Investigating the Suitability of Grid and Boundary Conditions on Simulation of A Curved Open Channel

U. Ghani[1], H. Nisar[2], A. Latif [3], N. Ejaz[4]

[1,2,4]*Civil Engineering Department, University of Engineering and Technology Taxila, Pakistan*
[3]*Civil Engineering Department, University College of Engineering and Technology BZU. Multan, Pakistan*
[1]usman.ghani@uettaxila.edu.pk

Abstract-This research paper presents results from a numerical modeling of an open channel flow. The channel investigated in this study had a semicircular section. The bed of the channel was rough. During this work, a three dimensional Computational Fluid Dynamics code Fluent has been used. The grid was developed with the help of Gambit. Three different grid shapes including structured, unstructured and structured with a boundary layer development were employed during the simulation work. Two different boundary conditions were considered to check their suitability in this numerical modelling. The first one was velocity inlet and pressure outlet boundary conditions whereas second one was periodic boundary conditions applied at the inlet and outlet of the channel. The results were presented in the shape of primary velocity contours. It was observed that the type of grid did not have any significant impact on the flow structure although structured grid with boundary layer in the vicinity of bed has given slightly better results. As far as boundary conditions are concerned, the periodic boundary condition has reduced the time consumption of the simulation work by reducing the domain size without compromising on the accuracy of the results. From this study it can be concluded that structured grid with boundary layer when used with periodic boundary conditions will produce accurate results with least simulation time and cost consumption.

Keywords-CFD, Fluent, Boundary Layer, Periodic Boundary Conditions, Turbulence Model

I. Introduction

In Computational Fluid Dynamics (CFD), selection of a suitable grid and application of appropriate boundary conditions are of prime importance to get proper simulation. Any problem in grid generation or wrong selection of boundary conditions will result in divergence and wrong solution. Proper results can only be obtained through good mesh and suitable boundary conditions.

Different researchers have worked on investigation of suitable boundary conditions for a particular problem. For example, reference [i] investigated the impact of boundary conditions on density dependent flow. Reference [ii] obtained different bench mark solutions in vertical channel flows under the action of varying boundary conditions. Some researchers conducted research work on geophysical flows with different boundary conditions [iii]. Reference [iv] performed research work on nonlinear flows in open channels with open boundary conditions. Reference [v] performed numerical modeling on levee failures using complex boundary conditions. Other researchers who investigated boundary conditions in different scenarios include research on two-dimensional tidal flows [vi] and investigation of incompressible flows with SPH [vii].

The open channel flows are very much complex especially during floods. In flood season, floodplains become inundated and water damages life and property. The flood flows are complex hydraulic processes due to a number of parameters. Among there are movement and deposition of sediments in the flow, presence of vegetation on the floodplains, variation of bed roughness from reach to reach and also in lateral direction over a cross section, varying cross section of the floodplains and main channel (such as converging or diverging channels), presence of floodplains on one or both sides of the main channel, etc. Flooding happens in different parts of the world every year. The river water overtops the main channel and starts flowing on the floodplain during floods. Hetrogeneous bed forms, strips and ridges exist in a variety of environments over floodplains such as in gravel-bed rivers and ephemeral stream beds. The research has been conducted by different investigators using numerical techniques and also through laboratory experimentation on open channel flow problems with above mentioned situations. Some of them include numerical modeling for cellular secondary currents and suspended sediment transport on smooth-rough bed strips[viii]. Similarly, reference [ix] worked on impact of non-uniform roughness on bedforms. Reference [x] conducted research on secondary flows in open

Channels.

This paper presents numerical modeling of an open channel comprised of a curved section. Different grids and boundary conditions were employed to get the most suitable for this particular case. A general purpose CFD code Fluent has been used for this purpose. Mesh was generated by Gambit 2.3. Results for different cases have been presented in the shape of primary velocity contours on different cross-sections along the length of the channel. The section II explains the governing equations of the CFD code, section III covers different options employed in numerical simulation. Section IV covers results and discussion which is followed by conclusions.

II. GOVERNING EQUATIONS

The CFD code employed in this research work is a finite volume based model. It uses 3D incompressible momentum and continuity equations. These equations can be written as below [xi]

Continuity equation

$$\frac{\partial U_i}{\partial x_i} = 0$$

Momentum equation

$$U_j \frac{\partial}{\partial x_j}(U_i) = \frac{\nu}{\rho}\frac{\partial}{\partial x_j}\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right) - \frac{1}{\rho}\frac{\partial P}{\partial x_i} + F_i + \left(-\overline{u_i u_j}\right)$$

Where ν and ρ are kinematic viscosity and density, P is pressure of the water, U_i is velocity vector in *ith* direction i.e. x, y, z directions, $u_i u_j$ are Reynolds stresses which are result of splitting instantaneous velocities into fluctuating components and average components of velocities.

