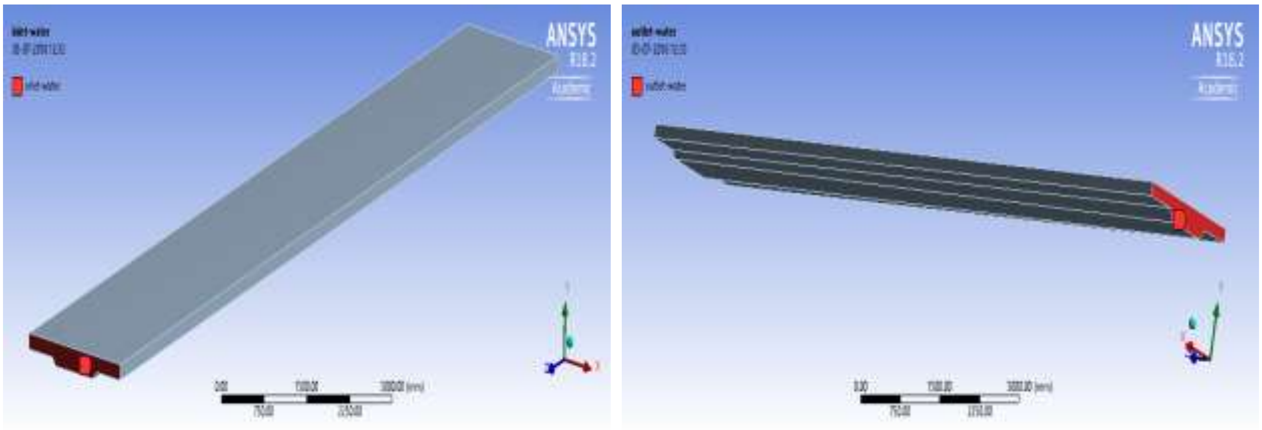

Figure 3. Water Inlet and Outlet of the Channel

Inlet boundary: The inlet section is at the upstream of the open channel and consists of the inlet of water at the bottom and the inlet of air at the top. For water inlet, a “velocity-inlet” boundary condition was selected as it was the best option that produced a stable flow in the solution domain. For discharge tested in physical modelling, the input velocity was calculated and set uniformly at the inlet. The air boundaries were defined as an inlet pressure with the atmospheric pressure conditions.

Outlet boundary: The outlet of the domain at the downstream part was specified with an outlet pressure so that water and air can flow out freely. To ensure atmospheric conditions, the air phase was allowed to flow back into the model.

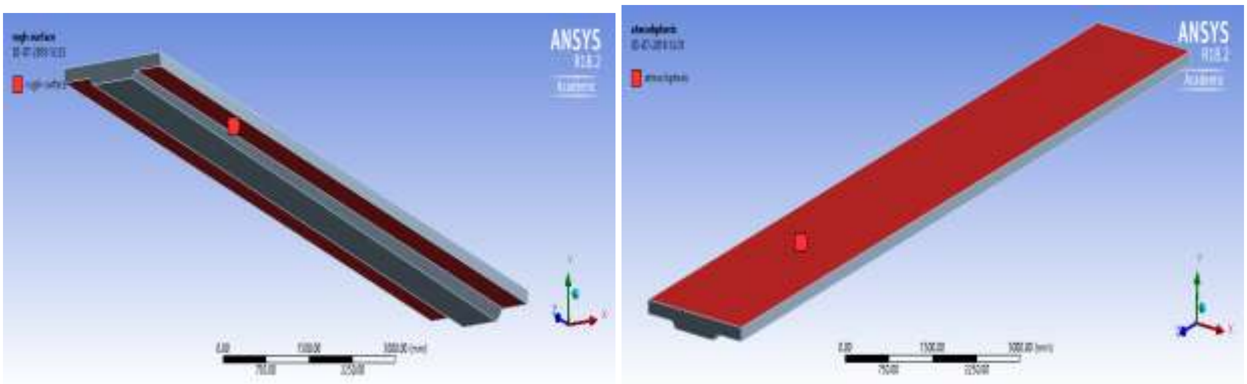

Figure 4. Rough surface and Free surface of the channel

Walls: Wall boundary conditions (walls and channel bed) were specified to simplify the operational conditions with no-slip conditions and to be in stationary conditions at all times.

