Three Dimensional CFD Modeling of Flow Structure in Compound Channels

USMAN GHANI*, PETER RICHARD WORMLEATON**, AND HASHHIM NISAR HASHMI***

RECEIVED ON 27.11.2008 ACCEPTED ON 04.03.2009

ABSTRACT

The computational modeling of three dimensional flows in a meandering compound channel has been performed in this research work. The flow calculations are performed by solving 3D steady state continuity and Reynolds averaged Navier-Stokes equations. The turbulence closure is approximated with standard-turbulence model. The model equations are solved numerically with a general purpose software package. A comprehensive validation of the simulated results against the experimental data and a demonstration that the software used in this study has matured enough for investigating practical engineering problems are the major contributions of this paper. The model was initially validated. This was achieved by computing streamwise point velocities at different depths of various sections and depth averaged velocities at three cross sections along the main channel and comparing these results with experimental data. After the validation of the model, predictions were made for different flow parameters including velocity contours at the surface, pressure distribution, turbulence intensity etc. The results gave an overall understanding of these flow variables in meandering channels. The simulation also established the good prediction capability of the standard k-ε turbulence model for flows in compound channels.

Key Words: Compound Channels, Point Velocities, Numerical Simulation, Turbulence Intensity.

1. INTRODUCTION

The hydraulic characteristics of natural river flood plains are not well understood at present. This is due to the problems encountered in monitoring spatially distributed patterns of flow depths, velocity, turbulence characteristics etc. For designing the flood protection strategies, it is very important for river engineers to accurately predict water levels that may be expected due to any flood discharge. One of the consequences resulting from the more recently recognized hazards of climate change is the potential to increase the levels and occurrence of flooding worldwide. All this requires research on natural river flows.

Meandering channel flows being highly complicated are a matter of recent and continued research. There are two broad types of studies namely loose boundary channel flows and rigid boundary channel flows. Loose boundary flows involve sediment transport investigations which result in change of width and bed levels of the channel.

* Assistant Professor, Department of Civil Engineering, University of Engineering and Technology, Taxila, Pakistan.
** Senior Lecturer, Department of Engineering, Queen Mary, University of London, UK.
*** Professor, Department of Civil Engineering, University of Engineering and Technology, Taxila, Pakistan.

The present study deals with investigations of rigid boundary meandering flows without any change in width and slope. A lot of early work on two stage channels has focused on straight main channels with parallel floodplains. A number of straight channel experiments with overbank flows were carried out in the past which were summarised at a later stage [1]. These straight channel overbank flow experiments observed zones of high shear stress in the region of interface between relatively fast moving inner channel flow and the slower moving floodplain flow. These shear zones create turbulent energy losses which result in reduction of the conveyance of the channel. Some researchers used photographic method in order to observe a bank of vertical vortices generated in the interface region between the fast flowing main channel and slower moving floodplain. The magnitude of this interaction and its effect upon flows has been the subject of much research in recent years. However, until recently, considerably less research has been conducted on meandering overbank channels. In case of straight channels with overbank flows, turbulent anisotropy generated from boundary resistance drives secondary circulation and this dominates at the main channel/floodplain interface. But for meandering compound channels the situation is different. Here, the flow mechanisms developed due to shear between main channel and floodplain are completely different from that of a straight channel. It depends not only on sinuosity of the main channel but also on the fact that main channel and floodplains are not parallel to each other. Inbank flows in meandering channels are characterized by circulation cells generated by pressure variation due to centrifugal forces around the meander bend. This circulation cell dies out and then re-establishes itself in the opposite direction while moving from one meander apex to the next. For overbank flows there are two forces contributing to secondary flows. One is the pressure driven force and the other one is the shear driven. The shear driven forces are due to shear at floodplain/main channel interaction. This force develops at the cross overs. The shear driven circulation reverses in direction from one apex to the other and thus becomes opposite to centrifugal force based secondary flows [2-3]. This has been shown in Fig. 1. Thus for meandering channels the flow behaviour is mainly controlled by these centrifugal and shear driven cells while anisotropic turbulence is relatively weak.

In past a lot of experimental research has been done on compound channel flows but relatively less usage has been made of numerical techniques. But after the development of powerful computers and sophisticated CFD (Computational Fluid Dynamics) techniques, much research is now being conducted using these techniques in different research areas. This is not only due to economy and less time consumption of CFD methodology but also due to the fact that through CFD one can cover those aspects of flow behaviour which are very difficult to observe through experimentation. A number of CFD packages (Fluent, CFX, Star-CD etc) are now available and are also being used for research in water flows. In recent past, a good number of researchers have used these software packages for prediction of different aspects of 3D flow fields [4-8]. They detected that flow features in compound channels are dependent on topography of the channel, its sinuosity, surface roughness etc. However how actually the flow behaviour changes is still an unresolved phenomenon and attempts are on to handle this problem These researchers attempted to predict the flow behaviour using different numerical models as it is difficult to capture all flow features experimentally but still a lot of work is to be done. This is due to various problems which are encountered in numerical modelling such as grid generation, choice of turbulence model, discretizing scheme, specifying the boundary and initial conditions etc.

In this work, an attempt has been made to improve the understanding of 3D flows in meandering channels. For this purpose, a 3D numerical code FLUENT has been tested for its suitability for simulation of flood flows. Initially, the closure problem of governing equations was handled. There is no universal closure model which is acceptable for all flow problems. Every one has its own advantages and disadvantages. Therefore, one has to attempt different models and then to choose the one producing best results. The models tested here were standard k-ε, RNG k-ε and k-ω. These were used for their capability in the simulating point velocities at section E. The one with best out put (standard k-ε in this case) was then used for all simulation works. It was used for prediction of resultant velocity contours on free surface, pressure, turbulence intensity and secondary flow velocities at different sections along the meander wavelength.

