International Journal of Water & Hydro Constructions – IJWHC
Volume 1 : Issue 3 *[ISSN 2374 –1600]*
Publication Date : 30 September, 2014

Numerical Simulation of 180˚ Meandering Bend of Barak River near Silchar City

(Study the flow structure by using of CFD)

[R. Karthik[1], U. Kumar[2] and K.M. Pandey[3]]

***Abstract*—This paper reports the three dimensional numerical simulation of 180˚ meandering bend for Barak river near Silchar city, Assam. The RNG k-ε turbulence model is used to predict the magnitude of velocity in the meandering bend. It explains the flow structure and velocity around the sharp bend that initiates the meandering process. Three dimensional CFD model were used to predict the velocity contour and velocity vectors at various sections at river bends and also at inlet and outlet sections. At the sharp curve of meandering bend at 50˚ to 70˚ the magnitude of velocity is higher in outer bank and helical flow are occurring due to which, year by year the concave bank of the sharp meandering of Barak River is getting eroded at the same time convex bank getting deposited. This also agrees with the resent image of satellite view of Barak River near Silchar town. This study will help the designer to understand the meandering at various angle of river bend to design proper river control structure.**

***Keywords*—Numerical Simulation, CFD, Barak River, 180˚ Meandering Bend.**

I. *Introduction*

Barak River one of the major river in south Assam and is highly meandering in nature. Silchar is one of the busy city in northeast India and nearly half of the city boundary area is covered by the meander loop of Barak River. In Barak River, the southern part of Assam with a length of about 134 km is showing highly sinuous pattern in the region, further the sinuosity is very high at around Silchar city [1].The major safety issue in these regions are threatening the household, agricultural lands and roads which are situated near the river for continues process of river bed migration and bank erosion.

R.KARTHIK, M.Tech Scholar
Department of Civil Engineering, NIT Silchar,
Assam, India.

U.KUMAR, Associate Professor,
Department of Civil Engineering, NIT Silchar,
Assam, India.

K.M.PANDEY, Professor,
Department of Mechanical Engineering, NIT Silchar,
Assam, India.

In the field of river engineering, the complex flow behaviour exits at sharp bends, so it is very important to understanding the flow structure especially near the banks. It is very difficult, time consuming and costly to get all the details by field observation and lab experiment, as the river is highly turbulent in nature. Numerical simulation can be better option to get all the details which is not possible to get experimentally. The computational fluid dynamic CFD is branch of fluid dynamics and is the prime techniques used to predicting the flow behaviour and migration of bed and banks in the field of river engineering. Most of the meander bends are at sharp curve, where there is a large flow separation zone along the inner wall of the bend and a counter-rotating secondary flow cell structure. A large lateral free-surface slope in the bend has also been observed [2, 3, 4].

II. *Literature Review*

Rivers are running in all times, several researchers have classified the rivers based on channel patterns and process. Alluvial rivers are classified like straight, meandering and braided based on relationship between slope and discharge by Leopold and Wolman (1994) [5]. Practically it is very difficult to find the straight alluvial channel for considerable length because meandering tends to occur at every regular interval based on the intensity of flow rate. In the sharp bend the flow structure is encounter as one of the complex flow situation in real life. The outer bank of the river is called as concave and inner bend is convex. Longitudinal velocity is less in bed and very high in top surface of channel and in the bends the transverse direction top layer of water will flow towards outer bank and bottom layer of water flows toward inner bank of river, due to difference in centrifugal force at bends, this action is called as flow separation and it will generate secondary current in the inner bank. The secondary current initiates the spiral motion of flow in bend portion and this flow is called as a helical flow. In the linear theory of Ikeda et al. (1981) the channel migration rate is assumed to be proportional to the near-bank water flow velocities [6]. Velocity at meander bend will be maximum in outer bank of river and minimum in the inner bank of river, so that concave bend, erosion will occur and at the same time in convex bend, deposition will occur.

