Research Article Impact Factor: 4.226 ISSN: 2319-507X
Kiran, IJPRET, 2015; Volume 3 (9): 9-17 IJPRET

INTERNATIONAL JOURNAL OF PURE AND APPLIED RESEARCH IN ENGINEERING AND TECHNOLOGY

A PATH FOR HORIZING YOUR INNOVATIVE WORK

THREE DIMENSIONAL NUMERICAL MODELINGOF BARAK RIVERUSING REYNOLDS STRESS MODEL

KIRAN[1], UPENDRA KUMAR[2]

1. M. Tech Scholar, Water Resources Engineering, Department of Civil Engineering, National Institute of Technology, Silchar, Assam, INDIA.
2. Associate Professor, Department of Civil Engineering, National Institute of Technology, Silchar, Assam, INDIA.

Accepted Date: 05/03/2015; Published Date: 01/05/2015

Abstract: This paper presents a three dimensional numerical study of flow pattern through natural channel using computational fluid dynamics (CFD). A five kilometer stretch of the Barak River has been simulated using the Reynolds Time-Averaged Navier Stokes (RANS) equation, closed by the Reynolds Stress Model (RSM). This model accounts for multiple effects in complex flows, especially stress induced secondary flows. Finite volume method was used for numerical modeling and the free water surface was simulated using the volume of fluid (VOF) method. The velocity at different sections of the channel showed that the outer bank had higher velocities compared to the inner bank. The corresponding velocity contours obtained were compared with the velocity contours simulated using K-Epsilon turbulence model. The results of RSM seemed more appropriate and plausible than the two equation K-Epsilon turbulence model. Erosion was found to occur at the concave bank between and where higher velocities are prominent. Satellite image of the study area gives a more explicit concurrence with the simulated results using RSM model.

Keywords: Computational fluid dynamics (CFD), Reynolds time-averaged Navier Stokes (RANS) equation, Reynolds stress model (RSM), Finite volume method (FVM), secondary flow, Velocity

Corresponding Author: MR. KIRAN

Access Online On:

www.ijpret.com

How to Cite This Article:

Kiran, IJPRET, 2015; Volume 3 (9): 9-17

PAPER-QR CODE

INTRODUCTION

The Barak River, 134 kilometers long, located in southern Assam is one of the most highly meandering rivers in North East India. Across its floodplain, the river shows different spatial and temporal vulnerability. Barak River within its alluvial reaches of Assam exhibits evidence of lateral movement, whereas some of its tributaries show both vertical and lateral movement of the river has undergone substantial changes in its channel position at several places in Barak valley with strong northward shift towards west of Silchar (Das et al., 2007).

The river-bank erosion occurs due to the higher flow velocities at the produced during the flow at the outer bank. Erosion and accretion has led the banks to retreat and advance respectively at various sections of flow along the Barak River.

In channel bends the difference between centrifugal forces and the pressure gradient leads to the secondary flow and formation of helicoids at the outer bank. The helicoid generated by the horizontal shear forces of flow is greater than the centrifugal force as the flow occurs through the bend (H. Moravan, 2002).

A complete second-order closure model of turbulence was used to predict the behavior of fully developed turbulent flows in open channels of both simple and compound cross sections especially turbulence-driven secondary motions by D. Cokljat and B. A. Younis (1995).

Booij (2003) and Van Balen et al. (2009) assessed the secondary flow structure along a 180 degree bend using the Large Eddy Simulation (LES). Existence of secondary flow pattern was observed at the outer bank with created local scour.

Lu et al. (2004) applied the three dimensional numerical model to simulate flow along a 180 degree channel bend using the K-Epsilon turbulence model. Though the velocity fields showed good results it failed to predict the secondary current at the outer wall.

Ramamurthy et al. (2013) used various turbulence models and suggested that RSM turbulence model has better agreement with experimental results.

In this study the numerical model has been used to simulate the velocity of flow along the bend of the considered channel. Modeling the free surface velocity accurately is of prime importance as it helps us to understand the secondary current mechanism. Further the steady state velocity contours of the RSM model is compared with that of the previously simulated RNG K-Epsilon model.

