River Flow 2010 - Dittrich, Koll, Aberle & Geisenhainer (eds) - © 2010 Bundesanstalt für Wasserbau ISBN 978-3-939230-00-7

Modeling flow in curved open channel by a quasi-3D model

A T M Hasan Zobeyer & Peter M. Steffler
Department of Civil and Environmental Engineering, University of Alberta, Edmonton, AB, Canada T6G 2W2

ABSTRACT: A quasi-3D model is developed and tested to simulate the vertical distribution of horizontal velocities in curved open channel. The water surface elevation and depth averaged velocity are modeled by the depth averaged open channel flow model River2D. Then using the depth averaged results as input, the Reynolds averaged Navier-Stokes horizontal momentum (RANS HM) equations are discretized and solved in a generalized coordinate system by a finite difference method with a zero equation turbulence model. Two correction terms are added to the RANS HM equations and modeled such that the mean of the modeled velocity profile at each vertical is equal to the depth averaged velocity. The vertical velocity is neglected and a hydrostatic pressure distribution is assumed. The model results show good agreement with the experiments. The computational time required by the quasi-3D model is less than one order of magnitude as compared to the depth averaged model.

Keywords: Quasi-3D model, Depth averaged model, Open channel flow

1 INTRODUCTION

A good understanding of river hydrodynamics is important for different environmental and engineering flows related to contaminant transport, effect of hydraulic structures, sediment transport and channel geomorphology. Currently numerical models are widely used in modeling river hydrodynamics due to its cost effectiveness over experiments and field measurements. A full 3D computational fluid dynamic (CFD) model with the capability of tracking the free surface and an appropriate turbulence closure model may give very good spatial distribution of all velocity and pressure variables, however its use is often hindered by the consideration of computational time and/or memory in a natural river (Xia and Jin 2007). Different simplifications of 3D CFD models such as hydrostatic pressure assumption with dynamic free surface computation or a rigid-lid assumption for the free surface have been used; however it may be difficult to use an adequately refined discretization with a sophisticated turbulence model (e.g. shear stress transport) with a higher order upwind method due to computational time limitations.

Due to the limitations of 3D CFD models for open channel flows, different depth averaged and Quasi 3D models have been developed and are currently being used. The classical St. Venant equations used for one and two dimensional depth averaged flow simulation in open channels, are derived assuming uniform velocity, hydrostatic pressure and small channel slope (Chaudhry 1993). Therefore these equations can not provide any vertical detail of the flow field and do not include the effects of the non uniform velocity distribution and the non hydrostatic pressure. Boussinesq equations include the effects of the nonhydrostatic pressure and vertical velocity distribution, but they are only applicable for flows with wavelength to depth ratio greater than ten (Steffler and Jin, 1993). They are improvements of the St. Venant equations but do not provide the vertical distribution of longitudinal velocities better than the St. Venant equations. Dressler (1978) attempted to incorporate the bed curvature using curvilinear coordinate in the Euler equations assuming the water surface curvature same as the bed curvature. Hager and Hutter (1984) improved the model assuming a linear variation of flow angle and curvature between the bed and surface. However all these potential flow assumptions can not include the turbulence and rotationality present in open channel flows. Steffler and Jin (1993) developed the vertically averaged and moment equations (VAM) by integrating the RANS equation over the flow depth with a linear approximation for the longitudinal velocity distribution and a quadratic approximation for the vertical velocity and pressure distributions. Multilayer models have been used by different researchers to compute the water surface and the velocity profiles (Lai and Yen 1993, Li and Yu 1996). Xia and Jin (2007) used the moment equations with the layer averaged equation in each layer to improve the multilayer model. They assumed linear profiles of flow variables within a layer.

Recently Zobeyer and Steffler (2009) developed a quasi-3D model for 2 dimensional plane flow by a combination of the depth averaged and the Reynolds averaged Navier-Stokes (RANS) equations in order to simulate the vertical distribution of horizontal velocity. They solved the depth averaged St. Venant or the VAM equations and the RANS *x* momentum equation to obtain a better distribution of horizontal velocity. The water surface elevation and depth averaged velocity were obtained from the St. Venant equations for hydrostatic flow conditions. Otherwise the VAM equations were solved to obtain the vertical velocity and the nonhydrostatic pressure in addition to the depth averaged velocity and water level which were used as input in the RANS *x* momentum equation. A correction was required in the RANS *x* momentum equation such that the mean of the horizontal velocity profile in a vertical becomes equal to the depth averaged velocity at that location. The model was used to simulate the flow over a sill and a free overfall.