2. GOVERNING EQUATIONS

FLUENT solves three dimensional Reynolds-averaged continuity and Navier-Stokes equations. The case considered here is steady state incompressible flow. In Cartesian coordinates these equations are:

Continuity equation

$$\frac{\partial U_i}{\partial x_i} = 0 \tag{1}$$

Momentum equation

$$U_j \frac{\partial}{\partial x_j}(U_i) = \frac{\nu}{\rho}\frac{\partial}{\partial x_j}\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right) - \frac{1}{\rho}\frac{\partial P}{\partial x_i} + F_i + \left(-\overline{u_i u_j}\right) \tag{2}$$

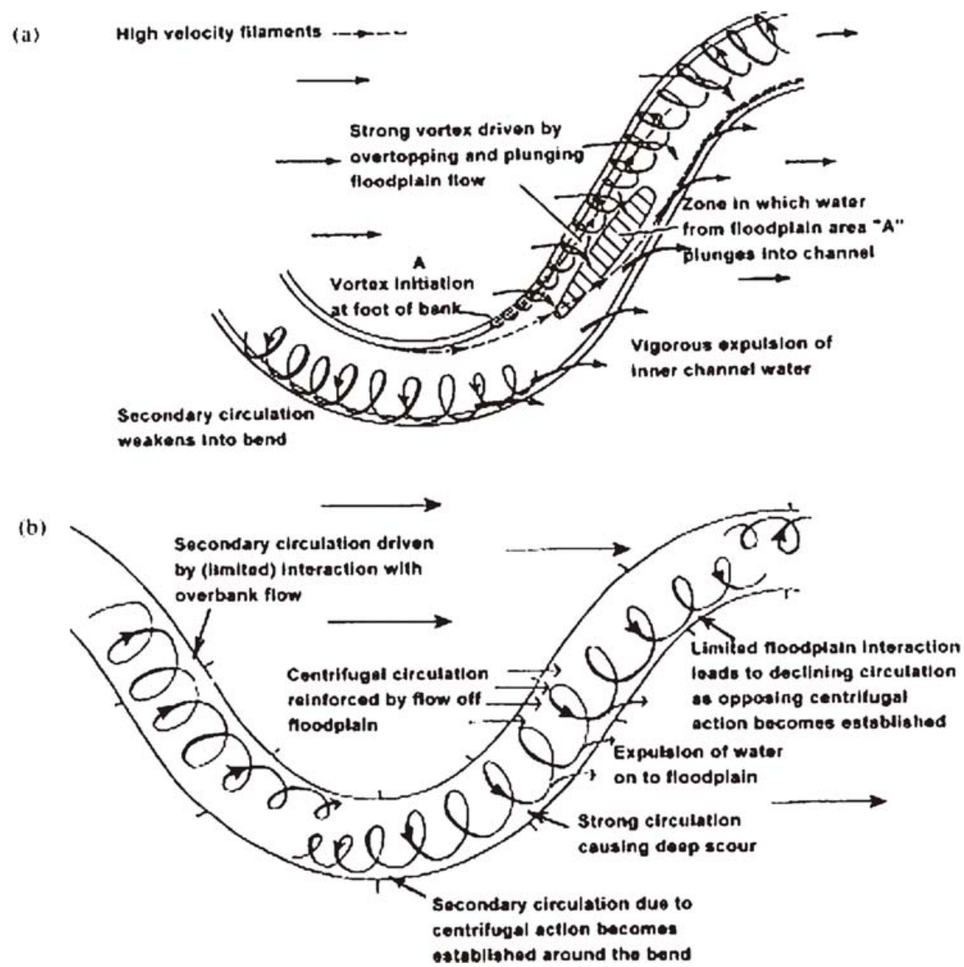


FIG. 1. FLOW MECHANISMS IN MEANDERING COMPOUND CHANNELS (A) STRONG FLOODPLAIN FLOW, (B) WEAK FLOODPLAIN

Where P is the pressure, ν and ρ are the kinematic viscosity and density of the water, U_i is the time-averaged velocity component in x_i direction, F_i denotes external force, and u_iu_j are the Reynolds stresses which result from the decomposition of instantaneous velocities into their mean and fluctuating components. The Reynolds stresses are expressed as:

$$-\overline{u_i u_j} = \nu_t \left(\frac{\partial U_i}{\partial U_j} + \frac{\partial U_j}{\partial U_i} \right) - \frac{2}{3} k \delta_{ij} \tag{3}$$

Where v_t is the eddy viscosity, k is the turbulent kinetic energy, δ_{ij} is the Kronecker delta function. The turbulent or eddy viscosity is expressed in terms of turbulent kinetic energy k and its dissipation rate ε using Kolmogorov-Prandtl expression as:

$$\nu_t = c_\mu \frac{k^2}{\varepsilon} \tag{4}$$

Where c_μ is a constant. The turbulent kinetic energy k and its dissipation rate ε are determined using transport equations (If we try to use Reynolds stress model, it will need more simulation time and powerful computer capability because Reynolds stress model solves seven extra equations instead of two equation k-ε model and this will require extensive computational effort and time).

$$U_i \frac{\partial k}{\partial x_i} = \frac{\partial}{\partial x_i} \left(\frac{\nu_t}{\sigma_k} \frac{\partial k}{\partial x_i} \right) + P_k - \varepsilon \tag{5}$$

$$U_i \frac{\partial \varepsilon}{\partial x_i} = \frac{\partial}{\partial x_i} \left(\frac{\nu_t}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x_i} \right) + \frac{\varepsilon}{k} \left(c_{1\varepsilon} P_k - c_{2\varepsilon} \varepsilon \right) \tag{6}$$

where σ_k, σ_ε, $c_{1\varepsilon}$, $c_{2\varepsilon}$ are empirical constants and P_k is the result of turbulent kinetic energy and is expressed as:

$$P_k = \nu_t \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \frac{\partial U_i}{\partial x_j}$$

