IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
e-ISSN: 2278-1684, p-ISSN: 2320-334X.
PP 01-06
www.iosrjournals.org

Longitudinal Velocity Distribution Modeling of a Highly Sinuous Meandering Channel using CFD

Sumit Kumar Jena[1], Kishanjit K. Khatua[2], Arpan Pradhan[3]
(Department of Civil engineering, Nit Rourkela, India)

Abstract : *Every day we people realize how important rivers are to people. These are expedient in numerous ways, often used to generate electricity, providing food, and for domestic purposes among others. So the study of river is authoritative. Previous research show that in defiance of numerous researches has on various aspects of velocity distribution in curved Meander Rivers, no systematic effort has been made to study the variation of velocity along a meander path. This detailed investigation of velocity distribution along the depth and width of a channel is also essential in many hydraulic engineering studies involving bank protection, sediment transport, conveyance, water intakes and geomorphologic investigation. So here , variation of velocity profile along the width and depth of the channel has been methodically analyzed at different cross-sections (3 sections) along a meander path of a sinuous channel of 120° cross-over angle experimentally and comparing the results of the same using ansys fluent as a result developing a simulation model for the same.*

Keywords: *Conveyance, Meander path, Simulation model, Velocity distribution*

I. Introduction

The study of Flow in compound meandering channel is quite equivocal for instinctive flow systems such as rivers. A meander in general is a bend in a sinuous water course or river. Meander forms when moving water in a stream erodes the outer banks and widens its valley, and the inner part of the river has less energy and deposits silt. Flow characteristics in channel bends are much more perplexed than those in straight channels. Not a single natural river possesses a straight geometry. Almost all natural rivers meander. In fact Straight River reaches longer than 10 to 12 times the channel widths are nonexistence in nature. Again Flow characteristics in channel bends are much more complicated than those in straight channels. Some researchers have carried out panoptic studies on flow characteristics in channels with different bend angels by using the experimental and numerical models. Shino [1] developed turbulence models and studied the behavior of secondary flows and centrifugal forces for straight and meandering channels. Blanckaert and Graf [2] investigated channel bed level changes at a 120º sharp bend with a movable bed using an experimental setup. They reported a minor secondary rotating flow cell at the outer wall of the bend. Booij [3] and Veblen et al. [4] modelled the flow pattern at a mildly-curved 180º bend and assessed the secondary flow structure using large eddy simulation (LES).The knowledge of velocity distribution helps to know the velocity magnitude at each point across essential in many hydraulic engineering studies involving bank protection, sediment transport, conveyance water intakes and geomorphologic investigation. In straight channel velocity distribution varies with different width-depth ratio whereas in meandering channel velocity distribution varies with aspect ratio, sinuosity, meandering making the flow more complex to analyze. In this paper, the velocity profile is studied throughout a meander path of a highly sinuous meandering channel of sinuosity 4.11. The velocity profile data are studied to find the flow pattern and movement of the local maximum velocity at each section along the meander path. The study is carried out on a 120° meandering path from the cross-over to the corresponding bend apex. Also here an effort has been made to investigate the velocity profiles for three different sections of the same compound meandering channel by using a computational fluid dynamics (CFD) modelling. The CFD model developed for a real open-channel is first validated by comparing the velocity profile obtained by the numerical simulation with the actual measurement carried out by experimentation in the same channel using pilot tube. Earlier a few experiments were carried out related to this field and various methods were investigated the suppress flow separation .To investigate complex flow characteristics of compound open channel expansion some recent flow measurements techniques and digital technology like Laser Doppler Anemometry (LDA) are created. Now a dayscomputational or numerical method is adopted for this type of case study. By using large eddy simulation (LES) method the flow pattern at a curved channel [4] and a curved 180º bend channel [5] are modeled and assessed the secondary flow structure as this model solves spatially-averaged Navier-Stokes equation. Large eddies are directly resolves, but the eddies smaller than mesh are modelled. Salvetti et al. (1997) has conducted LES simulation at a relatively large Reynolds number for producing results of bed shear, secondary motion and vorticity well comparable to experimental results. Cater and Williams (2008) reported a detailed Large Eddy

Simulation of turbulent flow in a long compound open channel with one floodplain. The Reynolds number is approximately 42,000 and the free surface was treated as fully deformable. The results are in agreement with experimental measurements and support the use of high spatial resolution and a large box length in contrast with a previous simulation of the same geometry. A secondary flow is identified at the internal corner that persists and increases the bed stress on the floodplain. However, as the LES model captures better the free surface and velocity variation with time. So here numerical simulation model is developed using Large Eddy Simulation model. Paper.