In river bend, additional form of energy dissipation will occur as compared to straight channel because of the energy loss due to changing the flow direction [5]. The helical flow plays a vital role in river channel migration, including erosion and deposition around meander bend of river. For controlling and reducing this issue it is needed to provide some control structure around river bend like bendway weirs, rock vanes, spur/dikes/ groynes or jetties. Investigation of flow field is done by either experimental or numerical simulation but experimental measurements are subject to some human errors and are difficult to get all flow measurement details and making the model very costly and time consumable as compared to numerical simulation. CFD working based on the governing equation like integral and partial derivatives with linear algebraic equation to solve and getting flow field values at discretization point in space and time. Basic form of CFD governing equations are Finite Difference Method (FDM), Finite Element Method (FEM) and Finite Volume Methods (FVM). CFD models that are used to predict the turbulence flow behaviour of rivers are Direct Numerical Simulation (DNS), Reynolds Averaged Numerical Simulation (RANS) and Large Eddy Simulation (LES). Nowadays many types of commercial CFD codes are available for example CFX, FLUENT, PHOENICS, STAR CD, FLOW 3D, CFD-ACE, ICEM CFD and many more.

III. Pre Review of previous Numerical Simulation

In recent years researchers successfully modelled several 2D and 3D numerical model for predict the flow behaviour around the meandering bend of river including helical flow, secondary current and migration process of channel [7, 8]. This is particularly true for sharp bend flows with a high ratio of width to the mean radius in which a flow separation zone is present along the inner wall [9, 10]. Renormalization group (RNG) k-ε models are more accurate and reliable for a wider class of flows than the standard k-ε model [11], 3D numerical model successfully modelled to investigate the flow behaviour around meandering bend and mechanisms of flow separation zone, rotating flow [3, 4].

IV. Methodology and model description

This paper is based on the CFD study of the flow field and velocity magnitude of Barak River, 180˚ meander bend near Silchar city. The inner perimeter of flow field is around 4766 m and the channel section is approximately trapezoidal and average top and bottom width is 180 m and 100 m respectively with an average depth of 5 m. Barak River model was made by using CATIA V6 assuming scale ratio as 1 in 100. Radius is not constant from 0 to 180 degree and is changing from point to point as observed at the present location in Barak River meander bend near Silchar. Meshing was done by ICEM CFD, total elements are 502768, and total nodes are 88857, number of element as part wise bed is 13506, inner wall is 9930, outer wall is 10820, inlet is 125, outlet is 138, top surface is 23975. The meshing type was all triangular and patch independent, number of cell in the section is 441441and faces of the section is 912129. The mesh type is unstructured quadrilateral and tetra mixed.

Three dimensional numerical simulation of 180˚ meandering bend is carried with help of ANSYS FLUENT software which is one of the well known commercial CFD codes. In multiphase flow, volume of fluid (VOF) is chosen in the VOF model, considering as open channel flow. For turbulence flow generally k-$\mathcal{E}$ models are used, so in this problem renormalization group (RNG) k-$\mathcal{E}$ model has been used. In multiphase flow number of phase is 2, the primary phase is air, and secondary phase is water and the material properties were taken from fluent data base. In the (RNG) k-$\mathcal{E}$ model, walls are considering as a concrete wall with enhance wall treatment and pressure gradient effects. Boundary condition of the turbulence model is considering the inlet as velocity inlet and outlet is pressure outlet. The average value of velocity in flow direction is 0.85 m/s and pressure value is considering an atmospheric pressure, the value of k and $\mathcal{E}$ is 0.218 and 0.0478 respectively. Wall is considering stationary with no slip condition. In the top surface the pressure is atmospheric pressure and the density is lightest. The pressure-velocity coupling is done by PISO and pressure is PRESTO, volume fraction is modified HRIC, momentum is second order upwind, turbulent and others parameters are first order upwind, iteration is time step method, time step size is 0.1 and maximum iteration per time step is 20.