In these equations c_μ, σ_k, σ_ε, $c_{1\varepsilon}$, $c_{2\varepsilon}$ are empirical constants. They have their universal constant values which have been derived on the basis of high quality data. Their values vary from one turbulence model to another. For any particular turbulence model, the values of these constants remain same for all simulation purposes. For standard k-ε, their values are as:

$c_\mu = 0.09$, $\varepsilon_k = 1.0$, $\sigma_\varepsilon = 1.3$, $c_{1\varepsilon}$=1.44, $c_{2\varepsilon}$=1.92

3. EXPERIMENTAL DATA

`The experimental results used in this work were obtained from Martin Marriott's experiments at the University of Hertford [9]. The planform consisted of a meandering rectangular main channel made up of centre line circular arcs with a radius of 500 mm and included angle of 140 degrees joined tangentially at the cross over points. The resulting sinuosity was 1.3. The floodplain and meander belt width were 1.23and 1.158m respectively. The depth of the main channel was 115mm. The overbank flow depth (depth of floodplain) was 41mm. Thus the total depth of the water became 156 mm. The point velocities were taken at depths of 20, 80 and 140mm above the main channel bed. These measurements were made at nine cross stream locations 50 mm apart from each other. The data was collected at five cross sections i.e. D, R, E, U and F. The plan view of channel geometry and floodplains is shown in Fig. 2(a). The cross sectional dimensions at the apex E alongwith locations of measured point velocities have been shown in Fig. 2(b). The same cross section exists throughout the main channel. The planform is comprised of three meander wavelengths, however only two out of these three wavelengths have been shown.

The inflow for the experiment was set at 49.7 litre/sec. The slope of the floodplain was 0.00175 and bed roughness was 0.0102.

4. BOUNDARY CONDITIONS

Four different types of boundary conditions were considered in this study. These are (i) inlet, (ii) outlet, (iii) water surface, and (iv) walls of the geometry

4.1 Inlet

The velocity distribution at the upstream cross section was taken as inlet boundary condition. At the inlet, turbulence properties i.e. k (turbulence kinetic energy) and (turbulence dissipation rate) must be specified. These were calculated as [11]:

$$k = (IU)^2$$

$$\varepsilon = \frac{k^{3/2}}{l}$$

Where I is the turbulence intensity and U is the mean value of streamwise velocity. l is the turbulence length scale

4.2 Outlet

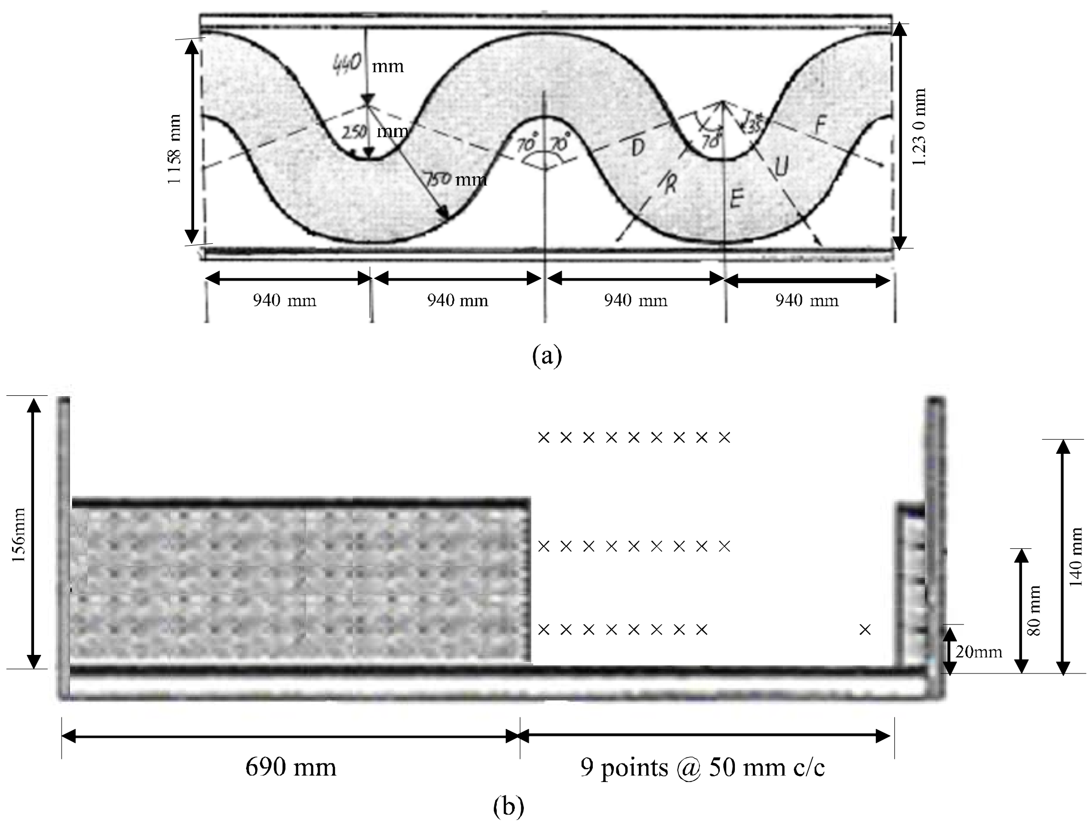
At the outlet, the pressure condition was given as the boundary condition and pressure was fixed at zero.

