Sindh Univ. Res. Jour. (Sci. Ser.) Vol. 47 (4) 713-716 (2015)

SINDH UNIVERSITY RESEARCH JOURNAL (SCIENCE SERIES)

Transient Analysis of Two–Dimensional Free Surface Flow Through Hydraulic Structure

A. A. MAHESSAR, Z. A. K. BALOCH*, A. L. QURESHI. A. BALOCH[++]**

US Pakistan Center of Advanced Studies in Water, Institute of Water Resources Engineering and Management, Mehran University Engineering and Technology, Jamshoro, Pakistan.

Received 13th March 2015 and Revised 22nd August 2015

Abstract: Computation of fluid flow in contraction and expansion channel located at upstream of channel is presented in this paper. A two–dimensional model is reported for computing incompressible flow of Newtonian fluid adopting Navier–Stokes equations. A semi–implicit Taylor–Gakerkin finite element algorithm. The flow is expected at downstream of expansion section is gradually/rapidly varied flow and generating vortices. This approach allows to explore very complex hydrodynamics phenomena like computation and analysis of unsteady/transient free–surface flows present hydraulic structures. The different Reynolds number (Re) values are used for computing of streamline flow pattern, velocity and pressure values. It is observed that, with increasing Reynolds number, the wake on the downstream of expansion of channel increase. The stability and capability of the numerical algorithm can be ascertained from the predicted results that the proposed model can be used efficiently for computing through solving of complex geometrical hydrodynamic problems.

Key words: Finite Element Method, Transient Flow, Newtonian Fluid, Streamline, Vortices and Two–Dimensional Flows.

1. INTRODUCTION

With the development and improvement in power of computer technology, computer simulation, using sophisticated numerical models, has become very popular. For the simulation of unsteady flows, many two–and three-dimensional hydrodynamic numerical models, both commercial software as well as in house computer programs, has been developed. The algorithms are suitable for predicting and ascertaining unsteady flows. In the vicinity of hydraulic structures of contraction and expansion channels, the determination of flow conditions is very important. The simulation of three-dimensional unsteady flow is cumbersome, difficult and time consuming than the computation of one-dimensional and two-dimensional flows, which are less complicated (Fennama and Chaundhry, 1990).

Explicit as well as implicit approach within the frame work of finite difference method had been employed by various researchers to compute unsteady two–dimensional flow and flood wave propagation. As explicit methods needs very small time–steps, so the convergence of numerical algorithms is very slow, however, implicit methods do not require small time steps. As two–dimensional model provide more pragmatic description of the flow comparatively one-dimensional models (Tou, 1991). It is quite common that after a mobility of the flood wave over a long one-dimensional reach of the natural watercourse, it arrives finally a low–lying plain. Thus, the two-dimensional nature, in the horizontal sense, of the flow has been taken into account. In the recent literature, numerous numerical models dealing with two-dimensional modelling of flood wave propagation was considered (Bradford & Sanders, 2002 and Jha *et al.*, 2000).

For regular hydrodynamic shape of problems, the finite difference method (FDM) is straight forward and involves less computation overhead. However, due to some limitations such as, inflexible to deal with complex boundaries and boundary conditions, also accurate (Hircsh, 1990). Whereas, finite element method (FEM) discretize a complex computational field relatively more flexible to deal complex boundary conditions (Qureshi and Baloch, 2013).

The flow behavior depends on various parameters, such as viscous and inertial forces. The dominating one is the non–dimensional Reynolds Number (Re). When inertial force i.e., Re values are very small, the flow become steady. Whereas, highly viscous flows i.e., very large values of Re, the flow become instable and unsteady, whereas, the vortices shedding are witnessed. As inertial force (Re) increases, the vortices become asymmetric and weaken in monotony, and the influences of turbulence manifest more visible (Johnson, *et al.* 1994).

A two-dimensional flow past square obstacle for Newtonian fluid was simulated by (Baloch and Qureshi, 2006). At different Reynolds number (Re) values, the velocity field and pressure had been calculated adopting Navier–Stokes equation on Taylor-Galerkin algorithm.