V. Result and Discussion

This paper reports the magnitude and direction of velocity at different channel section, as the velocity is playing a vital role in meandering process. Fig.1 shows the velocity contour of 180˚ meandering bend of Barak River. It is quite clear from this figure that the velocity contour are higher in sharp bend especially in outer bank when compared to inner bank. Velocity vector are shown in Fig.2 for 180˚ meandering bend of Barak River. This figure indicates very high velocity vector at bend and it is breaching the outer bank. It is observed that inner bank is having less velocity magnitude and secondary currents also occur in convex bank, which will lead to helical flows. Velocity contour at 50˚, 60˚ and 70˚ at sharp bend section is showing in Fig.3, Fig.4 and Fig.5 respectively which clearly indicate higher velocities at outer bank.

Fig.4 shows clearly demonstrate the higher velocity in concave bank and lesser velocity in convex bank as compared to outer bank of meandering bend. Fig.5 is located in sharp meandering bend. The colour indicates the magnitude of velocity, so in the sharp curve bend concave bank getting eroded because the velocity is higher and convex bank getting deposited because velocity is lesser. Compared to other images, the Fig.6 are having less intensity of velocity in outer bank because this is velocity contour at 90˚ channel section which is situated away from sharp bend. Fig 7 and 8 is shows the velocity vector near inlet and outlet, that clearly demonstrate the intensity of velocity is higher in centre and intensity of velocity is lesser in surrounding channel section and both section is locating on straight position. Velocity vectors at 50˚ and 60˚are shown in Fig 9 and 10 which clearly indicate the flow is rotating and resulting in helical flow, as it is at sharp curve of meandering bend. Fig. 11 is showing the satellite view of 180˚ meandering bend of Barak River near Silchar city. The highlighted portion indicates the area of inner banks getting depositions and outer bank getting eroded which agrees with the observed results at various sections.