4.3 Channel and Floodplain Boundaries

A no-slip boundary condition was considered at the walls. This means the velocity components should be zero at the walls. The standard wall-function which uses log-law of the wall to compute the wall shear stress was used [12].

4.4 Free Surface

The simplest way to deal with a free surface is through a rigid lid assumption. In it, the model is superimposed by



FLOW

FIG. 2. THE GEOMETRY USED IN THIS STUDY (A) PLANFORM SHOWING DIFFERENT X-SECTIONS AND DIMENSIONS OF

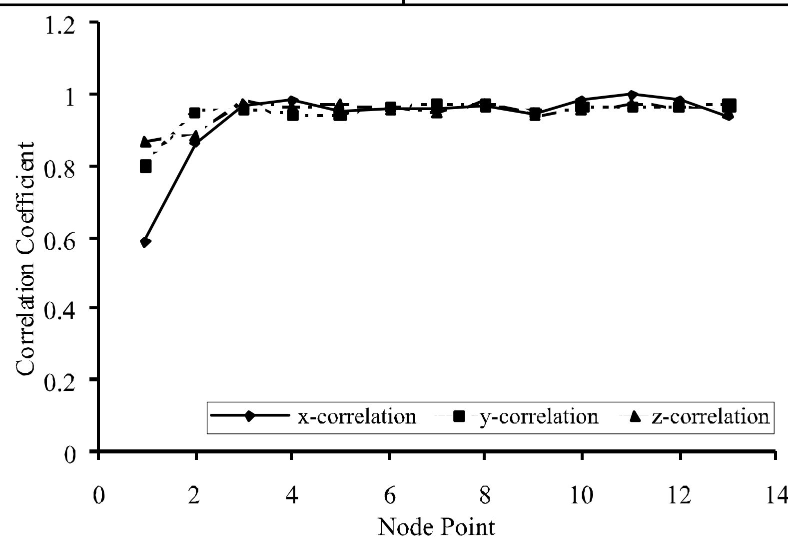
the slope computed from the measured water surface elevations along the channel. The water surface was defined as a plane of symmetry which means that the normal velocity and normal gradients of all variables are zero at this plane of symmetry.

5. FULLY DEVELOPED FLOW

It was ensured that the flow was fully developed and the simulated results in the test section were independent of the inlet conditions. For this purpose, the model should give results which are a mirror image in the y-direction at x-sections one half of a wavelength apart. This was checked by calculating the correlation coefficient between different components of the velocities i.e. x, y and z velocity components at different sections and their reflected partner one half wavelength away. If the correlation coefficient is one, it means the flow is uniform. These coefficients are equal to one in case of y velocity component whereas for x and z velocity components they are around 0.96 and 0.94 respectively. These correlation coefficients for x, y, and z velocities between different sections along the meander wavelength is shown for different as shown in Fig. 3.

6. GRID FORMATION

The GAMBIT software was used to create and mesh the geometry [13]. The geometry formation started first by making edges, then connecting edges to form faces. At the end the volumes were created by stitching six faces. The entire geometry was comprised of four volumes, two floodplains (one on each side of the main channel), third one was the part of the main channel below floodplain and fourth one was the upper part of the main channel within the floodplain. An unstructured mesh was used to discretize the geometry for simulation. This mesh was comprised of Tet/hybrid elements. A Tet/hybrid mesh consists primarily of tetrahedral elements but may include hexahedral, pyramidal and wedge elements where required. Once the grid size decided, it was used for further simulation and prediction. The grid used was made gradually finer as we moved towards the main channel. This was done to capture sharp velocity gradients in these regions. The mesh was coarsest at floodplain edges and then made gradually finer while moving towards the main channel with 201 nodes on the main channel. The final mesh used had the following node numbers 201x15x10 in the main channel. The mesh size used in the floodplains was 36x15x6.



FLOODPLAIN AND MAIN CHANNEL (B) CROSS-SECTIONAL DIMENSIONS AND LOCATION OF VELOCITY MEASUREMENT POINTS (

For checking the grid dependency the number of nodes were doubled in all three directions i.e. x, y, and z turn by turn. The absolute mean differences were calculated for the component velocities in these directions. The largest difference which occurred in any velocity component by doubling the node numbers was small enough that for all practical purposes the results could be termed as grid independent. A post processing check on mesh quality based on assessing the skewness of the generated cells indicated that the mesh is of high quality and would not compromise solution stability. The mesh used for simulation is shown in Fig. 4.

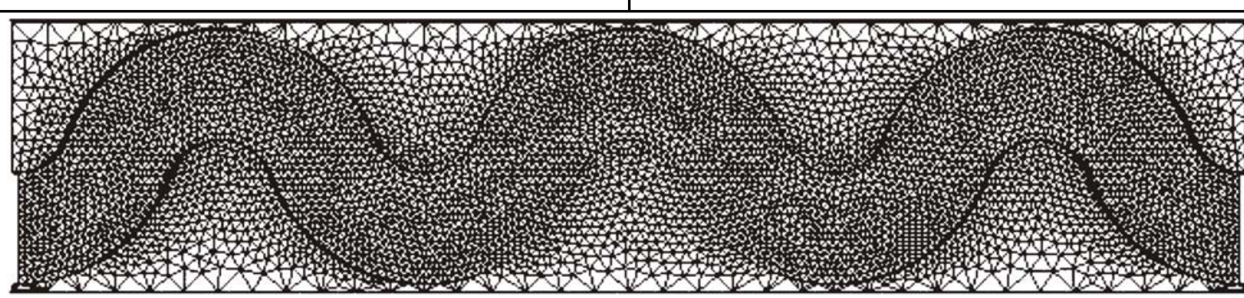
7. NUMERICAL SOLUTION

FLUENT is a control volume based software package [14]. It converts partial differential conservative equations to algebraic equations which can be solved simultaneously. In this research work, a staggered grid was used to define the discrete control volumes. The set of simultaneous equations was solved using explicit iterative scheme. For pressure-velocity coupling, SIMPLE algorithm was used. The derivatives of different variables were obtained using first order upwind scheme. A commonly used procedure to identify a converged solution is to use a residual definition. This definition implies that convergence is reached when the normalized changes in variables between successive iterations are equal to or less than a certain limit. In this case the limits were set as $1x10^{-6}$. The calculation was initiated using the default under-relaxation coefficients provided in the Fluent 6.3.