[++]Corresponding author: A. Baloch, email: ahsanullah.baloch@isra.edu.pk
*Department of Petroleum Engineering, University of Stavanger, Norway.
**Department of Computer Science, ISRA University, Hyderabad, Pakistan

In their study, they have noted that in the downstream of square obstacle enhancement of wake depending on inertia, i.e., increase in Reynolds number.

The numerical model, semi-implicit Taylor–Galerkin/ Pressure-Correction algorithm originally developed by Webster and Townsend 1987, and Baloch 1994, subsequently applied for predicting sediment concentration in alluvial channels by (Qureshi, *et al.* 2014) suggested that the scheme is the robust. For unsteady flow, hence, the algorithm has been utilized (Mahessar, 2015).

2. GOVERNING SYSTEM OF EQUATIONS

Consider the two-dimensional time-dependent unsteady flow of incompressible Newtonian fluid, the governing system of equations under velocity–pressure primitive variable formulation comprises the equation of continuity and Navier–Stokes equation omitting body forces, which are described as under:

Continuity Equation

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \qquad (1)$$

Momentum Equation

$$\frac{\partial u}{\partial t}=\nu\left(\frac{\partial^2 u}{\partial \boldsymbol{x}^2}+\frac{\partial^2 u}{\partial y^2}\right)-u\frac{\partial u}{\partial x}-v\frac{\partial u}{\partial y}-\frac{1}{}\frac{\partial p}{\partial} \qquad (2)$$

$$\frac{\partial v}{\partial t}=\nu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}\right)-u\frac{\partial v}{\partial x}-v\frac{\partial v}{\partial y}-\frac{1}{}\frac{\partial p}{\partial} \qquad (3)$$

Where u (x, t) is local component of velocity field in longitudinal axial (x) direction, v(y, t) is local velocity component in transversal vertical (y) direction. Whilst, isotropic pressure (per unit density) is denoted by *p*(x, t), and t is time. Whilst, material parameters are represented by ρ and μ are density and dynamic viscosity of the fluid respectively. While, ν is a kinematic viscosity, a ratio of dynamic viscosity and density of the fluid (μ/ρ).

For better determination, above equations (1–3), are usually cast in a non–dimensional form. The dimensionless unknowns (x*, t*, u* and p*) are selected via an appropriate scales, such as: $x = L_c\, x^*$, $t=\frac{L_c}{U_c}t^*$, $\boldsymbol{u}=U_c\boldsymbol{u}^*$ and $p=\rho U_c^2 p^*$. Whereas, L_c is the characteristic length (taken as size of the cross–section of channel) and U_c is the characteristic velocity (chosen as inlet velocity). Inserting non–dimensional values and for brevity and simplicity, omitting asterisks. The above system (1–3) can be presented in compact vectorial form:

$$\nabla.\mathbf{u}=0 \qquad (4)$$

$$\frac{\partial \mathbf{u}}{\partial t}=\frac{1}{Re}\nabla^2\mathbf{u}-(\mathbf{u}.\nabla)\mathbf{u}-\nabla\mathbf{p} \qquad (5)$$

Where additionally $Re=\frac{\rho UL}{\nu}$ is non–dimensional Reynolds number.

3. DEVELOPMENT OF NUMERICAL MODEL

A finite element algorithm is utilized to develop a two–dimensional numerical model. The method adopt two–step predictor corrector technique to enhance stability and achieve second order accuracy. The semi-implicit scheme has been suggested by (Qureshi, 2014) in multi steps. This model have four fractional solution steps of $U^{n+\frac{1}{2}}$, U*, P^{n+1} and U^{n+1} employing given values of U^n, P^n, $U^{n+\frac{1}{2}}$, U* and p^{n+1}. Fully–discrete system of above governing equations (4 and 5) has been realised in weak form in following steps.