MATERIAL & METHODS

Governing Equations

The numerical modeling involves the solution of the Navier-Stokes equations in finite volumes, which are based on the conservation of mass and momentum of the moving fluid. In channel flows which are turbulent the velocity, pressure and the stresses within the fluid are constantly fluctuating. The Navier-Stokes equations are time averaged which results in the Reynolds Averaged Navier Stokes equations can be written in the tensor form as, first, for the continuity equation and second for the conservation of momentum,

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho \overline{u_i}) = 0 \qquad (1)$$

$$\frac{\partial}{\partial t}(\rho \overline{u_i}) + \frac{\partial}{\partial x_j}(\rho \overline{u_i}\,\overline{u_j}) = \qquad (2)$$

$$-\frac{\partial \overline{p}}{\partial x_i}\frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i}\right)\right] + \frac{\partial}{\partial x_j}(-\rho\overline{\acute{u}_i \acute{u}_j})$$

Where ρ = density, $\overline{u_i}$= mean components of velocity, P = pressure, $\overline{\acute{u}_i}$= fluctuating part of velocity and μ = dynamic viscosity. The terms $-\rho\overline{\acute{u}_i\acute{u}_j}$ are referred to as Reynolds stresses and represent six additional unknowns in the RANS equations that should be modeled or resolved to allow for the solution of the above equations.

Reynolds Stress Model (RSM)

The poor representation of the individual Reynolds stresses in the two equation models are overcome by the 7 equation Reynolds stress model particularly in flows with complex strain fields and significant body forces. It also accounts for the directional effects of the Reynolds stress field. The equation of transport is given by

$$\frac{DR_{ij}}{Dt} = P_{ij} + D_{ij} - \varepsilon_{ij} + \Pi_{ij} + \Omega_{ij} \quad (3)$$

Where,

$R_{ij} = -\tau_{ij}/\rho = \overline{\acute{u}_i\acute{u}_j}$

$P_{ij} = -(R_{ik}\frac{\partial \overline{u_j}}{\partial x_k} + R_{jk}\frac{\partial \overline{u_i}}{\partial x_k})$

D_{ij}= Divergence $[\frac{\gamma_t}{\sigma_k}$ grad $(R_{ij})]$

$\varepsilon_{ij}=\frac{2}{3}\varepsilon\delta_{ij}$

$\Pi_{ij}= -C_1\frac{\varepsilon}{k}(R_{ij} - \frac{2}{3}k\delta_{ij}) - C_2(P_{ij} - \frac{2}{3}P\delta_{ij})$

$\Omega_{ij} = -2\omega_k (R_{jm}e_{ikm} + R_{im}e_{jkm})$

where, P_{ij} is the rate of production of R_{ij}, D_{ij} is the transport of R_{ij} by diffusion, ε_{ij} is the rate of dissipation of R_{ij}, Π_{ij} is the transport of R_{ij} due to turbulent pressure strain interactions, Ω_{ij} is the transport of R_{ij} due to rotation. $\Gamma_t = C_\mu \frac{k^2}{\varepsilon}$; $C_\mu = 0.09$; $\sigma_k = 1.0$. ε is the dissipation rate of turbulent kinetic energy, δ_{ij} is Kronecker delta. $C_1 = 1.8$, $C_2 = 0.6$. ω_k is the rotation vector with the alternating symbol.

Domain and Mesh Generation

A 180 degree bend of the Barak River was considered for the numerical analysis. The outer perimeter of flow field is around 5.3 kilometers and the inner perimeter was around 4.8 kilometers. The scaled three dimensional models were created using AutoCAD, Solid works and Ansys Design Modeler. Patch independent grids were generated using the Ansys ICEM CFD tool. The domain was meshed entirely using structured tetrahedral cells of uniform cell size. The cell size was kept at 1mm in all directions because it should be capable of capturing even the smallest of changes in the turbulent flow, reducing it further would not have produced acceptable results. Total numbers of elements are 514180, and total nodes are 95657, 140 at the inlet and 128 at the outlet. The commercially available CFD software Ansys Fluent 14.0 is used.

Boundary Conditions

A user defined function was used to define the velocity at the inlet maintaining the mean velocity at 0.75m/s. the boundary between air and water couldn't be distinguished at the outlet and hence the criteria was set to pressure outlet. The forces normal to the free water surface were considered to be zero. Bed and side walls were taken as non-slip walls with their respective roughness heights and the standard wall function (Launder and Spalding, 1974) approach was employed. The pressure strain was considered to be quadratic in nature owing to the complexity of natural channel flow. The flow domain was solved using the body force weighted discretization scheme due to the presence of gravity forces in the flow. Momentum, turbulent kinetic energy and turbulent dissipation rate was discretized using the second order upwind scheme. Reynolds stresses were discretizedusing the QUICK scheme. The modified high resolution interface capturing scheme was used for volume fraction. Fluent uses a staggered cell approach, hence the pressure and velocity were coupled using the PISO scheme which is designed specifically for transient analysis. Temporal discretization was done using the second order implicit time formulation.

