Numerical investigation of Velocity Field in Dividing Open-Channel Flow

BAHAREH PIRZADEH[*], HAMID SHAMLOO [**]
Civil Engineering Department
K.N Toosi University of Technology
No.1346 Valiasr Street, Tehran
IRAN

Abstract: This paper provides details of application of *FLUENT*-2D&3D software in simulation of lateral intake flows. Comparisons have been made between numerical results and measured experimental velocities for a lateral intake. Comparisons indicate that the 2D simulating captures most experimental trends with reasonable agreement and 3D simulating results have also good accuracy.

Key-Words: Open channel, Lateral Intake, Turbulence, Numerical modeling, Fluent

1 Introduction

Dividing flows in open channels are commonly encountered in hydraulic engineering systems. Flows through lateral intakes adjoining rivers and canals are turbulent. Transverse pressure gradients in the vicinity of the intake induce region of mean-velocity gradients, depth-varying surface of flow division and separation, vortices, and zone of flow reversal.

According to the physical modeling results, Taylor (1944) recommended that water depth variation for a wide range of discharge ratios and Froude numbers in vicinity of the intake entrance is around 2% of the maximum depth.

Neary and Odgaard (1993) examined the effects of bed roughness on the three-dimensional (3D) structure of dividing flows. For low Froude numbers, they presented detailed velocity-vector and particle-trace plots in the initial part of the separation zone. Their measurements indicated no depth variations in the junction (Neary and Odgaard 1993). Further, Neary et al. (1999) numerically investigated the lateral-intake inflows using 3D two-equation turbulence models without considering the water surface effects. The implication of the flow patterns on sediment transport shows that the branch channel will receive a relatively large amount of bed load because a larger portion of near bed flow is diverted.

Shettar and Murthy (1996) deployed depth-averaged mean flow equations associated by the standard k-ε model. Results obtained from their model for an open channel T-junction showed that for discharge ratio 0.52, a good agreement between measurements and model results can be obtained. Chen and Lian (1992) also performed the same simulation, and the results found to be in reasonable agreement with measurement only for small discharge ratio.

Weber et al. (2001) performed an extensive experimental study of combining flows in a 90° open channel for the purpose of providing a very broad data set comprising three velocity components, turbulence stresses, and water surface mappings.

Huang et al. (2002) provided a comprehensive numerical study of combining flows in open-channel junctions using the 3D turbulence model and validated the model by using the detailed test data of Weber et al. (2001).

Recently, assuming the velocities to be nearly uniform, Hsu et al. (2002) presented a depth-discharge relationship and energy-loss coefficient for a sub-critical, equal-width, right-angled dividing flow over a horizontal bed in a narrow aspect ratio channel.

Ramamurthy et al. (2007) presented experimental data related to 3D mean velocity components and water surface profiles for dividing flows in open channels. The data set presented in this paper is composed of water surface mappings and 3D velocity distributions in the vicinity of the channel junction region.

In the current study an attempt has been made to model fluid flow through lateral intake using *FLUENT* software and numerical 2D&3D results for velocity in main channel and intake were compared with measures velocities of Shettar and Murthy (1996).

2 Experimental Investigation

Velocity profiles obtained from the current numerical model were compared with laboratory experiment results performed by Shettar and Murthy (1996); in their experimental set-up, the main channel was 6m long and the intake was 3.0-m long, fitted at its midpoint. The width of both channels was 0.3-m; the bed slope was zero and 0.25-m deep. The channel bed was finished with smooth cement plaster and walls were built from Perspex sheets.

In Shettar and Murthy's experiments (1996) the discharge ration was 0.52 and the Froude number at inlet was 0.54 so the velocity at inlet was 0.85m/s. They presented depth-averaged mean velocity profiles in different sections across the main and intake channel.

3 Numerical Model Description

FLUENT is the CFD solver for choice for complex flow ranging from incompressible (transonic) to highly compressible (supersonic and hypersonic) flows. Providing multiple choices of solver option, combined with a convergence-enhancing multi-grid method, *FLUENT* delivers optimum solution efficiency and accuracy for a wide range of speed regimes. The wealth of physical models in *FLUENT* allows you to accurately predict laminar and turbulent flows, various modes of heat transfer, chemical reactions, multiphase flows, and other phenomena with complete mesh flexibility and solution-based mesh adoption [2].