III. NUMERICAL MODEL SETUP

The mesh was generated with the help of GAMBIT 2.3 software [xii] which is available with FLUENT 12 [xiii]. The meshes were comprised of quadrilateral, triangular and combination of two forms in three different cases as shown in Fig. 1, 4 and 6. Out of these Fig. 1 shows structured mesh, Fig. 4 depicts unstructured mesh whereas Fig. 6 indicates mesh comprising of structured elements and boundary layer regions. Once developed in Gambit, these meshes were exported to Fluent. The boundary conditions employed include, velocity inlet, pressure outlet, wall boundary condition on the bed and side walls and free surface was treated as symmetry plane. The velocity value at the inlet was taken as 0.39317 m/s. The roughness height at the bed and walls was taken as 3×10^{-6}. Atmospheric pressure was assumed at the outlet. In case of periodic boundary condition inlet and outlet were treated as periodic boundaries. For this periodic boundary condition, a value of mass flow rate of 1.645 kg/s was taken at the inlet/outlet. It was ensured that the y^+ distance was such that the first cell lied with in fully turbulent region and log-law exists in that part of the flow. The convergence criterion was set as 1×10^{-6}. The under-relaxation factors were set at their default values The SIMPLE algorithm was taken for pressure-velocity coupling. The first-order upwind scheme has been incorporated for momentum, continuity, turbulence kinetic energy etc. Mesh independence was performed before simulation of different cases.

IV. RESULTS AND DISCUSSIONS

The structured mesh has been shown in Fig. 1. The residual diagram for this mesh has been shown in Fig. 2. This diagram shows that there is a good convergence history for this mesh type. Fig. 3 represents the contour plots of primary velocity vectors over different sections along the span of the channel. From this diagram it is very clear that velocity was uniform at the inlet and then gradually develops as we moved downstream of the channel. The numerical model captured these aspects with much accuracy. The velocity profiles are shown on half of the channel due to symmetric cross-section of the channel. Doing so reduces the cost and time consumption required for simulation.

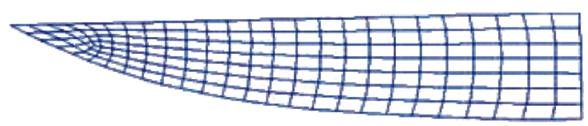

Fig. 1. Structured mesh over the cross section

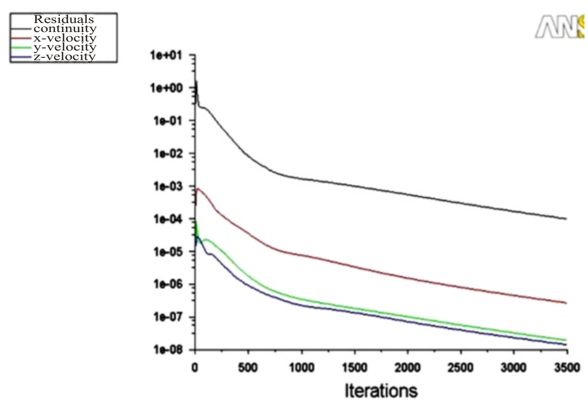

Fig. 2. Residuals for structured mesh

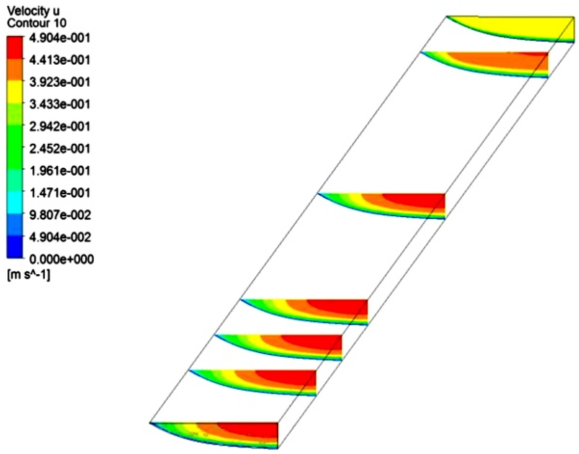

Fig. 3. Contour plots for primary velocities along the span of the channel

The above mentioned velocity profiles are perpendicular to mid section and free surface because these have been taken as axis of symmetry. The mesh shown in Fig. 4 is an unstructured mesh comprised of triangular elements. The corresponding velocity contours obtained by simulating this mesh are shown in Fig. 5. The results are not so good indicating poor performance of the mesh. This might be due to misappropriation of mesh elements for this case.