Convergence Criteria

All the residual targets were set at 10-3 value which indicates qualitative convergence. At this point almost all flow features had been established. In addition to monitoring the residuals mass balances are also to be checked to satisfy continuity. The net imbalance was less than 1 percent of smallest flux through the domain boundary.

RESULTS & DISCUSSION

Velocity contour (Fig 1) showed good agreement with the literature of Odgaard and Kennedy (1983) plotted at different sections along the flow showed, the maximum velocity at the outer bank especially between 500and 800(Fig 2, 3, 4, 5) bend of the Barak River. The higher near surface velocity at the corners of the outer bank confirm the existence of secondary current. They also explain the highly three dimensional flow nature of water along the outer bank of the river and it being a reason for vortex production.

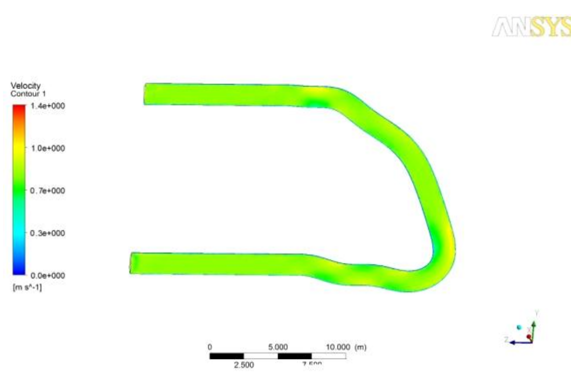

Fig 1. Velocity Contour of Barak River simulated using Reynolds Stress Model (RSM)