II. Methodology

2.1 EXPERIMENTAL SETUP

For carrying out research in meandering channels, experimental setup was built in Fluid Mechanics and Hydraulics Laboratory of NIT Rourkela by department of science and technology.

Table.1

Sl. No.	**Description**	
1	Channel Type	Highly Meandering Compound Channel
2	Flume size	4.0m x 0.5m x 15m
3	Geometry of channel section	Trapezoidal (Side Slope 1:1)
4	Nature of surface bed	Smooth and Rigid Bed
5	Channel Width	33cm at bottom and 46cm at top
6	Bank full depth of the channel	6.5cm
7	Bed slope of channel	0.00165
8	sinuosity	4.11

All observations are recorded along a meandering path from one bend apex to the next corresponding bend apex.A section at crossover perpendicular to both the inner and outer curves of the meandering channel is drawn and extended up to the bend apex line. An angle of 120^0 is formed for both the curves. These curves are divided into 6 equal sections of 20^0 each. The sections A to G including one section D are considered for measurement of velocities. Channel sections along the width i.e. perpendicular to both the sides are drawn at these points. The sections A through G are considered for measurement of the velocity profiles.Measurements are taken from left edge to the right edge of the main channel in the direction of flow. The lateral spacing of the grid points has been taken as 4cm on either side of the centerline.

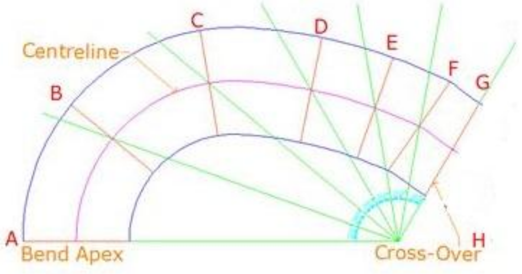


Fig.1:channel section from one bend apex A to cross over G

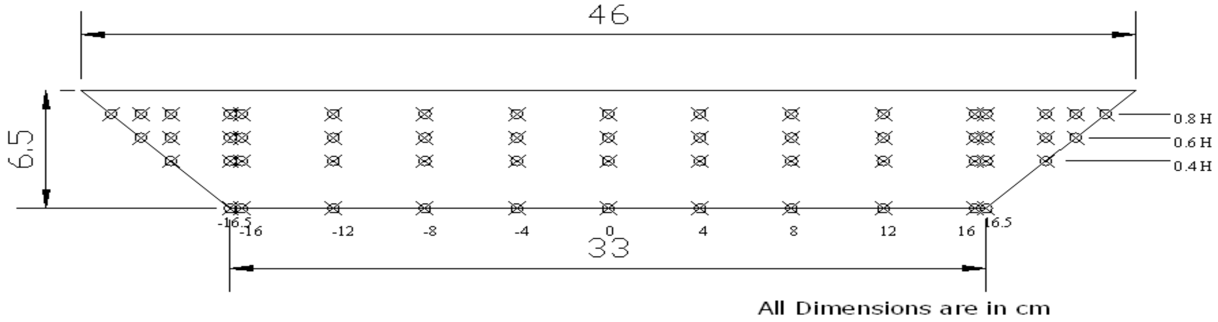


Fig.2:cross section of channel

Fig.3:Channel photograph

2.2Description of Numerical Model Parameters

Numerical codes used in this study (Fluent) are based on the full three-dimensional form of Navier-Stokes equations, and it uses a finite volume method (FVM). Fluent can use both structured and unstructured grids. For the solution that requires transient simulation, such as free-surface modeling e.g., VOF [22] model and the large Eddy Simulation (LES) turbulence modeling, the governing equations are discretized in both space and time. 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 [23]. PISO is a non-iterative solution method to calculate the transient problem, which converges faster [23]. 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.

2.3Governing Equations

The governing equations for the present study are the equations of conservation of mass, momentum and energy. The equation used for conservation of mass (the continuity equation) may be written as:

$$\frac{\partial \rho}{\partial t} + \rho \frac{\partial U_i}{\partial x_i} = S_m \qquad (1)$$

The source term Sm contains the mass added through phase changes or user defined sources. In general, and in the simulations described here, the source term was equal to zero.