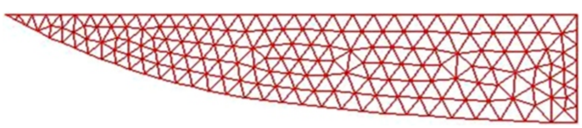

Fig. 4. Unstructured mesh over the cross section

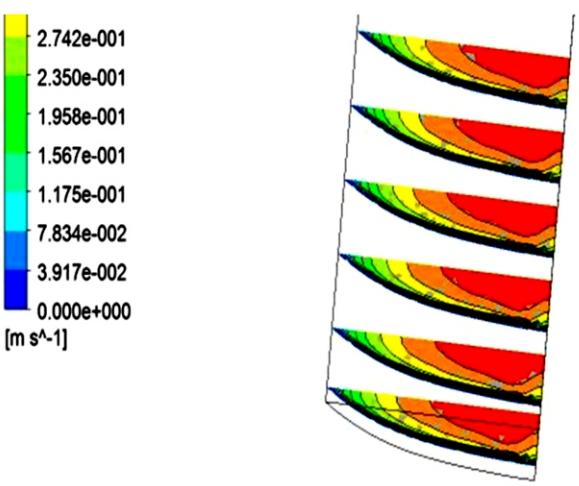

Fig. 5. Contour plots for primary velocities along the span of the channel for unstructured mesh

The Fig. 6 shows third type of mesh tested in this research work. It is comprised of boundary layer region close to bed of the channel whereas in the inner and upper regions, the quadrilateral elements have been used. Results from this mesh have been shown in Fig. 7 which are of good quality. These are also perpendicular to mid section and free surface due to condition of symmetry boundary condition employed there.

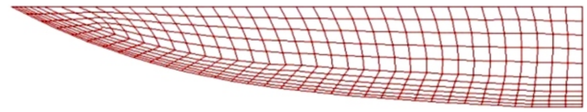

Fig. 6. Structured mesh with boundary layer over the cross section

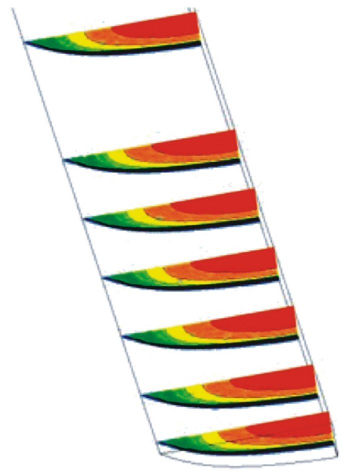

Fig. 7. Contour plots for primary velocities along the span of the channel for structured/boundary layer mesh

As far as periodic boundary condition is concerned, it was employed on the meshes of Fig. 1, 4 and 6 but with a reduced domain. The channel length for this case was reduced to only 1 m. With periodic boundary condition applied at inlet and outlet for these cases, the fully developed flow was achieved quickly. The results of primary velocity contours obtained for structured and boundary layer regions are similar to the one obtained previously however for unstructured it was slightly better but in all these cases, simulation converged quickly due to reduced domain. The results for these cases have been shown in Fig. 8-10.

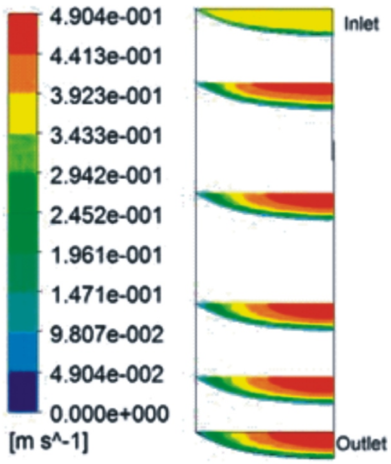

Fig. 8. Contour plots for primary velocities along the span of the channel with periodic boundary conditions for structured mesh

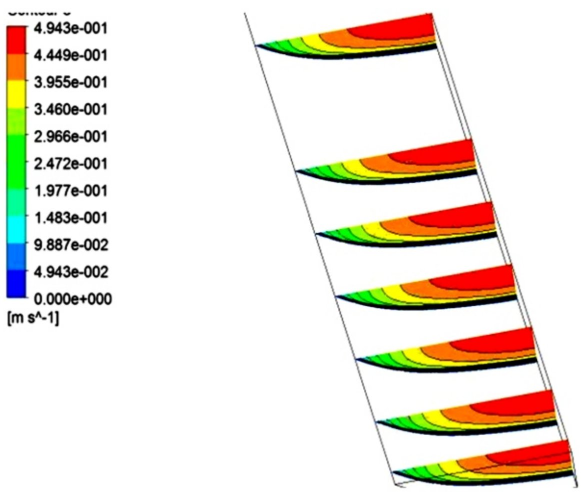

Fig. 9. Contour plots for primary velocities along the span of the channel with periodic boundary conditions for unstructured mesh