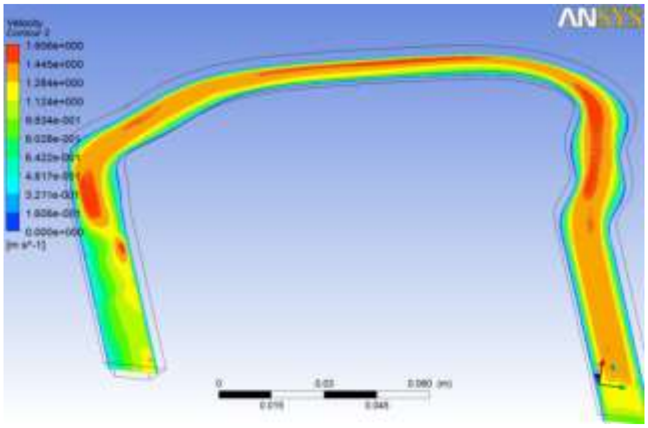


Figure 1. Velocity contour of 180˚ meandering bend

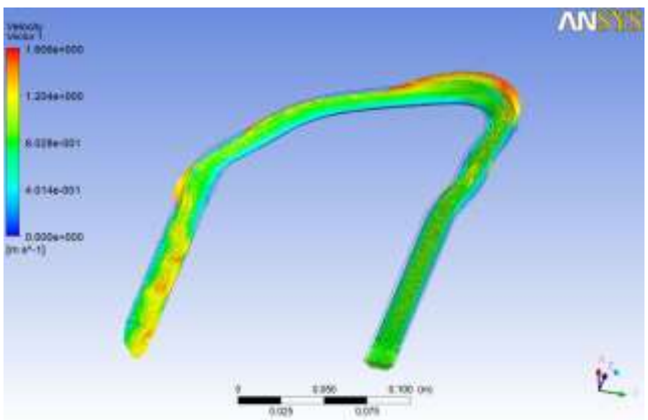


Figure 2. Velocity vector of 180˚ mender bend

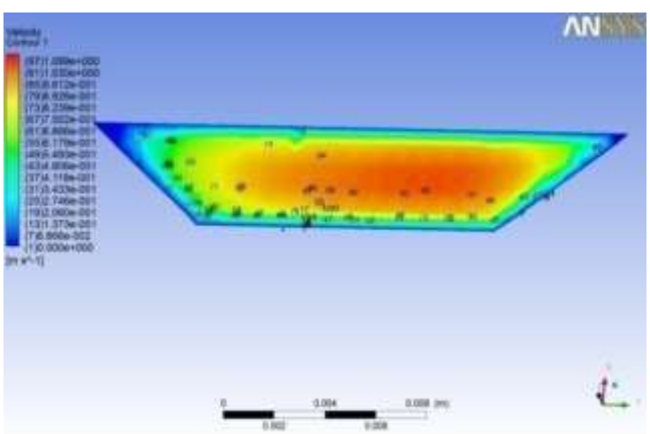


Figure 3. Velocity contour at 50˚ sharp bend

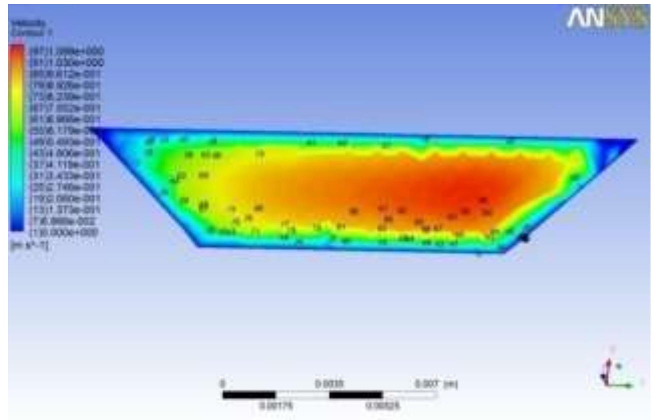


Figure 4. Velocity contour at 60˚ sharp bend section

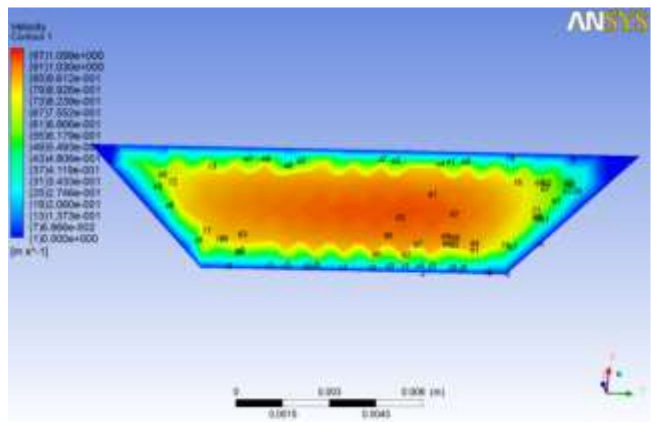


Figure 5. Velocity contour at 70˚ sharp bend section

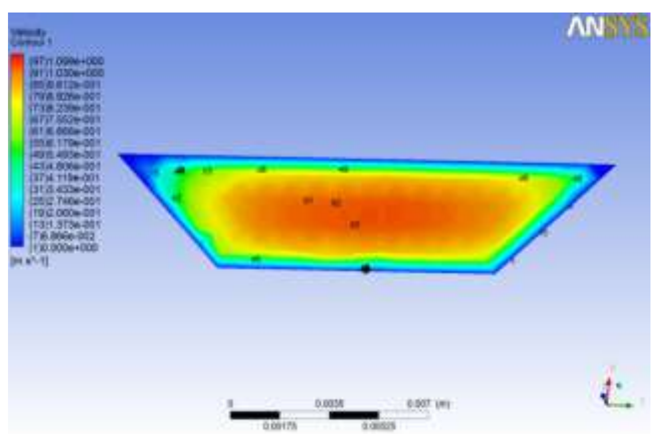


Figure 6. Velocity contour at 90˚channel section

International Journal of Water & Hydro Constructions – IJWHC
Volume 1 : Issue 3 *[ISSN 2374 –1600]*
Publication Date : 30 September, 2014