III. RESULTS

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.

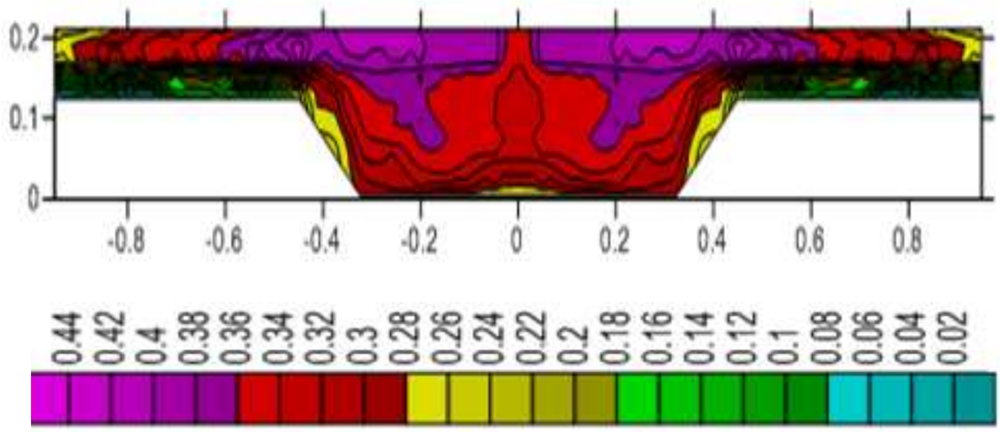

Figure 5. Velocity Contour Using Experimental Data Conducted By K.K Khatua[4]

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.

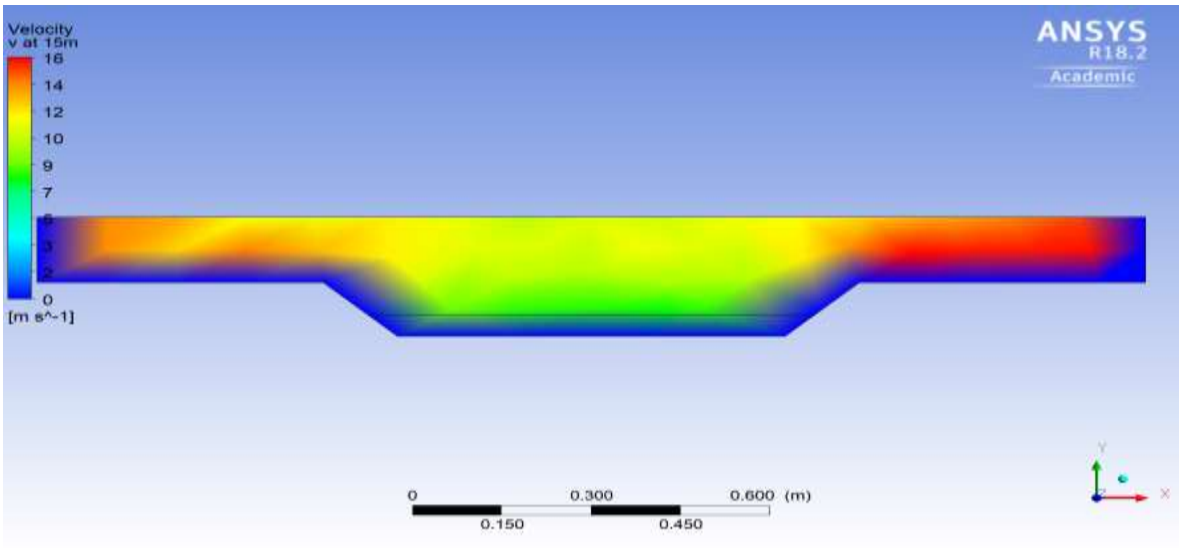

Figure 6. Velocity contour using Ansys

Volume: 5, Issue: 07, [July– 2018] ISSN (Online):2349–9745 ; ISSN (Print):2393-8161

3.1 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, % *Sfp* and % *Smc* are usually power or exponential functions of % *Afp* and % *Amc* 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.