FLUENT solves governing equations sequentially using the control volume method. The governing equations are integrated over each control volume to construct discrete algebraic equations for dependent variables. These discrete equations are linearized using an implicit method. As the governing equations are nonlinear and coupled, iterations are needed to achieve a converged solution.

Conservative form of the Navier-stokes equations using the finite volume method on structured/ orthogonal, Cartesian coordinates grid system.

Turbulent flows can be simulated in *FLUENT* using the standard K-ε, LES, RNG, or the Reynolds-stress (RSM) closure schemes. The model optimizes computational efficiency by allowing the user to choose between various spatial (Second-order upwind, third-order, QUICK) discritization scheme.

The under-relaxation factors are chosen between 0.2 and 0.5. The small values of the under-relaxation factors are required for the stability of the solution of this interpolation scheme [2].

Turbulent stresses in Reynolds-averaged equations can be closed using any of several exiting turbulence models. No single turbulence model is accepted universally for solving all closes of problems but each model has certain advantages over the others depending on the type and the nature of the flow field to be simulated and the desired accuracy of results [5].

The simplest and most widely used two-equation turbulence model is the k-ε model that solves two separate equations to allow the turbulent kinetic energy and dissipation rate to be independently determined.

The turbulence kinetic energy, *k*, is modeled as:

$$\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\frac{\nu_T}{\sigma_k}\frac{\partial k}{\partial x_j}\right) + P_k - \varepsilon \qquad (1)$$

Where P_k is given by:

$$P_k = \nu_T \frac{\partial U_j}{\partial x_i}\left(\frac{\partial U_j}{\partial x_i} + \frac{\partial U_i}{\partial x_j}\right) \qquad (2)$$

$$\nu_T = c_\mu \frac{k}{\varepsilon^2} \qquad (3)$$

The dissipation of k is denoted ε, and modeled as:

$$\frac{\partial \varepsilon}{\partial t} + U_j \frac{\partial \varepsilon}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\frac{\nu_T}{\sigma_k}\frac{\partial \varepsilon}{\partial x_j}\right) + C_{\varepsilon 1}\frac{\varepsilon}{k}P_k + C_{\varepsilon 2}\frac{\varepsilon^2}{k} \qquad (4)$$

The constants in the k-ε model have the following values: cm=0.09, $c_{\varepsilon 1}$=1.44, $c_{\varepsilon 2}$=1.92, σ_k=1.0 and σ_ε=1.30

We used second order upwind discritization scheme for Momentum, Turbulent kinetic energy and turbulent dissipation rate; used body force weighted discritization scheme for Pressure and PISO algorithm for Pressure-Velocity Coupling Method. Also the standard k-ε model have used in the present study as used by Shettar and Murthy (1996).

4 Governing equations

The governing equations of fluid flow in rivers and channels are generally based on three-dimensional Reynoldes averaged equations for incompressible free surface unsteady turbulent flows.

The governing equations of fluid flow in rivers and channels are generally based on three-dimensional Reynoldes averaged equations for incompressible free surface unsteady turbulent flows as follows [2]:

$$\frac{\partial U_i}{\partial t}+U_j\frac{\partial U_i}{\partial x_j}=\frac{1}{\rho}\frac{\partial}{\partial x_j}\left[\left(-P+\frac{2}{3}k\right)\delta_{ij}+\nu_T\left(\frac{\partial U_i}{\partial x_J}+\frac{\partial U_j}{\partial x_i}\right)\right] \quad (5)$$

There are basically five terms: a transient term and a convective term on the left side of the equation. On the right side of the equation there is a pressure/kinetic term, a diffusive term and a stress term.

In the current study, it is assumed that the density of water is constant through the computational domain. The governing differential equations of mass and momentum balance for unsteady free surface flow can be expressed as [1,2]:

$$\frac{\partial u_i}{\partial x_i}=0 \quad (6)$$

$$\frac{\partial u_i}{\partial t}+u_j\frac{\partial u_i}{\partial x_j}=-\frac{1}{\rho}\frac{\partial P}{\partial x_i}+g_{xi}+\upsilon\nabla^2 u_i \quad (7)$$

Where t=time; u_i is the velocity in the x_i direction; P is the pressure; ν is the molecular viscosity; g_{xi} is the gravitational acceleration in the x_i direction, and ρ is the density of flow.