Practically it is harder to find a natural river with a straight reach longer than 10 channel width (Leopold and Wolman 1960), therefore the modeling of river bends has drawn special attention from hydraulic engineers and researchers. The characteristics of flow in a river bend are the superelevation, secondary flow and redistribution of longitudinal velocity. To simulate these characteristics, different models ranging from analytical (Johannesson and Parker 1989), quasi-3D (Shimuzi et al. 1990, Jin and Steffler 1993, Ghamry and Steffler 2005) to full 3D (Meselhe and Sotiropoulos 2000, Wilson et al. 2003, Nguyen et al. 2007) have been developed and verified. In this study, the model of Zobeyer and Steffler (2009) is extended for 3D flow and verified to simulate the flow in a channel bend. The water surface and the depth averaged velocity are obtained from the depth averaged finite element model River2D (Steffler and Blackburn 2002) which solves the 2D St. Venant Equations. Then the computed water surface is used as a rigid lid for the solution of the Reynolds horizontal momentum equations to obtain the vertical distribution of the horizontal velocities. The vertical velocity and nonhydrostatic pressure are neglected for this study.

The main purpose of this study is to evaluate the accuracy and computational efficiency of the proposed model. Therefore the simplest modeling techniques such as finite difference discretization, first order upwind discretization, and zero equation turbulence model are used and implemented through Matlab computer program.

2 MATHEMATICAL MODEL

The River2D model solves the depth averaged St. Venant equations in conservation form which can be expressed as:

$$\frac{\partial h}{\partial t}+\frac{\partial q_x}{\partial x}+\frac{\partial q_y}{\partial y}=0; \tag{1}$$

$$\begin{aligned}&\frac{\partial q_x}{\partial t}+\frac{\partial(\bar{u}q_x)}{\partial x}+\frac{\partial(\bar{v}q_x)}{\partial y}+\frac{g}{2}\frac{\partial d^2}{\partial x}\\&=gd(S_{0x}-S_{fx})+\frac{1}{\rho}\frac{\partial}{\partial x}(d\tau_{xx})+\frac{1}{\rho}\frac{\partial}{\partial y}(d\tau_{xy});\end{aligned} \tag{2}$$

$$\begin{aligned}&\frac{\partial q_y}{\partial t}+\frac{\partial(\bar{u}q_y)}{\partial x}+\frac{\partial(\bar{v}q_y)}{\partial y}+\frac{g}{2}\frac{\partial d^2}{\partial y}\\&=gd(S_{0y}-S_{fy})+\frac{1}{\rho}\frac{\partial}{\partial x}(d\tau_{yx})+\frac{1}{\rho}\frac{\partial}{\partial y}(d\tau_{yy});\end{aligned} \tag{3}$$

In the above equations, *d*=water depth, $\bar{u}$ and $\bar{v}$= depth averaged velocity in the *x* and *y* directions, *g*=acceleration due to gravity; q_x and q_y = *x* and *y* component of discharge per unit width; S_{ox} and S_{oy} =bottom slope, S_{fx} and S_{fy} =bed stress, τ_{xx}, τ_{xy}, τ_{yx} and τ_{yy} are the depth averaged turbulent stresses.

Since River2D uses the finite element method where coordinate transformations are done on the element level these equations are left in Cartesian form.

The *x* and *y* momentum RANS equations for incompressible fluid including two momentum correction terms (*X* and *Y*) can be expressed in the non-conservative form after transforming it from the Cartesian to the nonorthogonal bodyfitted coordinate by the partial transformation approach, that is $(x,y,z)\rightarrow(\xi,\eta,\zeta)$ but leaving the velocity components in the Cartesian coordinate,

$$\frac{\partial u}{\partial t} + U\frac{\partial u}{\partial \xi} + V\frac{\partial u}{\partial \eta} + W\frac{\partial u}{\partial \zeta} = -\frac{1}{\rho}\left(\xi_x\frac{\partial p}{\partial \xi} + \eta_x\frac{\partial p}{\partial \eta} + \zeta_x\frac{\partial p}{\partial \zeta}\right) + \frac{1}{\rho}\left(\xi_z\frac{\partial \tau_{xz}}{\partial \xi} + \eta_z\frac{\partial \tau_{xz}}{\partial \eta} + \zeta_z\frac{\partial \tau_{xz}}{\partial \zeta}\right) + X; \quad (4)$$