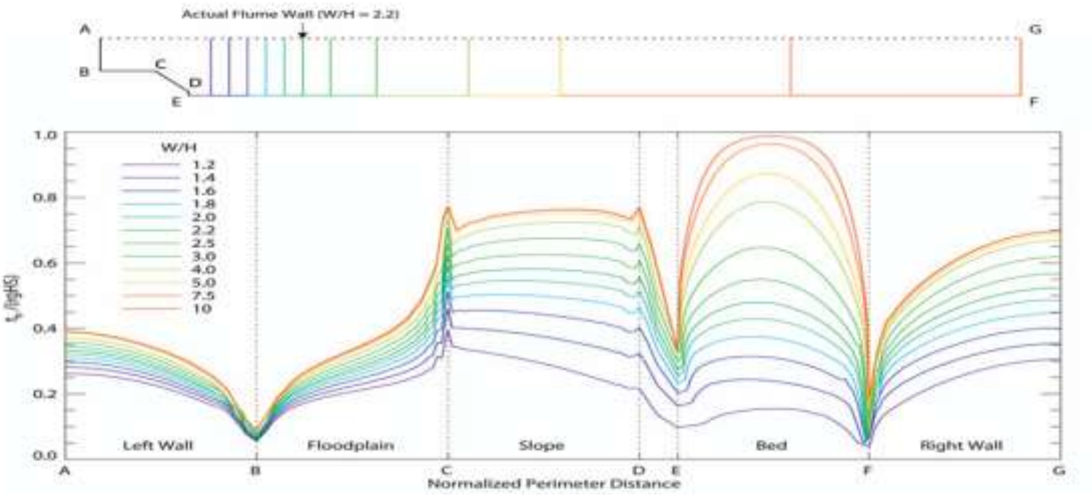

Figure 7. Color Calculated boundary shear1stress

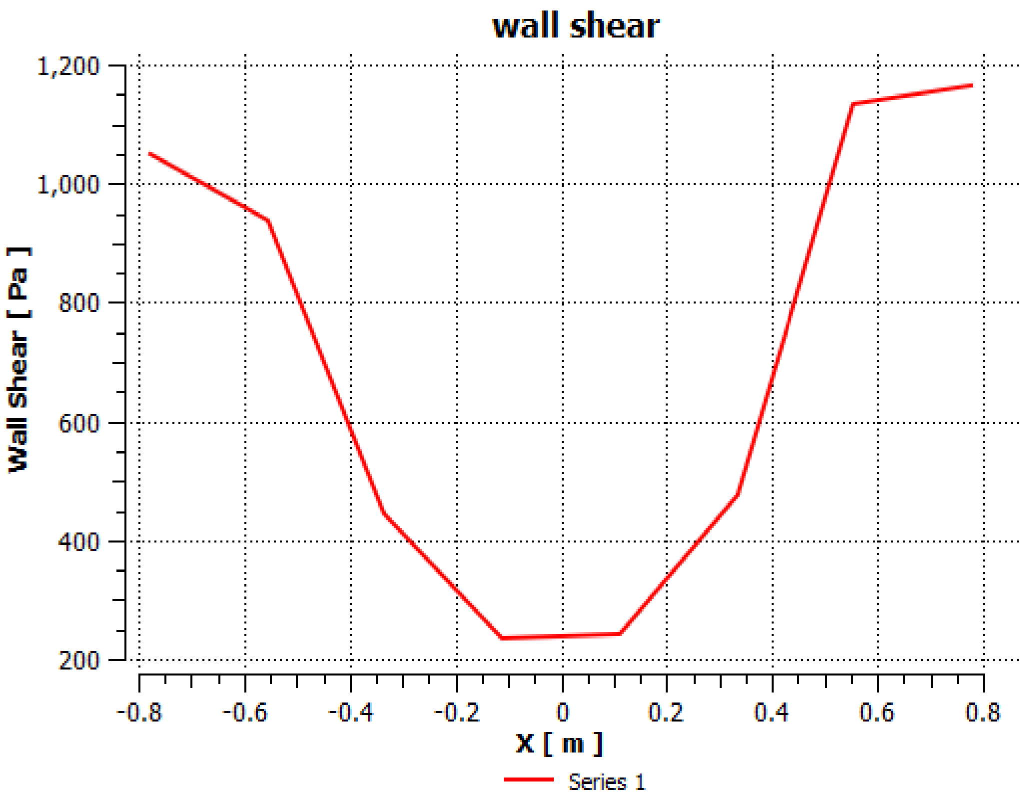

Figure 8. Wall shear calculated by Ansys

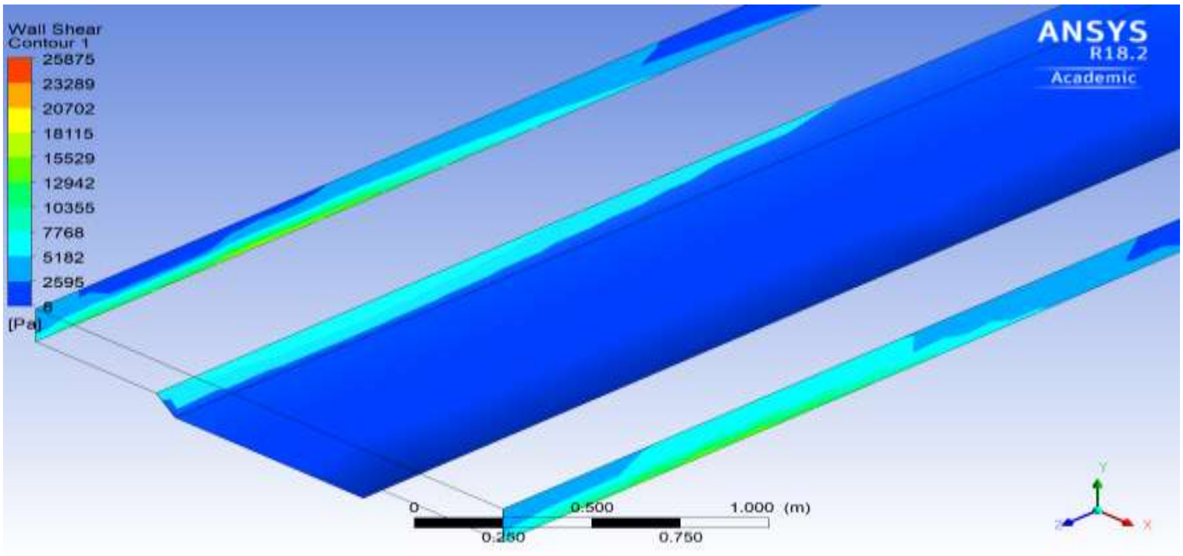

Figure 9. Wall Shear Contour

IV. 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).

Studies performed using Ansys 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. 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.

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.

Volume: 5, Issue: 07, [July– 2018] **ISSN (Online):2349–9745 ; ISSN (Print):2393-8161**

[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] Hin and Bessaih (2004). "Hydraulic radius for evaluating resistance induced by simulated emergent vegetation in open-channel flows." J. Hydraul. Eng.,137(9), 995–1004.
[9] Seckin (2004). "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.