8. MODEL VALIDATION

Before making any predictions, the model was validated for its accuracy. Model validation involves: (a) mathematical model validation making checks on mass conservation and solution convergence, (b) ensuring grid independence of the simulated results, (c) comparison of the modelled results with the observed ones.

The grid independence has been discussed in section 5 where it was established that the grid chosen was the coarsest one for which results did not differ appreciably from solutions of the finer grids. For making a comparison between simulated and observed results, point velocities were collected in the experiment over five x-sections D, R, E, U and F at locations shown in Fig. 2(b). The model output was sampled at the grid cells equivalent to these measurement locations. The correlation between observed and simulated stream wise velocities (u-velocity component) was made as shown in Fig. 5. Primary DAV (Depth Averaged Velocities) at different x-sections R, E and U were calculated from observed point velocities and compared with modelled results as shown in Fig. 6. The correlation coefficients for stream wise velocities ranged between 0.92-0.947 which is acceptable. Depth averaged



AFTER MARRIOTT [10])

velocities also showed good agreement between simulated and observed ones. This establishes the model ability to predict flows in compound channels.

9. NUMERICAL PREDICTIONS

Once the ability of the model to reproduce the flow structure measured in the experiment was established, it can now be used for predictions of different flow parameters. Predictions have been made for surface velocity contours, pressure and turbulence intensity. The resultant secondary velocity vectors at different sections along the main channel were also predicted. All these results have been shown in Figs. 7-10.

10. IMPLEMENTATION

This research work established the capability of k-ε turbulence model along with it validated the ability of 3D CFD finite volume based code FLUENT in simulating three dimensional flood flows. Using the methodology described in this study, one can design different aspects for

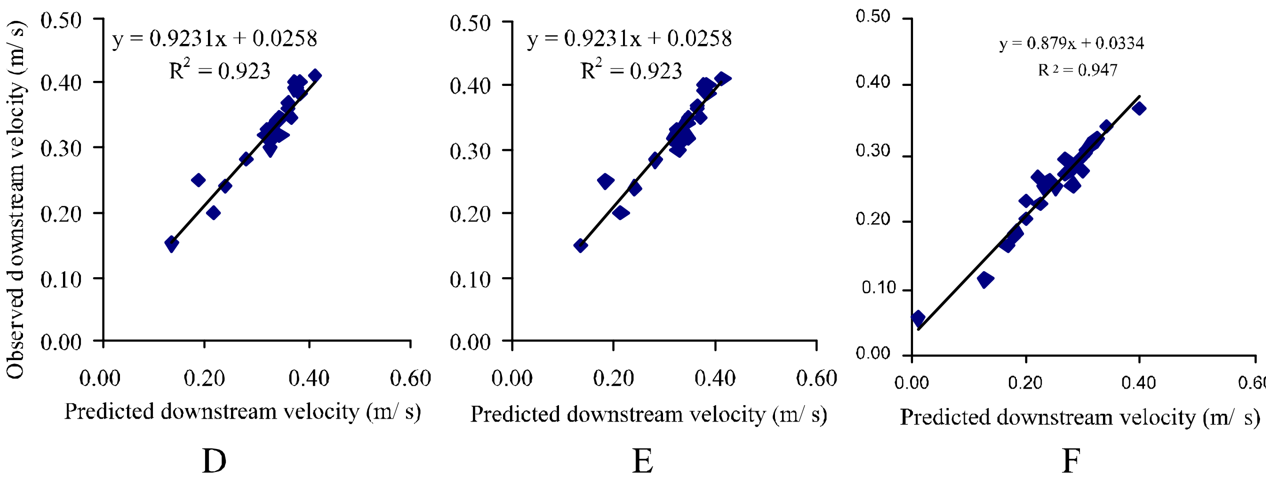
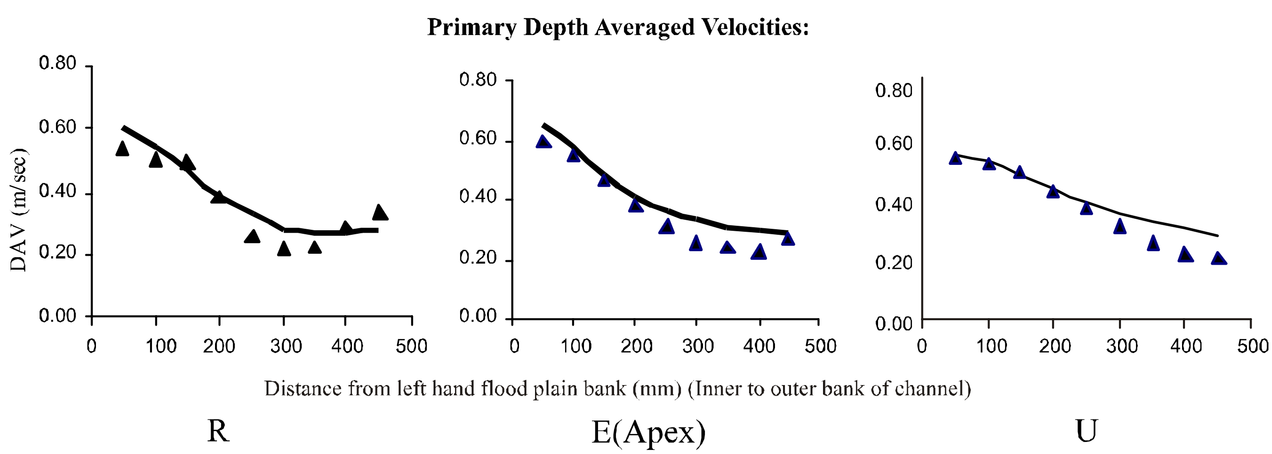