$$\frac{\partial v}{\partial t} + U\frac{\partial v}{\partial \xi} + V\frac{\partial v}{\partial \eta} + W\frac{\partial v}{\partial \zeta} = -\frac{1}{\rho}\left(\xi_y\frac{\partial p}{\partial \xi} + \eta_y\frac{\partial p}{\partial \eta} + \zeta_y\frac{\partial p}{\partial \zeta}\right) + \frac{1}{\rho}\left(\xi_z\frac{\partial \tau_{yz}}{\partial \xi} + \eta_z\frac{\partial \tau_{yz}}{\partial \eta} + \zeta_z\frac{\partial \tau_{yz}}{\partial \zeta}\right) + Y; \quad (5)$$

where *u, v,* and *w* = Cartesian velocity components in the *x*, *y* and *z* directions respectively, *U*, *V* and *W*= contravariant velocity components ($U = u\xi_x + u\xi_y + w\xi_z$ $V = u\eta_x + v\eta_y + w\eta_z$, $W = u\zeta_x + v\zeta_y + w\zeta_z$), *g* =acceleration due to gravity, ρ= density of water, p=pressure, τ_{xz}, τ_{yz}=shear stress, $\xi_x, \xi_y, \xi_z, \eta_x, \eta_y, \eta_z, \zeta_x, \zeta_y$ and ζ_z=matrices of transformation. The ζ direction coincides with the vertical direction while ξ and η follows the bed and water surface profile and so ξ_z, η_z=0.

p can be split into the hydrostatic (p_h) and the nonhydrostatic (p') components as $p = p_h + p' = \rho g(h-z) + p'$; where *h*= water surface elevation, *z* = elevation of any point from a reference level. The vertical velocity *w* and nonhydrostatic pressure p' are neglected in this study. The normal stresses are also neglected.

In the derivation of the depth averaged equations, uniform velocity and hydrostatic pressure conditions are assumed and the bed stress is computed from the depth averaged velocity. On the other hand, the quasi-3D model equations include the nonuniform velocity and in general can include the nonhydrostatic pressure and bed stress is computed from the near bed velocity. Therefore the correction terms are added to the RANS HM equations in order to make the two sets of equation consistent both mathematically and numerically such that the mean of the computed velocity profile in any vertical becomes equal to the depth averaged velocity in that vertical.

NUMERICAL SCHEME

The River2D model uses fully implicit Streamlined Upwind Petrov-Galerkin (SUPG) finite element schemes known as Characteristics dissipative Galerkin (CDG) method (Hicks and Steffler 1992) to discretize the St. Venant equations. For stability upwinding is achieved by weight functions based on both characteristics scaled to their absolute values. The discretization of the computational domain is performed by triangular elements. For the subcritical upstream boundary, a known discharge is specified. For the subcritical downstream boundary, a known water depth is used. Sidewall boundaries are treated as a free slip boundary with no flow perpendicular to the wall. Details of the model can be found in Steffler and Blackburn (2002).

The computational mesh for the quasi-3D model could be generated by using the 2D triangular mesh in plan and dividing the flow depth in several layers and the nodal values of water depth can directly be used to define the upper boundary for the discrete 3D domain. However, the quasi-3D model is modeled by a finite difference method and so the locations of the verticals are different from the nodes of the depth averaged model. Therefore the water depth and depth averaged velocity are interpolated at the locations of the verticals of the quasi-3D model by the post-processor of River2D. The convective terms are discretized by the first order upwind method. The pressure gradient terms are approximated by a centered difference method. The gradient terms in the ξ and η directions are discretized explicitly with an implicit coupling in the ζ direction which allows to solve each vertical independently using the values of the neighboring verticals from the previous time step and avoids the solution of a large matrix, rather a hexadiagonal matrix of size 2*m* is solved for each vertical, where m is the number of node in the vertical.

For the *x* and *y* momentum equations vertical profiles of *u* and *v* are specified as the upstream boundary condition. For the side walls free-slip boundary conditions are used. They are implemented by setting the contravariant velocity component U_{wall} equal to *U* at the first interior point while the condition of no flux perpendicular to the solid wall is applied by setting the contravariant velocity component V_{wall} to zero. The law of the wall is used as the bottom boundary condition and a zero shear stress is specified at the water surface. Logarithmic velocity profiles based on depth averaged velocity are specified as the initial condition at each vertical. The shear stress is computed by the eddy viscosity hypothesis. The eddy viscosity (ν_t) is given by

$$\nu_t = \kappa u_*(z - z_b)\left(1 - \frac{z - z_b}{d}\right)$$

where κ = Von Karman's constant (=0.41), u_*= shear velocity, z_b = bed elevation, d = depth of flow.

MODELING OF THE CORRECTION TERMS

In order to model the correction terms, approximate equations have been developed as follows.