Step 1: At half time step (n + $\frac{1}{2}$) level, Calculate velocity field U through following equation:

$$\left(\frac{2M}{\Delta t}+\frac{S_u}{2R_e}\right)\left(U_j^{n+\frac{1}{2}}-U_j^{n}\right)=\left[\frac{-S_u}{R_e}U_j-N(U)U_j+PL_k+b.t\right]^n \qquad (6)$$

Step 2: Exercise the above information, compute U* and applying the following formulation:

$$\left(\frac{M}{\Delta t}+\frac{S_u}{2R_e}\right)\left(U_j^{*}-U_j^{n}\right)=\left[\frac{-S_u}{R_e}U_j+LP_k+b.t\right]^n-N(U)U_j^{n+\frac{1}{2}} \qquad (7)$$

Step 3: For full time (n + 1) step, estimate pressure differential 'P^{n+1}' from Poisson equation using non–divergence–free velocity U* and pressure 'P^n'values:

$$K\left(P_k^{n+1}-P_k^{n}\right)=\frac{-2}{\Delta t}LU_j^{*} \qquad (8)$$

Step 4: At full time (n + 1) level, compute the second order accurate U utilising non–solenoidal velocity U* and pressure differential P^{n+1} values.

$$\frac{2M}{\Delta t}\left(U_j^{n+1}-U^{*}\right)=L^T\left(P_k^{n+1}-P_k^{n}\right) \qquad (9)$$

Where, mass matrix M, diffusion matrix S_u, non–linear convection matrix N(u), pressure differential stiffness matrix K and pressure matrix L, L^T are matrices. Whereas, b. t. denotes boundary terms; for details reader is referred to (Mahesar, 2015).

4. PROBLEM SPECIFICATION

An asymmetrical contraction and expansion channel is considered as a computational domain as shown in **(Fig.1).** The region of the interest is 100 meters long

and 30 meters wide channel. The length of upstream of portion is 30 meters and downstream portion is 60 meters long. Whilst, the contraction portion is 10 meters long. Total number of finite elements are 9360, whereas, vertex nodes are 4863 and total degree–of–freedoms are 43033 (**Fig. 2).**

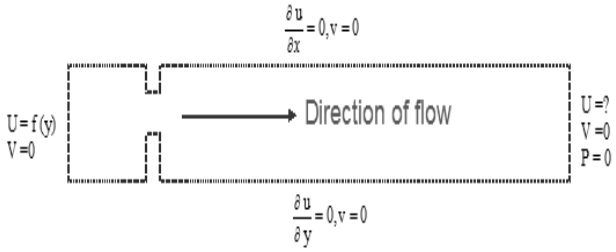


Fig. 1: Schematic diagram of a two–dimensional contraction/expansion channel

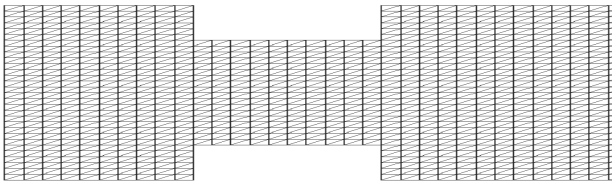


Fig. 2: Finite Element mesh

5. RESULTS AND DISCUSSIONS

5.1 Streamline Contours and Vortex Shedding

The semi–implicit scheme and time step of 0.01 has been used for Re values. The stream function has been computed based on the Galerkin formulation by iterative method, so that the program has been used to calculate the streamlines from the values of velocity gradients (u and v) along x– and y–axis. The streamline and velocity values have been predicted and the streamline contours and velocity vectors have been developed using Tec–plot software.

The stream functions have been computed using the bunch of predicted velocity values at different time units for Reynolds number value of 2.5, 5, 7.5 and 10 **(see Fig. 3–7)**. During the process of vortex shedding, relative positions of vortex alter. The upper vortex in the wake starts growing in side at say t = 50 units and at t = 70 units it breaks away; while, the lower vortex is diffused. In the above Fig. 3-7, fluid is entering from left up–stream plane and contracting in the small channel and subsequently expanding in the expansion plane and show transient development of asymmetric vortices with increasing Reynolds number to reach at steady–state solutions.