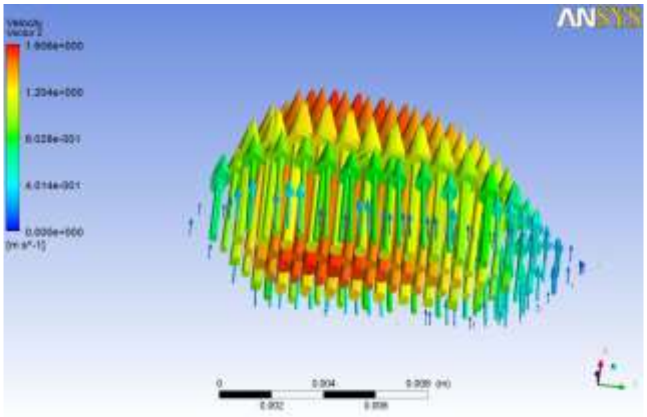


Figure 7. Velocity vector at inlet section

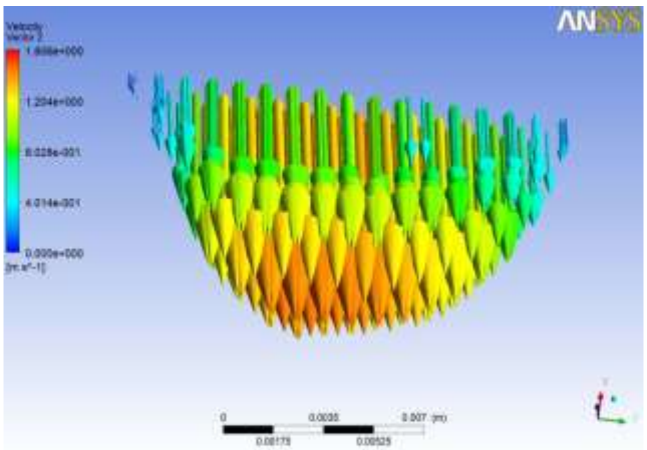


Figure 8. Velocity vector at outlet section

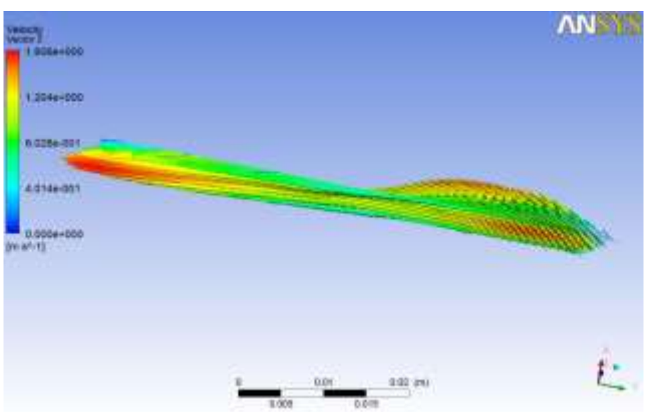


Figure 9. Velocity vector at 50˚ sharp bend section

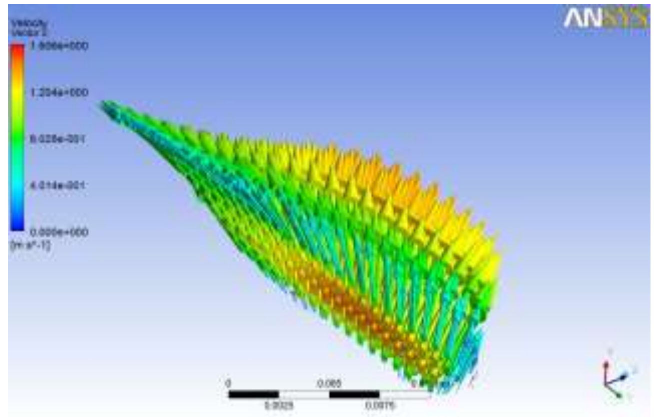


Figure 10. Velocity vector at 60˚ sharp bend section

Figure 11. Satellite view of Barak River photo. [Courtesy: Google maps]

VI. ***Conclusion***

The present study shows the velocity field of three dimensional CFD model of Barak River. Some contour and vector images of velocity magnitude at different angle of channel section are compared. At the sharp curve of meandering bend at 50˚ to 70˚ the magnitude of velocity is higher in outer bank and helical flow are occurring due to which, year by year the concave bank of the sharp meandering of Barak River is getting eroded at the same time convex bank getting deposited. This also agrees with the resent image of satellite view of Barak River near Silchar town. The present research work will help to better understanding the flow structure particularly the velocity magnitude for controlling this meandering process by providing control structures to control the velocity and regulate the flow field. This study will help the designer for designing the proper river control structure.

References

[1] P. Das, 'Study of Barak River meander and associated hazard around silchar town, Assam, using remote sensing and GIS', Journal of earth science India, vol. 5(2) pp-51-59, April 2012.

[2] K. Blanckaert, 'Hydrodynamic processes in sharp meander bends and their morphological implications', J. Geophys. Res.,116, F01003, 2011.