Let us consider equation 4. Considering the numerical scheme for the quasi-3D model, equation 4 can be written as

$$\frac{u^{n+1}-u^n}{\Delta t}+\left(U\frac{\partial u}{\partial \xi}\right)^n+\left(V\frac{\partial u}{\partial \eta}\right)^n+W^n\left(\frac{\partial u}{\partial \zeta}\right)^{n+1}= -\frac{1}{\rho}\left(\xi_x\frac{\partial p}{\partial \xi}+\eta_x\frac{\partial p}{\partial \eta}+\zeta_x\frac{\partial p}{\partial \zeta}\right)^n+\frac{1}{\rho}\left(\zeta_z\frac{\partial \tau_{xz}}{\partial \zeta}\right)^{n+1}+X; \tag{6}$$

With a guessed value of the correction term say X_g, equation 4 can be written as

$$\frac{u_g^{n+1}-u^n}{\Delta t}+\left(U\frac{\partial u}{\partial \xi}\right)^n+\left(V\frac{\partial u}{\partial \eta}\right)^n+W^n\left(\frac{\partial u}{\partial \zeta}\right)_g^{n+1}= -\left(\frac{1}{\rho}(\xi_x\frac{\partial p}{\partial \xi}+\eta_x\frac{\partial p}{\partial \eta}+\zeta_x\frac{\partial p}{\partial \zeta})\right)^n+\frac{1}{\rho}\left(\zeta_z\frac{\partial \tau_{xz}}{\partial \zeta}\right)_g^{n+1}+X_g; \tag{7}$$

where the superscripts n and n+1 refer to the solution at the current and next time level, the subscript g refers to the values computed using X_g and Δt is the discrete time step.

Subtracting Equation (7) from Equation (6), assuming that in the vertical direction the respective velocity and shear stress gradients in both equations are the same and integrating the remaining terms over the flow depth give the following relationship

$$X=\frac{\bar{u}-\bar{u}_g^{n+1}}{\Delta t}+X_g$$

Because of the approximations in this derivation more than one iteration may be required to obtain the appropriate value of the correction term. Therefore the above expression can be written as

$$X_{k+1}=\frac{\bar{u}-\bar{u}_k^{n+1}}{\Delta t}+X_k \tag{8}$$

where subscript k refers to the iteration level for the correction term.

Similar relationship can be developed for Y as

$$Y_{k+1}=\frac{\bar{v}-\bar{v}_k^{n+1}}{\Delta t}+Y_k \tag{9}$$

The iteration procedure is described in the next section.

3 SOLUTION PROCEDURE

In the River2D model, a time marching procedure is used to obtain the steady state solution. At the beginning of the simulation relatively a small time step is used. As the solution advances, time step size is increased based on relative solution change to expedite the convergence speed. The nonlinear discretized equations are linearized by the Newton-Raphson method. The Jacobian matrix is computed numerically and solved by a direct solver.

Once the steady state water level and depth averaged velocity are obtained, they are used in the Quasi-3D model with the water surface as a fixed lid. A time marching procedure is also used to obtain the steady state solution in the quasi-3D model. For each vertical a local time step is computed from a fixed Courant number (Cr),

$$\Delta t_i=\frac{Cr}{\max(U_i,V_i)} \tag{10}$$

where i refers to a particular vertical, U_i, V_i are the contravariant velocities at all nodes of the vertical. At the beginning of the simulation, the set of the discretized horizontal momentum equations of each vertical is solved for one time step after setting the values of X and Y equal to zero. During this simulation if the mean of the computed u and/or v velocity profile is not within a specified tolerance of their corresponding depth averaged velocities, Equation 8 and/or 9 is used to compute X and Y and the velocity profiles are recomputed. This procedure is repeated until the mean of the computed velocity profiles are within the specified tolerance of their depth averaged velocities. Once all the verticals are solved for one time step, the procedure is advanced to the new time level using the values of X and Y computed from the previous time step and the above procedure is repeated. This solution procedure continues until the solution converges to the steady state. The convergence to the final steady state solution is assessed by the following criteria:

$$\sqrt{\frac{\sum(\delta\phi)^2}{\sum\phi^2}}\leq \text{tolerance}$$

where ϕ=velocity, $\delta\phi$= difference between velocity at two successive time steps. A tolerance of 10^{-6} is used for the convergence criteria.