FIG. 3. CORRELATION COEFFICIENTS AT HALF WAVELENGTH APART BETWEEN VELOCITY COMPONENTS(x, y, z VELOCITY



COMPONENTS)

FIG. 4. MESH USED IN THE SIMULATION WORK

meandering compound channels, which may include flood protection embankments, design of bridges over rivers, width of flood plains, development of vegetation over flood plains, etc.

11. RESULTS AND DISCUSSION

The velocity magnitude contours on the free surface are shown in Fig. 7. Here the flow is from left to right. The figure shows that the contour patterns are not so simple. It has been observed that there is a repetition in velocity distribution after regular intervals on both sides of the centre line of the floodplain. The maximum values are on right side of the floodplain while they are relatively weak on left side of the floodplain.

The distribution of pressure in the form of filled contours is shown in Fig. 8. The values of the pressure are decreasing in the streamwise direction. They are maximum towards inner bank of the channel at section D and at outer bank at section R. At the apex there is a gradual increase in pressure values while moving from inner to outer bank. Fig. 9 shows the turbulence intensity contours. A similarity in distribution pattern of turbulence intensity was observed at cross overs D and F. The same happened for sections R, U. It decreases from inner to outer bank in case of cross overs while a mixed distribution was observed at R and U. At the apex it gradually decreases from inner to outer bank. The secondary velocity vectors are shown in Fig. 10. These have been calculated at two sections upstream of the apex i.e. D and R, at the apex E and at two sections downstream of the apex i.e. U and F. The results show that there is a complete circulation of the secondary flows at the apex. At the downstream of the apex, the secondary flows are stronger than those of upstream. This is due to