As in the current study, only the steady state condition has been considered, therefore equation (10) to (11) incorporate appropriate initial and boundary conditions deployed to achieve equilibrium conditions.

5 Boundary conditions

Appropriate condition must be specified at domain boundaries depending on the nature of the flow. In simulation performed in the present study, velocity inlet boundary condition is specified, and set to 0.85 m/s for comparing velocity across the main channel and branch corresponding to the Shettar & Murthy Experiments.

Outflow boundary condition used for two outlets for all of runs, increased the main and branch channel length therefore sufficient distance is provided between the junction and two outlets to ensure that the flow returned to the undisturbed pattern. Fig.(1) represents the layout of the simulated main channel and intake.

Discharge ratios $R=Q_b/Q$ equal to 0.52 (as used by Shettar and Murthy-1996) was used. The no-slip boundary condition is specified to set the velocity to be zero at the solid boundaries and walls and bed assumed to be smooth.

In 3D simulations performed in the present study, two separate inlets for air and water are specified. At each inlet, uniform distributions are given for all of dependent variables. Two separate outflows for air and water are specified at the two ends. At the top surface above the air, zero normal velocity and zero normal gradients of all variables are applied by defining a symmetric boundary condition.

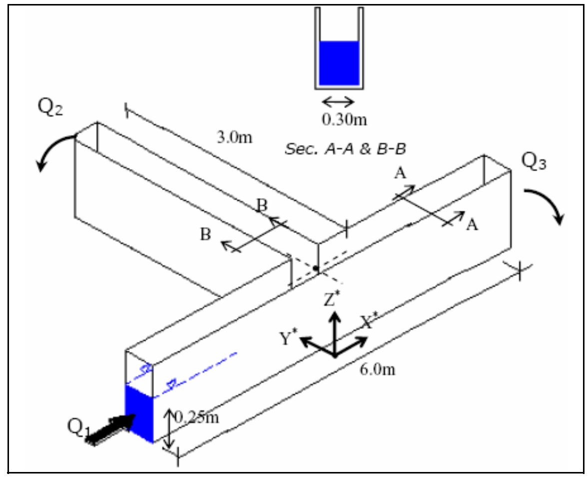

Fig1. Layout of main channel and intake

It is also important to establish that grid-independent results have been obtained. The grid structure must be fine enough especially near the wall boundaries and the junction, which is the region of rapid variation. Various flow computational trials have been carried out with different number of grids in x and y directions. It was found that results are independent of grid size, if at least 3500 nodes are used in 2D simulating and 253120 nodes in 3D simulating. Computational mesh is shown in Fig.2.

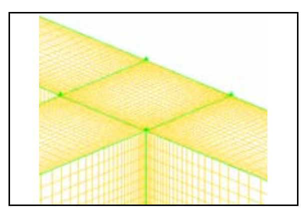

Fig.2. Computational geometry and grid

7 Results and Discussions

In this paper numerical investigations are performed for evolution of the ability of an available 2D&3D flow solver to cop with the fully turbulent flow in a T-junction.

In the Figs.(3) to (8) results of the numerical model are compared with experimental depth averaged velocity profiles in the main channel and In the Figs.(9) to (12) are compared in the branch channel. From these figures, it can be concluded that 2D-results generally have reasonable agreement with measured ones, but at some sections the computed results do not agree very well with those measured, Which might be partly due to the three dimensional effects. Further, Shettar and Murthy (1996) presented depth-averaged mean flow velocities but in the current 2D-study surface velocities have been used.

3D-results of numerical modeling represent a better agreement than results of 2D-modeling indicating the effect of turbulence in the region with 3D-complex features. It is found that the *FLUENT* is an effective tool for predicting flow pattern of the complex flow in lateral intakes.

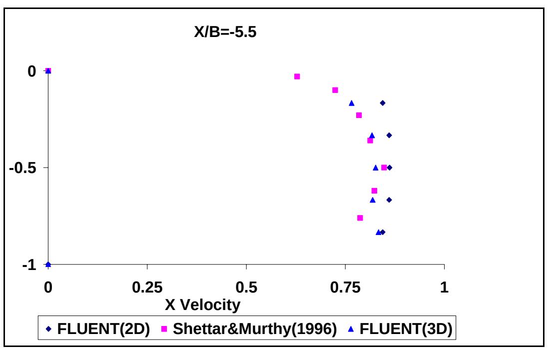

Fig3. X-Velocity Profile in the Main channel (X*=-5.50)