4 RESULTS AND DISCUSSION

The experimental data of Rozovskii (1957) have widely been used to validate the capability of different numerical models to simulate the flow field and predict the secondary flows in a bend that exhibits strongly three dimensional characteristics. Rozovskii performed his experiments in a 180° curved rectangular flume with a very strong degree of curvature ($Rc/2b$=1, Rc= radius of the centre of the bend, b= half channel width) The secondary velocities produced in his experiments were very strong due to the sharp curvature of the channel. The results of run 1 are presented here to test the numerical predictions of the present model. Rozovskii's channel consisted of a 6-m-long straight approach and a 3-m-long straight exit with a 180° bend (Figure1). The width of the channel was 0.8 m, and the radius of the channel centerline was 0.8 m for the circular reach. The channel bed was horizontal and smooth with a Chezy coefficient of 60 $m^{1/2}$/s. The inflow discharge was 0.0123 m^3/s with the flow depth at the entrance equal to 0.06 m. This gives a mean velocity at entrance equal to 0.256 m/s.

For simulating the flow in the curve bend by the current model, first the River2D model is used to compute the depth average velocity and water surface elevation. The simulation is performed using a finite element mesh composed of 2754 triangular elements and 1685 nodes. At the upstream boundary a steady inflow discharge of 0.0123 m3/s was used. For the downstream boundary a constant depth equal to 0.057m was used. Based on the measured Chezy coefficient the corresponding roughness height is estimated to be 0.0003 m. Once the steady state simulation was performed, the post processor of River2D was used to extract the depth averaged velocity and water level at predefined locations which are eventually the locations of the verticals in the 3D model as mentioned earlier. A total of 2380 verticals are used with 140 in the streamwise direction and 17 in the transverse direction. For the streamwise direction a denser spacing is used in the bend region with a coarser spacing in the straight reaches, for the transverse direction equal spacing is used. In the vertical direction 15 nodes are used for each vertical giving a total of 35700 computational nodes.

The computed longitudinal velocity (u_L) profiles are shown in Figure 2. The computed profiles are compared with the experimental results. Generally all the profiles show good agreement with the experimental results. At section 6 that is located at 100 degree of the bend, the profiles at the center (Y/b=0), outer side (Y/b=0.5) and inner side (Y/b=-0.5) of the bend matches very well with the experimental results. At section 8 (144 degree) the quasi-3D model predicts the center (Y/b=0) and outer profiles (Y/b=0.5) very well. In the inner side (Y/b=-0.5) the model overpredicts the velocity, however it predicts the shape of the profile quite well. Just downstream of the exit (Section 10), the model undepredicts the velocity in the outer side (Y/b=0.50) and slightly overpredicts velocity near the surface in the inner side (Y/b=-0.5), however the model predicts the shape of the profiles reasonably well.

The simulated transverse velocity (u_T) profiles are compared with the experimental results and shown in Figure 3. At section 6 the simulated profiles agree well with the experimental results in the center (Y/b=0.0) and outer side (Y/b=0.5). At the inner side (Y/b=-0.5) the model slightly underpredicts the near surface velocity. At section 8 the model can reproduce the experimental results quite well. At section 10, the model slightly overpredicts the velocity at the center and inner side of the bend and overpredicts the near bed velocity in the outer side (Y/b=0.5). Therefore the secondary current has been simulated very well.

Generally, this model can properly simulate both the longitudinal and transverse velocity profiles with some overprediction in the inner side and underprediction in the outer side near the exit for longitudinal velocity. This might happen because the modeled profiles were corrected to give a mean equal to the depth averaged velocity and the depth averaged model does not consider the lateral momentum exchange due to non uniform longitudinal and transverse velocity which is responsible for the flow shifting from inner towards the outer bank. However, once the quasi-3D model results are obtained, the effect of nonuniform velocity and bed stress could be incorporated into the depth averaged model. This would model the flow shifting in the depth averaged computation. Then the updated water level and depth averaged velocity can be used in the quasi-3D model.

5 COMPARISON OF COMPUTATIONAL EFFORT

To get an estimate of the computational efficiency of the current quasi-3D model, a comparison between the River2D model and the quasi-3D model in terms of computational time required to obtain the steady state solutions is made. The computer programs of the River2D model are written in the C programming language and the computational time for the steady state depth averaged modeling required by the River2D model is approximately 30 seconds. The computer programs of the quasi-3D model are written in Matlab and the computational time is approximately 180 seconds. This indicates that the new model requires less than one order of magnitude of the computational time with more than one order of magnitude of computational nodes as compared to the River2D model.