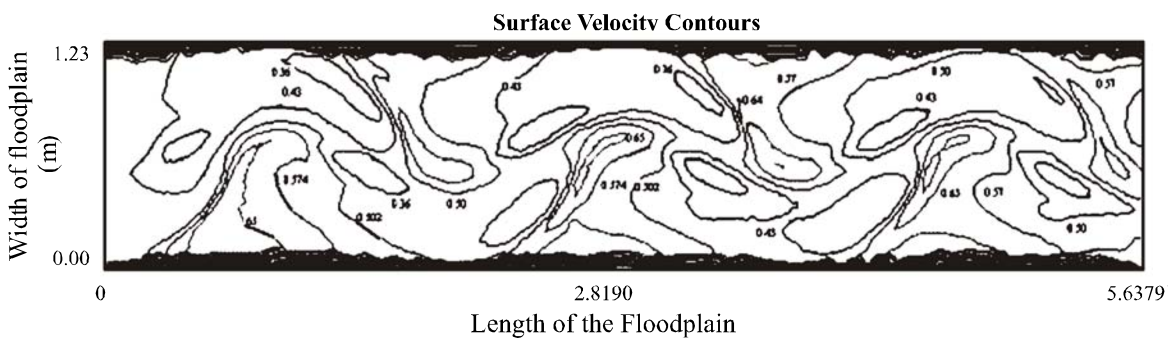


FIG. 5. COMPARISON OF OBSERVED AND MODELLED DOWNSTREAM VELOCITIES

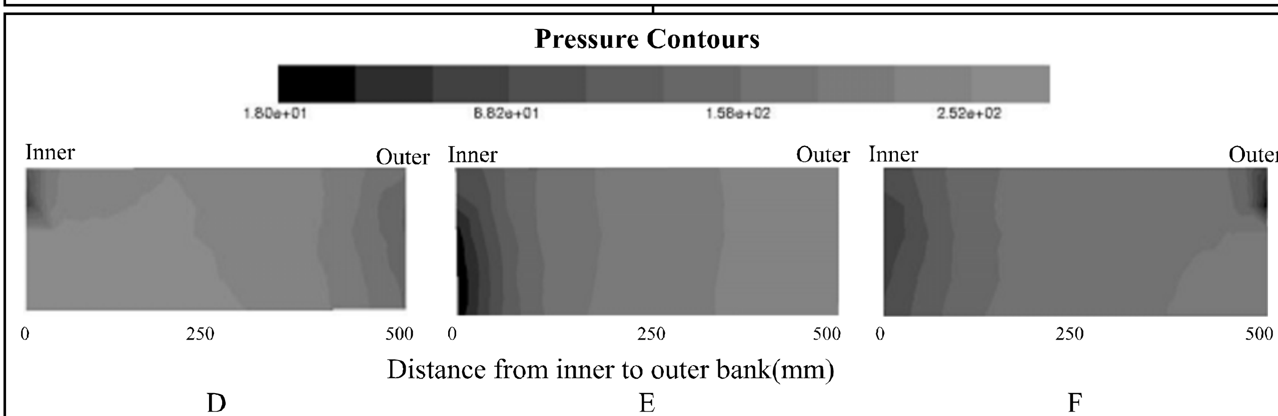


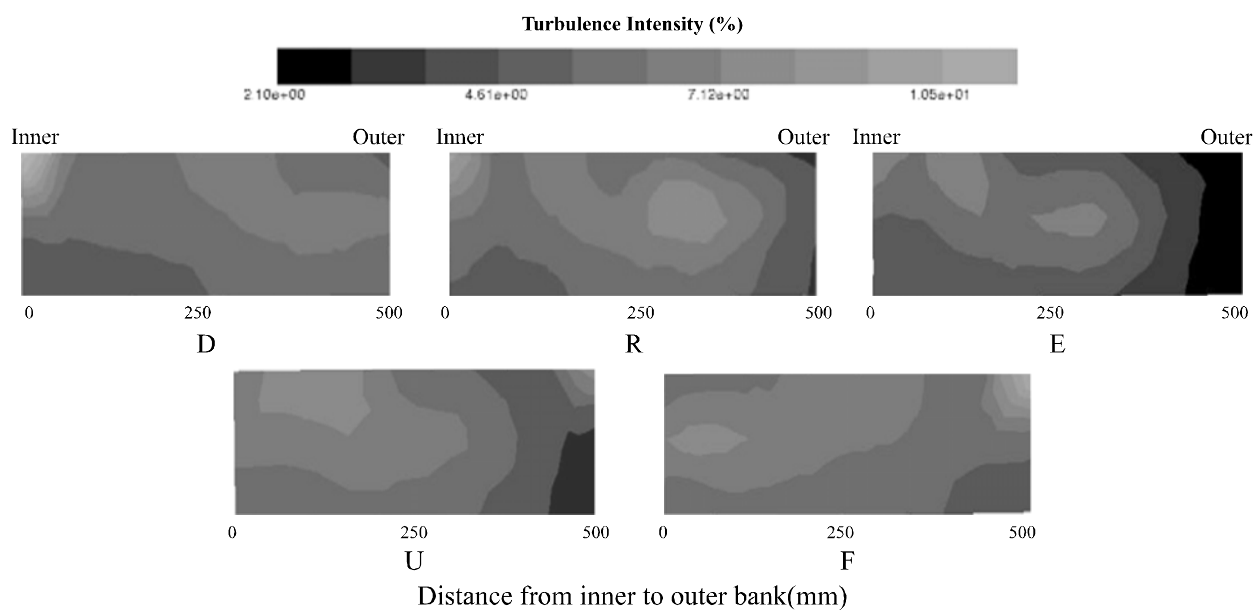
FIG. 6. COMPARISON BETWEEN OBSERVED AND MODELLED DAV (POINTS INDICATE OBSERVED VALUES WHILE SOLID LINE

the fact that both the secondary flows i.e. shear driven circulations from cross over and pressure driven from apex merge together. This makes a strong secondary circulation which flows from inner to outer bank of the main channel resulting not only in erosion of outer bank but also in expulsion of water towards floodplain.

12. CONCLUSIONS

In the past, less focus was made to get hydraulic characteristics of natural floodplains using numerical models. This is basically due to inadequate field data available from such studies. For floodplain research, physical models have been extensively used like the one reported in this paper. But even from such models it is very difficult to visualise a complete picture of turbulent flow structure. Therefore, CFD techniques have to be used for analysis of such turbulent flows.

In this research work the use of a numerical model to simulate the distribution of different flow parameters was demonstrated. It also established that an in depth study of flow structure could be executed through a three dimensional network of three orthogonal velocity components distributed over the whole domain. The standard model which assumes isotropic turbulence was used for simulation of meandering compound channel. It was observed that the flows in meandering compound channels with rectangular cross sections involve secondary flows with multiple vortices. The model showed that at the upstream of the apex (at section R) the secondary flow near the bed was from the outer part of the bend towards the inner part which was expected. This study proved the capability of k-ε model to predict the meandering overbank flows. For many design purposes this model will be sufficient. However, the model may give poor results in cases where more accurate velocity distributions are required for example in the calculation of bed shear stresses, bed morphology and sediment transport. This study also proved that the FLUENT software has the ability to simulate 3D meandering flows.