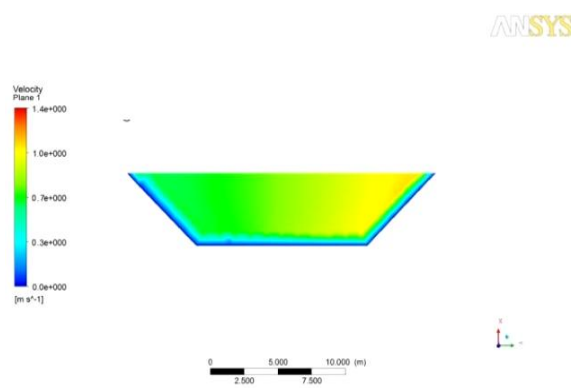

Fig 2. Velocity Contour at 50^0 channelsection

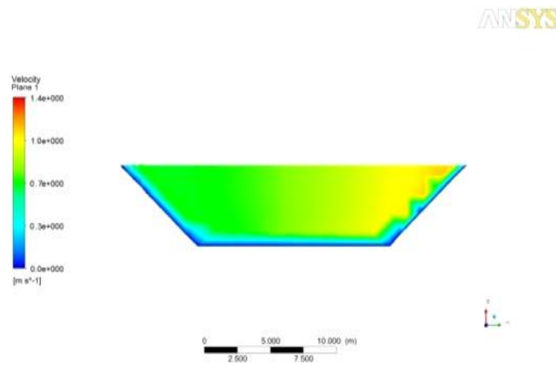

Fig 3. Velocity Contour at 60^0 channelsection

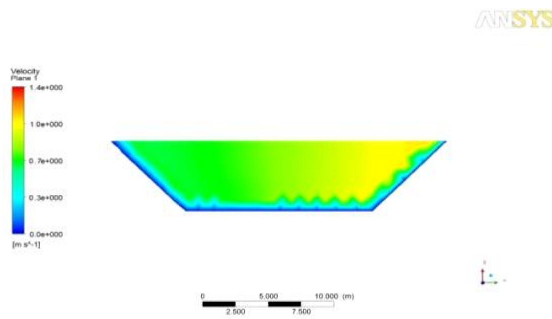

Fig 4. Velocity Contour at 70^0 channelsection

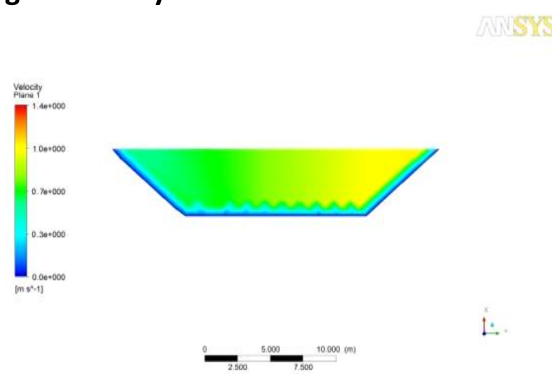

Fig 5. Velocity Contour at 80^0 channelsection

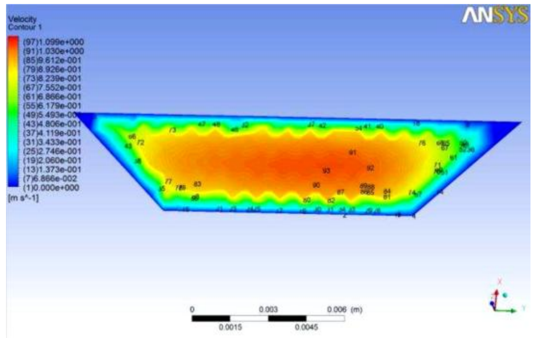

Fig 6. Velocity Contour at 80^0 channel section k–ε turbulence model, Karthik (2014)

Fig 7. Satellite image of Barak river (Courtesy: Google maps)

A comparison has been made with simulation done considering the K-Epsilon model by Karthik (2014) (Fig 6) where the free surface velocity was predicted to be zero which seemed implausible.

The shortcomings of the k-ε model in simulating complex channel flows, was overome by the anisotropic Reynolds stress model. The free surface velocity was precisely modeled. Centrifugal force leads to the phenomenon of super elevation, which causes the water in the curved reach of the outer bank to have high near surface velocities resulting in secondary circulation or vortex. As the vortices are carried downstream the secondary flow causes erosion and changes in the bed topography. The two equation k-ε turbulence model couldn't predict the near surface velocity patterns and thus the secondary and helicoids flow pattern due to its isotropic nature. Erosion will be maximum between 500 and 800 zone of the bend due to the higher velocity at the outer banks and subsequent vortex positions. The satellite (Fig 7) image also confirms the same. The velocity contours also give us an idea about the helical nature of flow in this zone which may be utilized to design river training works to prevent bank erosion.

CONCLUSION

The velocity at different sections of the channel showed that the outer bank had higher velocities compared to the inner bank. Erosion was found to occur at the concave bank where higher velocities are prominent and at the same time in the convex bend deposition of the eroded materials occurs. The corresponding velocity contours obtained were compared with the velocity contours simulated using K-Epsilon turbulence model. The results of RSM seemed more appropriate and plausible than the two equation K-Epsilon turbulence model. Satellite image of the study area gives a more explicit concurrence with the simulated results using RSM model.

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. H. Morvan, G. Pender, N. G. Wright, and D. A. Ervine "Three-Dimensional Hydrodynamics of Meandering Compound Channels"(2002) DOI: 10.1061 /(ASCE) 07339429(2002)128:7(674)

3. D. Cokljat and B. A. Younis, "Second-order Closure Study Of Open-channel Flows" (1995) Journal of hydraulic engineering. Vol. 121, no.2. ASCE ISSN 0733-9429/95/0002-0094-0107.

4. W. Van Balen, S. J. Uijttewaal and K. Blanckaert, "Large-eddy simulation of a mildly curved open-channel flow", Journal of Fluid Mechanics / Volume 630 / July 2009, pp. 413- 442.

5. Q. Y. Yang, T. H. Liu, W. Z. Lu, and X. K. Wang, "Numerical Simulation of Confluence Flow in Open Channel with Dynamic Meshes Techniques". Advances in Mechanical Engineering volume 2013 (2013), Article ID 860431.

6. S. Han, P.M. Biron and A. S Ramamurthy, "Three dimensional modeling of flow in open channel sharp bends with vanes."J. Hydraulic. Res., 49, pp. 64–72, 2011.

7. Sanjiv K. Sinha, Fotis Sotiropoulos and A. Jacob Odgaard, "Three-dimensional Numerical Model for Flow through Natural Rivers" (1998) Journal of Hydraulic Engineering, Vol. 124, No. I, January, 1998. ASCE ISSN 0733-9429/98/0001-00130024.

8. K. Ansari, H. P. Morvan and D. M. Hargreaves, "Numerical Investigation into Secondary Currents and Wall Shear in Trapezoidal Channels" (2011) DOI: .1061/(ASCE)HY.1943-7900.0000317

9. Anupam Dewan, "Tackling Turbulent Flows in Engineering" (2011) Springer ISBN DOI 10.1007/978-3-642-14767-8.

10. Donald W. Knight, Mazen Omran and Xiaonan Tang, "Modeling Depth Averaged Velocity and Boundary Shear in Trapezoidal Channels with Secondary Flows" (2007) DOI:0.1061/ASCE0733-9429 (2007)133:139.