6 CONCLUSION

A quasi-3D model is verified by modeling the flow in a curved open channel. The water surface and depth averaged velocity are obtained from the 2D depth averaged model River2D. Using this information, the RANS horizontal momentum equations are solved in a generalized coordinate by a finite difference method neglecting the vertical velocity and the nonhydrostatic pressure. Two correction terms are added to the momentum equations and modeled such that the mean of the computed velocity profile at any vertical is equal to the corresponding depth averaged velocity. The model results when compared with the experimental results showed reasonably good agreement. The computational efficiency of the 3D model is also quite satisfactory. Further improvements of the model are needed by incorporating the effects of nonuniform velocity computed by the quasi-3D model into the depth averaged model. Then the velocity profiles can be recomputed using updated depth averaged velocity and water surface. Also the effects of vertical velocity and non hydrostatic pressure together with higher order upwind methods and better turbulence models on the accuracy and efficiency of the quasi-3D model need to be investigated.

REFERENCES

Chaudhry, M. H. 1993. Open channel flow. Prentice-Hall, Englewood Cliffs, N.J.

Dressler, R. F., 1978. New nonlinear shallow-flow equations with curvature. J. of Hydraul. Res., 16(3) 205-220.

Ghamry, H. K., and Steffler, P. M. 2005. Two dimensional vertically averaged and moment equations for rapidly varied flows. J. Hydraul.Res.43(1), 44-55

Hager, W. H., Hutter, K. 1984. Approximate treatment of plane channel flow. Acta Mechanica, 51, 31-48.

Hicks, F. E., Steffler, P. M. 1992. Characteristic dissipative Galerkin scheme for open-channel flow. J. of Hydraul. Eng., 118 (2), 337-352.

Jin, Y.-C., Steffler, P. M. 1993. Predicting flow in curved open channel by depth averaged method. J. of Hydraul. Eng., 119(1), 109-124.

Johannesson, H. Parker, G. 1989. Secondary Flowin Mildly Sinuous Channel. J. Hydraul. Eng., ASCE115(3),289–308.

Lai, C. J., Yen, C. W. 1993. Turbulent free surface flow simulation using a multilayer model. Int. J. Numer. Meth. Fluids, 16(11), 1007-1025.

Leopold, L.B. and Wolman, M.G. (1960). River Meanders. Bull. Geol. Soc. Am. 71(6), 769–794.

Li, C. W., Yu, T. S. 1996. Numerical investigation of turbulent shallow recirculating flow by a quasi-three-dimensional κ-ε model. Int. J. Numer. Meth. Fluids, 23(5), 485-501.

Meselhe, E. A., Sotiropoulos, F. 2000. Three-dimensional numerical model for open-channels with free-surface variations. J.Hydraul.Res., 38, 115–121.

Nguyen, V. TH., Nestmann, F., Scheuerlein, H. 2007. Three-dimensional computation of turbulent flow in meandering channels and rivers. J.Hydraul.Res., 45, 595–609.

Rozovskii, I. L. 1961. Flow of water in bends of open channels. Acad. Sci. Ukrainian S. S. R., Translated from Russian, Israel Program for Science Translation, 1–233.

Shimizu, Y., Yamaguchi, H., and Itakura, T. 1990. Three-dimensional computation of flow and bed deformation. J. Hydraul. Eng., 116(9) 1090–1108.

Steffler, P., Blackburn, J. 2002. River2D: Two-Dimensional Depth-Averaged Model of River Hydrodynamics and Fish habitat, Introduction to Depth Averaged Modeling and User's Manual. University of Alberta, Canada.

Steffler, P. M., Jin, Y. 1993. Depth-averaged and moment equations for moderately shallow free surface flow. J. of Hydraul. Res., 31(1), 5-17.

Wilson, C. A. M. E., Boxall, J. B., Guymer, I., Olsen, N. R. B. 2003. Validation of a three-dimensional numerical code in the simulation of pseudo-natural meandering flows. J. of Hydraul. Eng., 129 (10), 758-768.

Xia, C., Jin, Y. -C. 2006. Multilayer averaged and moment equations for one-dimensional open-channel flows. J. of Hydraul. Eng., 132 (8), 839-849.

Zobeyer, H., Steffler, P. M. 2009. A quasi 3D numerical model for open channel flow. 33rd IAHR Congress, Vancouver, 921-928.