2.3Turbulence Modeling

Turbulent flows have wide range of length and time scales. The large scale motions are generally energetic than the small ones and their size makes them the most effective transporters. Large eddies depend highly on boundary conditions which determine the basic feature of flow. Large scale eddy helps in transfer of momentum and heat. This technique directly resolves the large eddies present in turbulent flows and models the smaller scale eddies. LES models are based on filtering the spectrum of turbulent eddies in the Navier–Stokes equations. The filter is related to grid size, and is usually chosen to be the actual grid size. Eddies smaller than the grid sizes are thus removed and must be modeled by a subgrid scale (SGS) model. Larger eddies are directly solved by the filtered transient Navier–Stokes equation

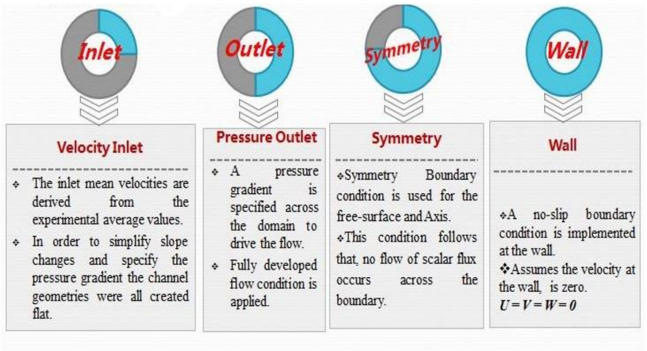
$$\frac{\partial \bar{u}_i}{\partial t} = \frac{\partial \bar{u}_i \bar{u}_j}{\partial x_j} = \frac{1}{\rho}\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_i}\left(\vartheta \frac{\partial \bar{u}_i}{\partial x_j}\right) - \frac{\partial \tau_{ij}}{\partial x_j} \qquad (2)$$

Whereτ_{ij} is the subgrid scale turbulent stress, which must be modeled. For our simulations, I chose the Smagorinsky–Lilly model which was the standard one available in Fluent. We also tried the WALE model on a test case with no difference in the results.

2.4Geometry Domain and Boundary condition

For a given computational domain, boundary conditions are imposed which can sometimes over specify or under-specify the results of experimentation. Usually, after imposing boundary conditions in a non-physical domain, it's may lead to failure of the solution.The channel analysed here allows the values on the inlet and outlet boundaries to coincide and a pressure gradient was further specified across the domain to drive the flow. A mean velocity was specified over the whole inlet plane to initialize the flow upon which velocity fluctuations were imposed. The inlet mean velocities were derived from the experimental average values. In order to specify the pressure gradient the channel geometries were all created with a bed slpoeof 0.00165and 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 asρg=(0,ρgcosθ,ρgsinθ)

Where θ= angle between bed surface to horizontal axis. Here, the z-component denotes the direction responsible for flow of water along the channel and the y-component is responsible for creating the hydrostatic pressure. From the simulation, y-component of the gravity vector (-ρgsinθ) is found to be responsible for the convergence problem of the solver. 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.



Velocity Inlet
- The inlet mean velocities are derived from the experimental average values.
- In order to simplify slope changes and specify the pressure gradient the channel geometries were all created flat.

Pressure Outlet
- A pressure gradient is specified across the domain to drive the flow.
- Fully developed flow condition is applied.

Symmetry
- Symmetry Boundary condition is used for the free-surface and Axis.
- This condition follows that, no flow of scalar flux occurs across the boundary.

Wall
- A no-slip boundary condition is implemented at the wall.
- Assumes the velocity at the wall, is zero.

$U = V = W = 0$

2.5Grid Generation

It must be noted that the grid-independent results are obtained and the running time is low. The grid structure must be fine enough, especially near the wall boundaries (in order to consider the viscous flow), in the bend (the rapid changes area) and free surface. In this numerical model, various computational trials are conducted with different number of grid cells in x, y and z directions. It was concluded that the results are almost independent from the grid size and running time is optimal. The fluid flow governing equations (momentum equation, continuity equation) are solved based on the discretization of domain using the cartesian co-ordinate system. The CFD computations need a spatial discretization scheme and time marching scheme. This method is suitable with respect to both structured and unstructured mesh.