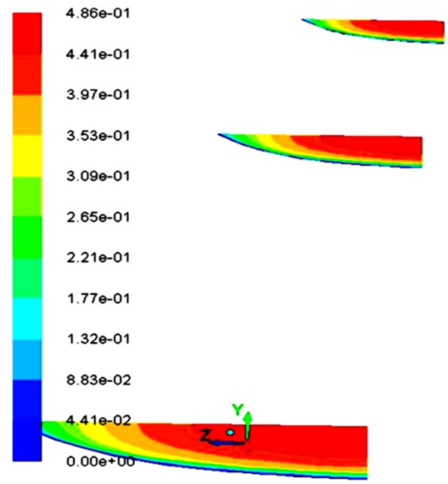

Fig. 10. Contour plots for primary velocities along the span of the channel with periodic boundary conditions for structured/boundary layer mesh

V. CONCLUSIONS

The results from a numerical investigation have been given in this research paper. It included the investigation regarding suitability of mesh type and boundary condition on simulation process of a curved open channel. It was observed that structured and boundary layer meshes produced similar results but unstructured gave poor results. As far boundary condition is concerned it was found that periodic boundary was helpful in reducing number of cells, time consumption due to less number of cells, and thus less cost of simulation was required for this case without compromising the accuracy of the results. So it can be concluded that boundary layer structured mesh with periodic boundaries will be most suitable for this type of open channel flow.

REFRENCES

[i] B. Ataie-Ashtiani, C. T. Simmons, and A. D. Werner, Influence of Boundary Condition Types on Unstable Density-Dependent Flow. Journal of Groundwater, 2014. 52(3): p. 378-387.

[ii] G. Desrayaud, E. Chénier, A. Joulin, A. Bastide, B. Brangeon, J. P. Caltagirone, Y. Cherif, R. Eymard, C. Garnier, S. Giroux-Julien, Y. Harnane, A. Sergent, S. Xin, and A. Zoubir, Benchmark solutions for natural convection flows in vertical channels submitted to different open boundary conditions. International Journal of Thermal Sciences, 2013. 72: p. 18-33.

[iii] A. Bousquet, M. Marion, M. Petcu, and R. Temam, Multilevel finite volume methods and boundary conditions for geophysical flows". Journal of Computers & Fluids, 2013. 74: p.66-90.

[iv] J. Nycander, A. M. Hogg, and L. M. Frankcombe, Open boundary conditions for nonlinear channel flow. Journal of Ocean Modelling, 2008. 24: p. 108-121.

[v] M. H. Yu, Y. Deng, L. Qin, D. Wang, Y. Chen, Numerical Simulation of Levee Breach Flows Under Complex Boundary Conditions. Journal of Hydrodynamics, Ser. B, 2009. 21 (5): p.633-639.

[vi] J. Zhang, Y. P. Wang, A method for inversion of periodic open boundary conditions in two-dimensional tidal models. Journal of Computer Methods in Applied Mechanics and Engineering, 2014. 275: p. 20-38.

[vii] S. M. Hosseini, and J. J. Feng, Pressure boundary conditions computing incompressible flows with SPH. Journal of Computational Physics, 2011. 230 (19): p. 7473-7487.

[viii] S. K. Choi, M. Park, and H. Kang, Numerical simulations of cellular secondary currents and suspended sediment transport in open channel flows over smooth-rough bed strips. Journal of Hydraulic Research, 2007. 45(6): p. 829-840.

[ix] S. R. McLean, The role of non-uniform roughness in the formation of sand ribbons. Journal of Marine Geology, 1981. 42: p. 49-74.

[x] A. Muller, and X. Studerus, Secondary flow in an open channel. Proceedings of 18th IAHR Congress, 1979. 3: p. 19-24.

[Xi] W. Rodi, Turbulence Models and Their Applications in Hydraulics, A State of the art Review. IAHR Monograph series Delft, Netherland, 1980.

[xii] User Guide Gambit 2.3. Gnterra Resources Park 10 Cavendish Court, Lebanon: New Hamshire, USA, 2010.

[xiii] User Guide Fluent 12. Lebnon: FLUENT 12 Incorporated Lebanon, New Hamshire, USA, 2011.