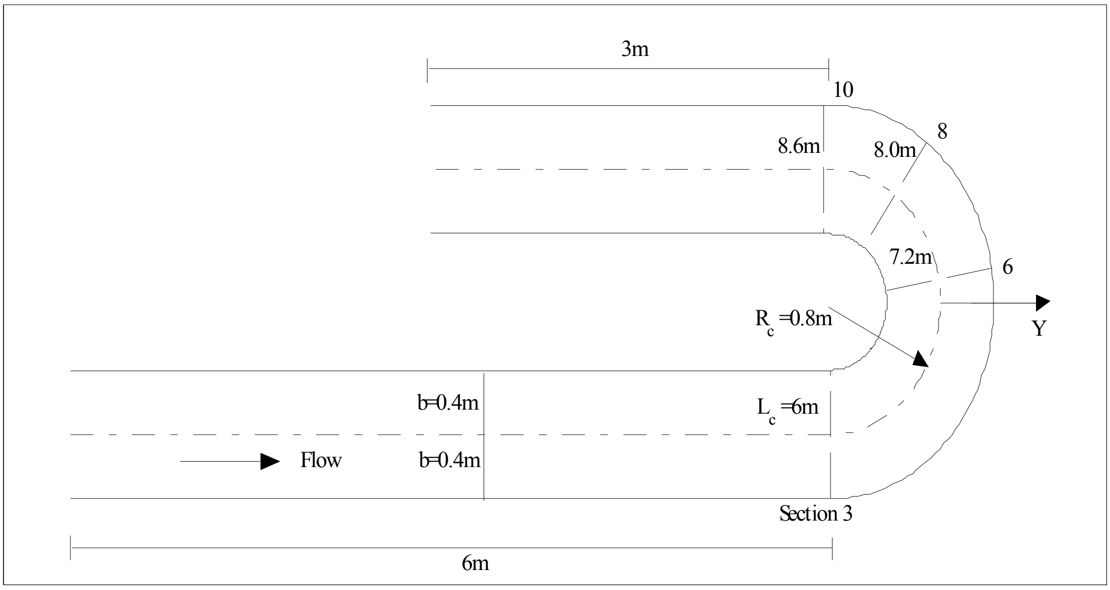

Figure 1 Layout of Rozovskii's experiment

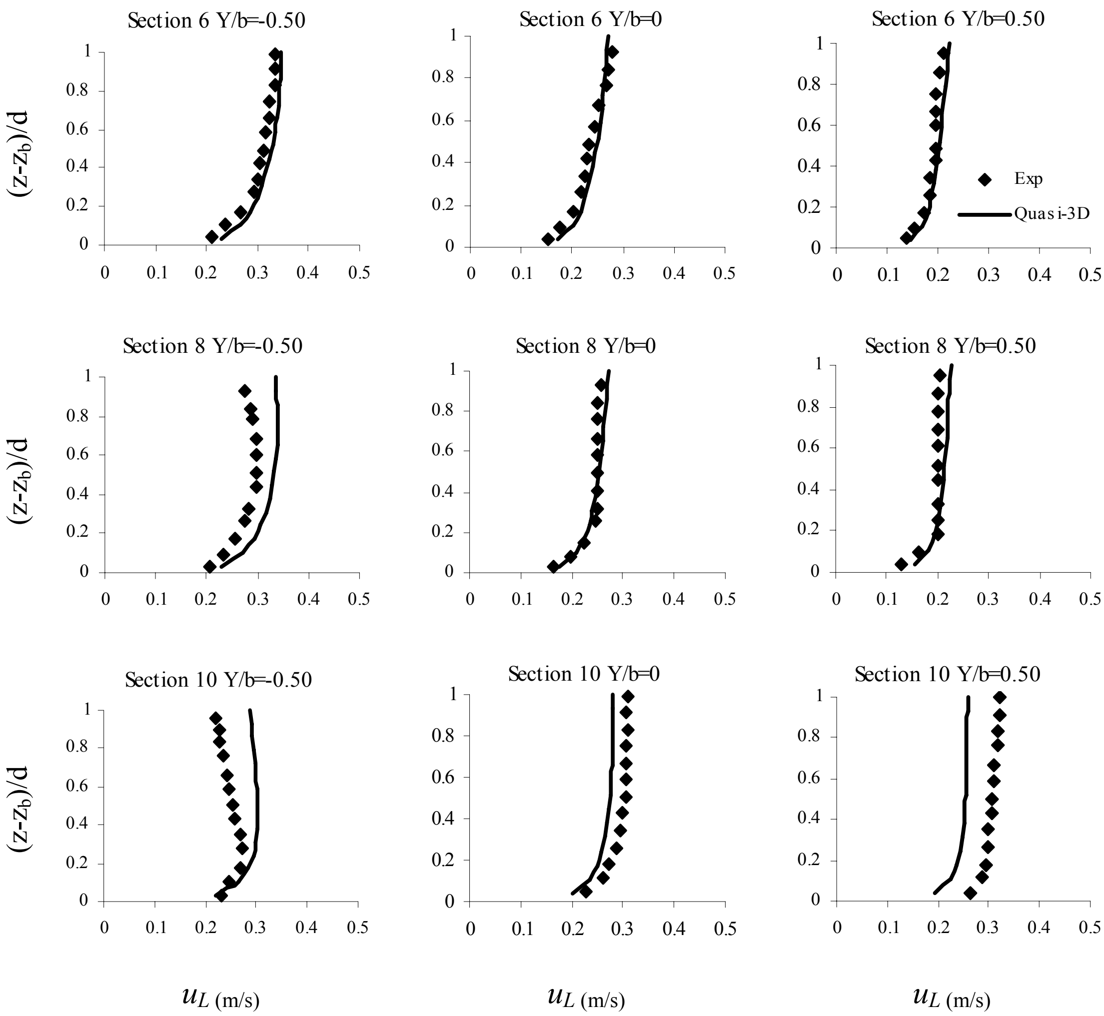