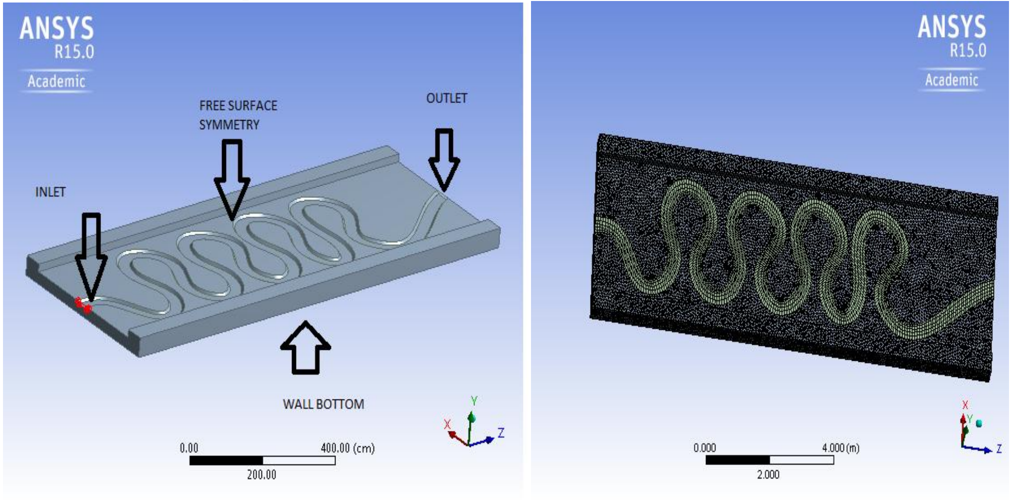


Fig.4Design of the channel and meshing

III. RESULTS

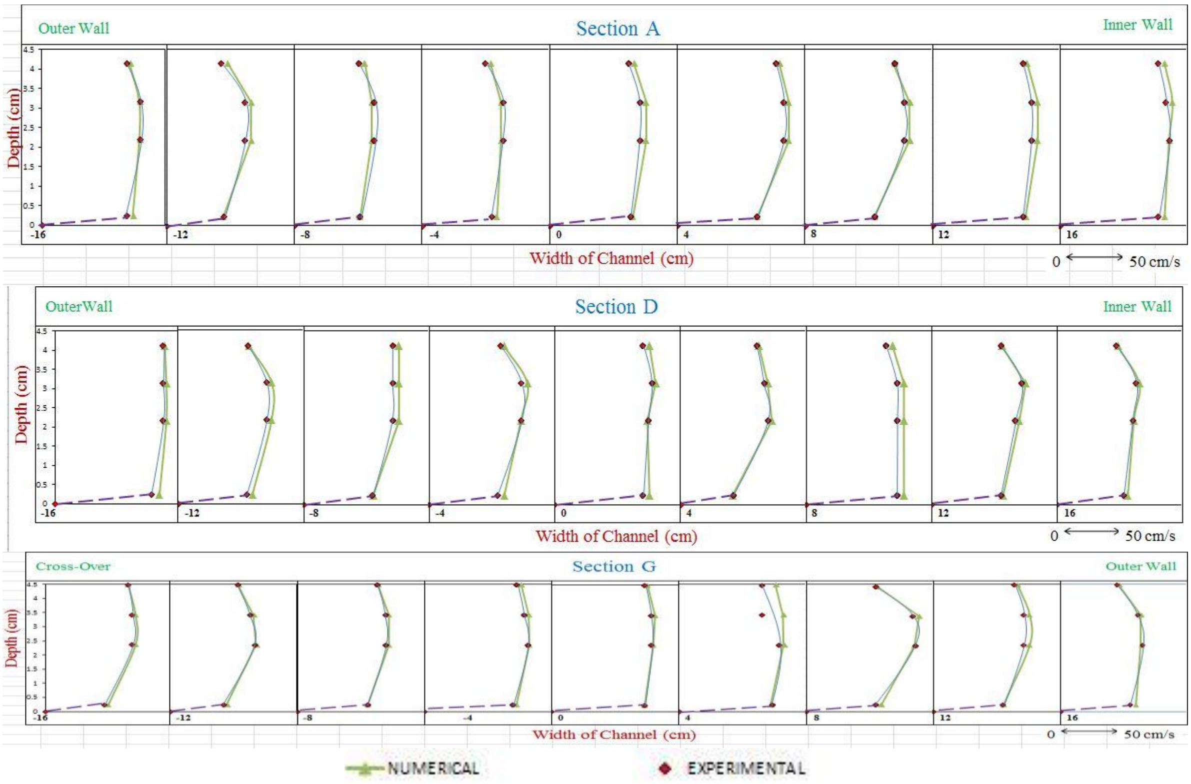


Fig.5: distribution of longitudinal velocity profiles at different sections.