The generation of expansion–vortices and flow separations are displayed inside the downstream expansion section. An incoming flow from the exit corner of the contraction region, separate in the expansion region and reattach to the downstream wall, creating an asymmetric complex recirculation region on both nearby to side walls. At Re = 2.5, twin asymmetric vortices develop on both sides of channel with single vortex centres up to non–dimensional time units of fifty (50).

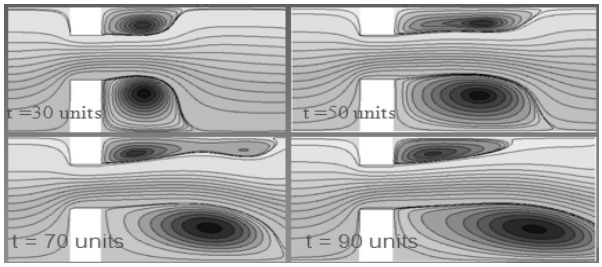


Fig. 3: Streamline pattern (Transient Development for Re=2.5)

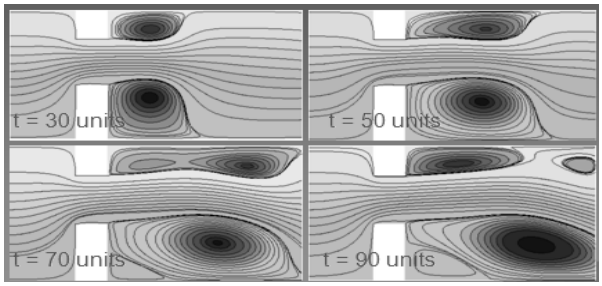


Fig. 4: Streamline pattern (Transient Development for Re=5)

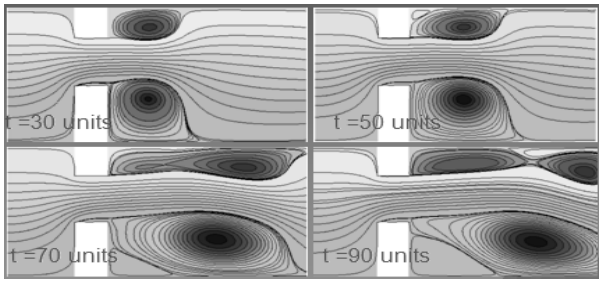


Fig. 5: Streamline projection Transient Development of Flow structure at Re = 7.5.

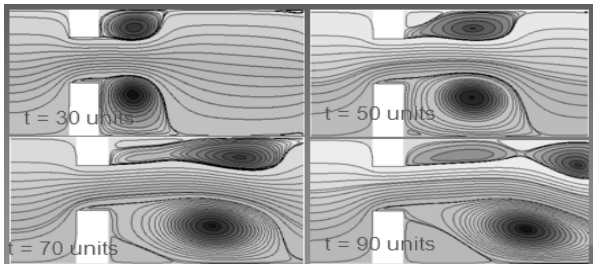


Fig. 6: Streamline pattern: Transient Development at Re=10.

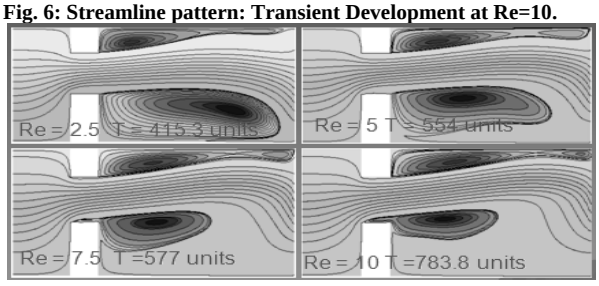


Fig. 7: Steady–State Streamline Projection at different Re values.

As time–marching process proceeds further, at time unit equal to seventy (70), an embryo vortex centre appear in the vicinity of upper wall of channel. With increasing Reynolds number (>=5), this embryo vortex centre strengthen further, span out and occupy larger area of downstream and also an intense vortex enhancement has been observed. At Re values around 5, a more complex nature of streamline pattern was anticipated with the formation of two salient corner vortices, followed by an intense enhancement of twin vortex centres, which engulfs the salient corner vortex at around Re = 7.5, forming a long vortex that continues to grow with increasing of the Re values. Whereas, at Re = 7.5, a development of trailing edge vortex is also observed in the small contraction plane.