Figure 2. Comparison of longitudinal velocity (u_L) profiles

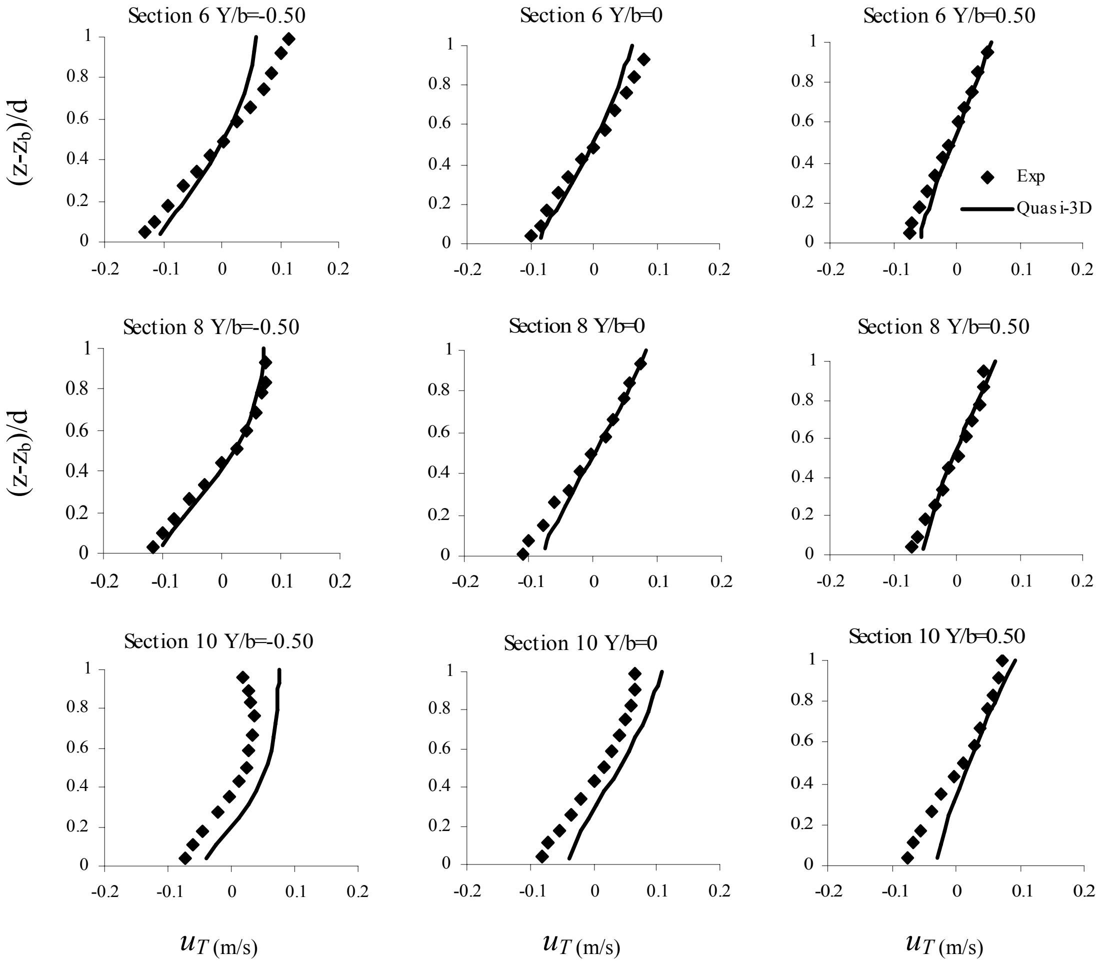

Figure 3. Comparison of transverse velocity (u_T) profiles