IV. CONCLUSION

- For a given discharge the velocity profile data are taken throughout the meander path across the channel. The variations of velocities from point to point along the meander path of the highly sinuous channel are plotted.
- Longitudinal velocity profiles for each section at 4cm interval on either side of the centre-line are analysed. The sections indicated are the bend apex sections A and M, the cross-over section G.
- Longitudinal velocity plots given later clearly indicate that from sections A and G profiles tend to remain higher towards the inner wall.
- At the cross-over it is observed that the velocity profiles are somewhat uniform with its maximum velocity occurring close to the centre of the channel section. This observation is usually seen to be similar in straight channels. Hence the channel follows nearly a steady and uniform flow at the cross-over reach, where the meandering channel changes its sign of curvature and is assumed to behave nearly as a straight channel.
- By analysing the profiles, it is clearly observed that the velocity remains higher on the inner wall than at the outer wall while moving over a curve.
- Accurate prediction of velocity distribution in channels is very important for flood studies and estimation of stage discharge curve in natural channels. The present research attempts to analyse the flow profile characteristics of water on a meander path, where the path continues as a meandering channel on both the ends.
- The numerical results we found are somewhat higher than the experimental value.

REFERENCES

Journal Papers:

[1] Toebes, G.H., and Sooky, A.A.(1967). “Hydraulics of Meandering Rivers with Floodplains.”Journal of the waterways and Harbor Division, Proceedings of ASCE, Vol.93, No.WW2, May, pp. 213-236.

[2] Coleman, N. L. (1981)Velocity profiles with suspended sediment. J. Hydro. Res, 19(3), 211-229.

[3] Chang, H.H. (1983). "Energy Expenditure in Curved Open Channels." Journal of Hydraulics Engineering, ASCE, Vol.109, No.7, pp. 1012-1022.

[4] Johannesson, H., and Parker. (1989). “Velocity Distribution in Meandering Rivers.” Hydraulic Engineering, ASCE, Vol. 115, No. 8, pp. 1019-1039.

[5] James, C. S. (1994). ‘‘Evaluation of methods for predicting bend loss in meandering channels.’’ Journal of Hydraulics Engineering. ASCE, 120(2), 245–253.

[6] Cokljat D, Younis BA. Second-order closure study of open channel flows. Journal of Hydraulic Engineering (ASCE) 1995; 121:94 – 107.

[7] Shiono, K., Muto, Y., Knight, D.W. & Hyde, A.F.L.(1999). “Energy Losses due to Secondary Flow and Turbulence in Meandering Channels with Overbank Flow.” Journal of Hydraulic Research, IAHR, Vol. 37, No. 5, pp. 641-664

[8] Afzal, N., Seena A., Bushra A.2007. Power law velocity profile in fully developed turbulent pipe and channel flows. J. Hydro. Energy, Vol. 133, No. 9, 1080-1086.

[9] 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).

[10] Jing, H., Guo, Y., Li, C., & Zhang, J. (2009). Three dimensional numerical simulation of compound meandering open channel flow by the Reynolds stress model. InternationalJournal for Numerical Methods in Fluids, 59, 927-943.

[11] Jing, H., Guo, Y., Li, C., & Zhang, J. (2009). Three dimensional numerical simulation of compound meandering open channel flow by the Reynolds stress model. InternationalJournal for Numerical Methods in Fluids, 59, 927-943.

[12] Saine S. Dash, K.K.Khatua, P.KMohanty (2013), “Factors influencing the prediction of resistance in a meandering channel”, International Journal of Scientific & Engineering Research ,Volume 4, Issue 5, May-2013.

Thesis:

[1] Mohanta.A (2014)” Flow Modeling of Non Prismatic compound channel By Using C .F .D”.Thesis Presented to the National Institute of Technology, Rourkela, in partial fulfillment of the requirements for the Master of technology

[2] Pradhan .A (2014)” Analysis Of Flow Along The Meander Path Of A Highly Sinuous rigid Channel” Thesis Presented to the National Institute of Technology, Rourkela, in partial fulfillment of the requirements for the Master of technology.