[3] G. Constantinescu, M. Koken and J. Zeng, 'The structure of turbulent flow in an open channel bend of strong curvature with deformed bed: Insight provided by detached eddy simulation', J. Water Resour. Res., 47,W05515, 2011.

[4] S. Han, P. M. Biron and A. S Ramamurthy, 'Three dimensional modelling of flow in open-channel sharp bends with vanes' J. Hydraul. Res., 49, pp. 64–72, 2011.

[5] L. B Leopold, R. A. Bagnold, M. G. Wolman and L. M. Brush, 'Flow resistance in sinuous or irregular channels', U.S. Geol. Survey Prof. Paper 282-D, pp. 111-134, 1960.

[6] S. Ikeda, G. Parker and K Sawai, 'Bend theory of river meanders, linear development', Journal of Fluid Mechanics, 112, pp. 363–377, 1981.

[7] Y. Jia, S. H. Scott, Y. Xu, S. Huang and S. S. Y. Wang, 'Three-dimensional numerical simulation and analysis of flows around a submerged weir in a channel bendway' J. Hydraul. Eng., 131, pp. 682–693, 2005.

[8] J. D. Abad, B. L. Rhoads, I. Guneralp, and M. H. García, 'Flow structure at different stages in a meander-bend with bendway weirs', J. Hydraul. Eng., 134, pp. 1052–1063, 2008.

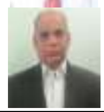
[9] J. Zeng, G. Constantinescu, K. Blanckaert and L. Weber, 'Flow and bathymetry in sharp open-channel bends: Experiments and predictions', J. Water Resour. Res., 44, W09401, 2008.

[10] W. van Balen, W. S. J. Uijttewaal and K. Blanckaert, 'Large-eddy simulation of a mildly curved open-channel flow', J. Fluid Mech., 630, pp. 413–442, 2009.

[11] V. Yakhot, S. A. Orszag, S. Thangam, T. B. Gatski and C. G. Speziale, 'Development of turbulence models for shear flows by a double expansion technique', J. Phys. Fluids A, 4, pp. 1510–1520, 1992.

About Author (s):





The colour indicates the magnitude of velocity, so in the sharp curve bend concave bank getting eroded because the velocity is higher and convex bank getting deposited because velocity is lesser.