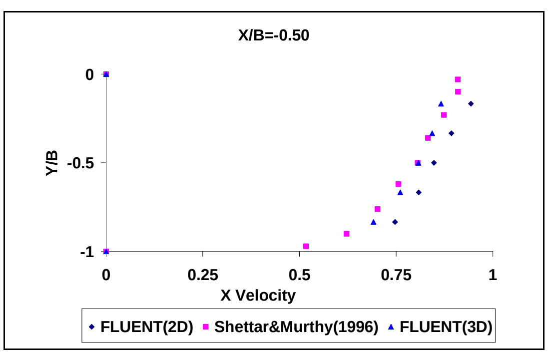

Fig4. X-Velocity Profile in the Main channel (X*=-0.50)

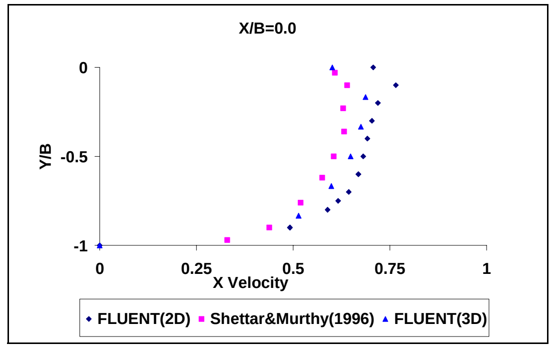

Fig5. X-Velocity Profile in the Main channel (X*=0.0)

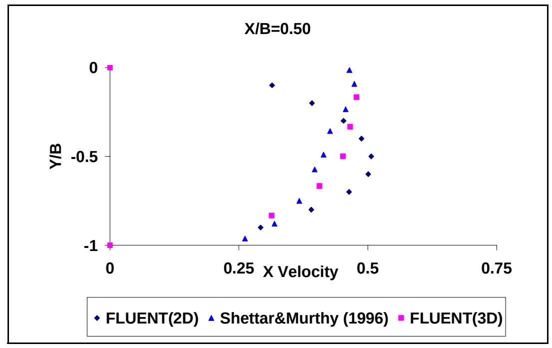

Fig6. X-Velocity Profile in the Main channel (X*=0.50)

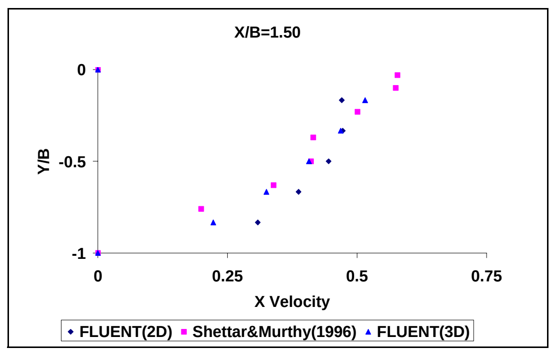

Fig7. X-Velocity Profile in the Main channel (X*=1.50)

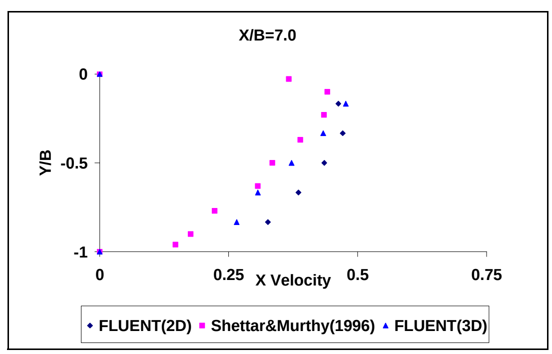

Fig8. X-Velocity Profile in the Main channel (X*=7.0)

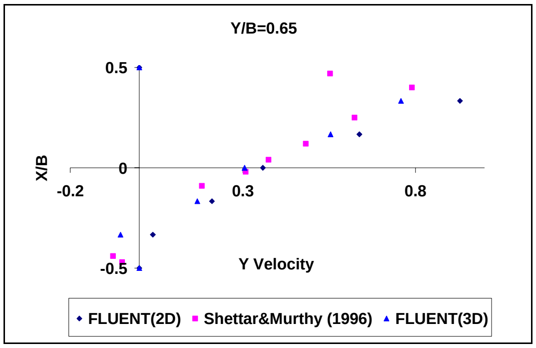

Fig9. Y-Velocity Profile in the Branch channel(Y*=0.65)