SHOWS MODELLED RESULTS)
FIG. 7. RESULTANT VELOCITY MAGNITUDE CONTOURS ON THE FREE SURFACE OF WATER

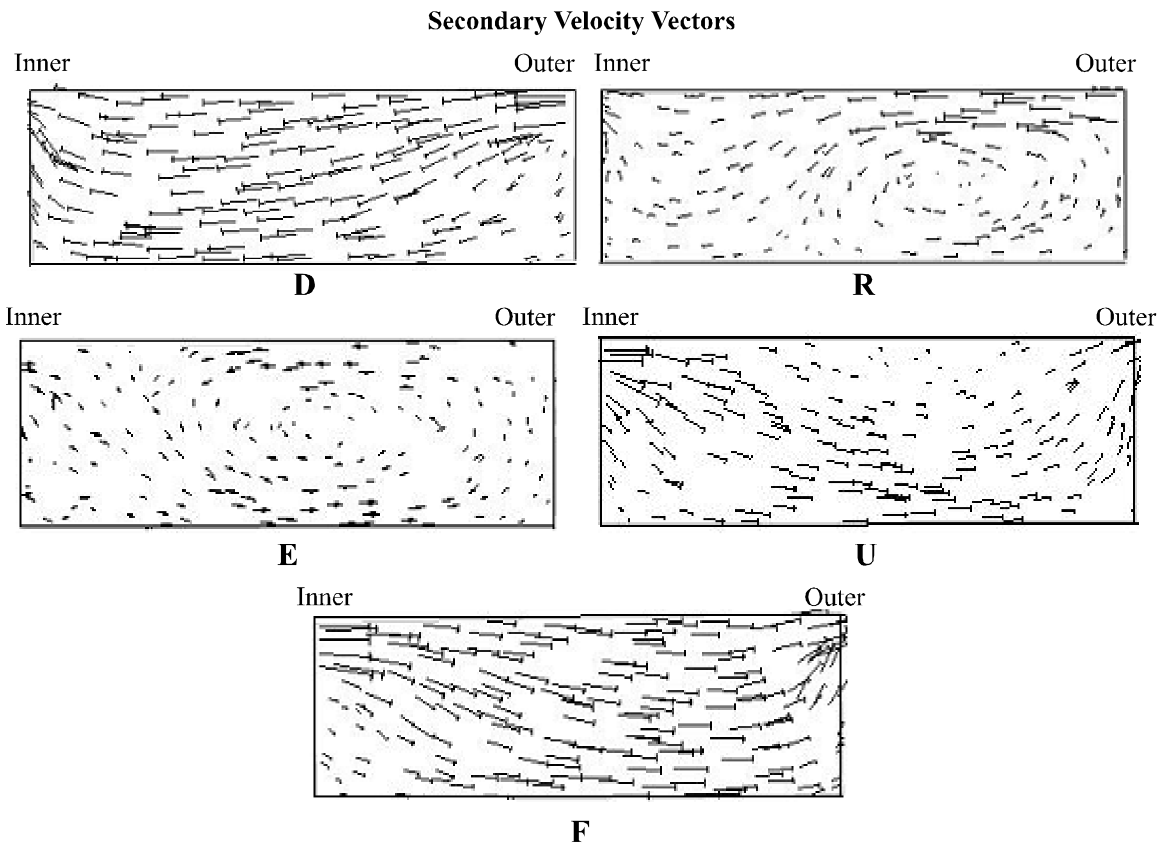


FIG. 8. PRESSURE CONTOURS AT DIFFERENT SECTIONS OF THE MAIN CHANNEL

FIG. 9. TURBULENCE INTENSITY DISTRIBUTION (SHOWN THROUGH FILLED CONTOURS) ON DIFFERENT SECTIONS ALONG MEANDER WAVELENGTH

FIG.10. SECONDARY FLOW VELOCITY VECTORS PLOT ON DIFFERENT CROSS-SECTIONS

ACKNOWLEDGEMENT

The authors are thankful to authorities of University of Engineering and Technology, Taxila, Pakistan, who facilitated to conduct this work.

REFERENCES

[1] Hollinrake, P.G., "The Structure of Flow in Open Channels-A Literature Search HR Wallingford Reports, SR96-January 1987, SR153-January 1988,SR209-June 1989 and SR-227-March 1990" A Literature Search Report, HR Wallingford, England, March, 1990.

[2] Chang, H.H., "Fluvial Processes in River Engineering", John Wiley and Sons, New York, USA, 1988.

[3] Wormleaton, P.R., Sellin, R.H.J., Bryant, T.B., Loveless, J., Hey, R., and Catmur, S.E., "Flow Structures in a Two-Stage Channel with Mobile Bed", IAHR Journal of Hydraulic Research, Volume 42, No. 2, pp.145-162, Delft, Netherland, March, 2004.

[4] Nguyan, V.TH., Nestmann, F., and Scheuerlein, H., "Three Dimensional Computation of Turbulent Flows in Meandering Channels and Rivers", IAHR Journal of Hydraulic Research, Volume 45, No. 5, pp. 595-609, Delft, Netherland, September, 2007.

[5] Zhang, M.L., and Shen, Y.M., "Three Dimensional Simulation of Meandering River Based on 3-D RNG Turbulence Model", Journal of Hydrodynamics, Volume 20, No. 4, pp. 448-455, China, August, 2008.

[6] Shiono, K., Spooner, J., Chan, T., Rameshwaran, J., and Chandler, J., " Flow Characteristics in Meandering Channels with Non-Mobile and Mobile Beds for Overbank Flows", IAHR Journal of Hydraulic Research, Volume 46, No. 1, pp. 595-609, Delft, Netherland, January, 2008.

[7] Shukla, D.R., and Shiono, K., "CFD Modeling of Meandering Channels During Floods", Proceedings of ICE, Water Management, Volume 161, No. 1, pp. 1-12, England, February, 2008.

[8] Wormleaton, P.R., and Ewunteu, M., "Three Dimensional-Numerical Modeling of Overbank Flows in a Mobile Bed Meandering Channels with Floodplain of Different Depths, Roughness and Planform", IAHR Journal of Hydraulic Research, Volume 43, No. 1, pp. 1-15, Delft, Netherland, January, 2005.

[9] Marriott, M.J., "Hydrodynamics of Flow around Bends in Meandering and Compound Channels", Ph.D. Thesis, University of Hertfordshire, UK, 1998.

[10] Marriott, M.J., "The Effect of Overbank Flow on the Conveyance of the Inbank Zone in Meandering Compound Channels", IAHR Congress, Volume 28, pp. 145-156, Theme E3 Graz, Austria, 1999.

[11] Kennedy, M.G., Ahlfeld D.P., Schmidt, D.P., and Tobiason, G.B., "Three Dimensional Modelling for Estimation of Hydraulic Retention Time in a Reservoir", ASCE Journal of Hydraulic Engineering, Volume 132, No. 9, pp. 976-984, USA, September, 2006.

[12] Spalding, D.B., "Mathematical Modelling of Fluid Flow, Heat Transfer and Chemical-Reaction Processes", A Lecture Course, HTS/80/1, Imperial College London, England, 1980.

[13] "User Guide Fluent 6.3", FLUENT Incorporated, Lebnon, N.H.

[14] "User Guide Gambit 2.3", FLUENT Incorporated, Lebnon, N.H.