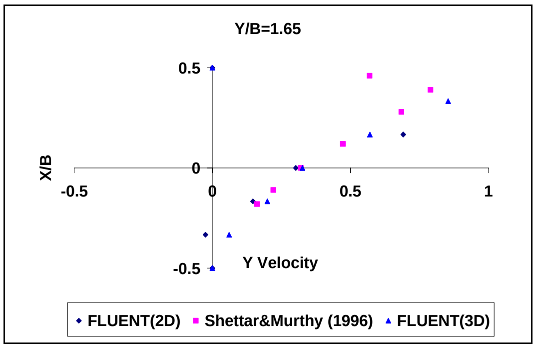

Fig10. Y-Velocity Profile in the Branch channel(Y*=1.65)

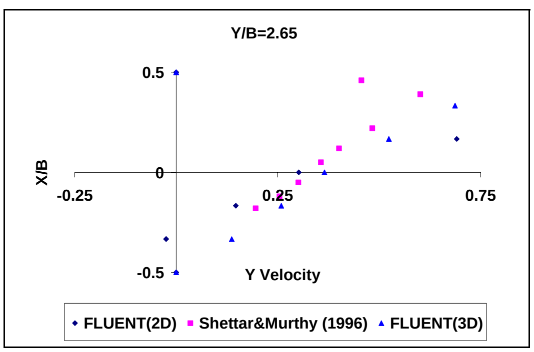

Fig11. Y-Velocity Profile in the Branch channel (Y*=2.65)

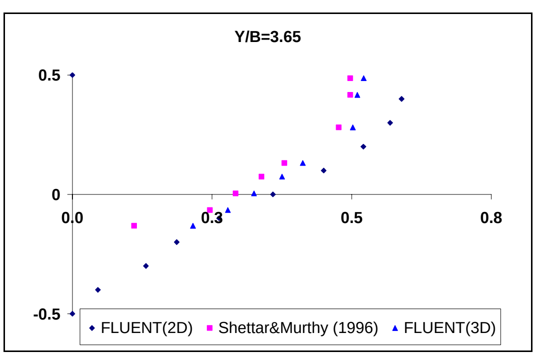

Fig11. Y-Velocity Profile in the Branch channel (Y*=3.65)

References

[1] CHEN H. and LIAN G. " The numerical computation of turbulent flow in tee-junctions.", *Journal of Hydrodynamics*, Series B, No.3, 1992, pp19-25

[2] FLUENT user's guide manual-version 6.1., Fluent Incorporated, N.H., 2003

[3] HUANG J. ,WEBER L.J., LAI Y.G., "Three dimensional numerical study of flows in open channel junctions", *Journal of Hydraulic Engineering*, Vol.128, No.3, 2002, pp268-280

[4] HSU C-C, TANG C-J, LEE W-J and SHIEH M-Y, "Sub-critical 90° equal-width open-channel dividing flow.", Journal of Hydraulic Engineering, Vol.128, No.7, 2002, pp.716-720

[5] NEARY V.S. and ODGAARD A.J., "Three-dimensional flow structure at open-channel diversions.", *Journal of Hydraulic Engineering*,Vol.119, No.11, 1993,pp.1223-1230

[6] NEARY V.S., SOTIROPOULOUS F. and ODGAARD A.J., "Three-dimensional numerical model of lateral-intake inflows.", *Journal of Hydraulic Engineering*, Vol.125, No.2, 1999,pp.126-140

[7] KASTHURI,B. and PUNDARIKHANTHAN N.V. , "Discussion on separation zone at open channel junction.", *Journal of Hydraulic Engineering*, Vol.113, No.4, 1987,pp.543-548

[8] RAMAMURTHY A.S., JUNYING QU and DIEP VO. "Numerical and Experimental study of Dividing open-channels flows", *Journal of Hydraulic Engineering*, Vol.133, No.10, 2007,pp.1135-1144

[9] SHETTAR A.S. and MURTHY K.K.," A Numerical study of division flow in open channels.", *Journal of Hydraulic Research*, Vol.34, No.5, 1996,pp.651-675

[10] TAYLOR, E.H. "Flow characteristics at rectangular open-channel junctions." *Trans. ASCE*, 109, 1944, pp. 893–902.