The resulting steady–state flow patterns in the expansion region have been classified with increasing Reynolds number, as asymmetric salient corner vortices and trailing edge vortex, or as a combination of both trailing edge vortex and asymmetric salient corner vortex structures.

With respect to the accuracy and stability solution process is monitored through computing minimum and maximum increment error norms, i.e., least square L_2 and L_∞. For time–marching algorithm, convergence of steady-state solutions achieved at the tolerance of 10^{-6}.

6. CONCLUISION

The water surface flow behaviour predicted by the proposed model simulations describes that the generation of vortices at downstream of contraction section as well as along the channel. The capability of Navier-Stokes model has to correctly simulate transient free surface flows. The simulated results of the proposed model encourages that the model can be utilized for computing flow behaviour.

For physical quantities, 2-D model has provided appropriate evidence in both space and time, for flow structure and velocity profile, free surface profile, and wave front dynamics. Therefore, proposed model, in the vicinity of field region and furthermore, can be efficiently and effectively applied.

AKNOWLEGEMENTS

The authors are thankful to Mehran University of Engineering and Technology, Jamshoro, Pakistan for providing the facilities to perform the research work.

REFERENCES:

Baloch, A., (1994). Numerical Simulation of Complex Flows of Now-Newtonian Fluids, PhD Thesis, University of Wales, Swansea, UK.

Baloch, A., and A. L. Qureshi, (2006). “Parallel Simulation of Fluid Flow past a Square Obstacle”, Mehran University Research Journal of Engineering and Technology, Jamshoro, Pakistan, 25 (1): 1-8.

Bosch, M and G. Rodi, (1996). “Simulation of Vortex Shedding past a Square Cylinder near Wall”, Intl. J. Heat and Fluid Flow, 17: 267–275, Elsevier Science Inc.

Bradford, S. F., and B. F. Sanders, (2002). “Finite-volume model for shallow water flooding of arbitrary topography”, ASCE J. Hydr. Eng., 128: 289-298.

Fennema J, and M. H. Chaudhry, (1990). “Explicit Methods for 2-D Transient Free-Surface Flows”, Journal of Hydraulic Engineering, ASCE, 116(8): 1013-1034.

Hircsh, C., (1990). “Numerical Computation of internal and external flows, velocity; computational Methods for in viscid and viscous flows”, John Wiley and sons, Chichester-Singh.

Jha, A. K.., J. Akiyama and M. Ura, (2002). “Flux-difference splitting schemes for 2D flood flows”, *J. Hydraul. Eng.,* 126(1): 33–42.

Johnson, C., (1994), “A New Paradigm for Adaptive Finite Element Methods, In the Mathematics of Finite Elements and Application”, (Whiteman, J. R., Ed.) Johan Wiley and Sons, Inc., New York.

Mahessar, A. A., (2015). “Numerical Simulation of Unsteady Free Surface Flow”, PhD Thesis, Mehran University of Engineering and Technology, Jamshoro.

Qureshi, A. L., and A. Baloch, (2013). “Finite Element Simulation of Sediment Transport: Development, Validation and Application of Model to Photo minor of Jamrao West Branch, Sindh, Pakistan”, *In Advances in Sediment Transport Research*, edited by Fukuoka et al. CRC Press book, Taylor and Francis Group, London, UK.

Qureshi, A. L., A. A. Mahessar, and A. Baloch, (2014). “Finite element model for unsteady open channel flows with sudden contraction/expansion”, *IOSR Journal of Mechanical and Civil Engineering,* 11(5): 10-17.

Tou, S. K. W., (1991) “A 2-D Finite Element Model for Unsteady Free-Surface Flow and its Experimental Verification”, Computer-Aided Civil & Infrastruc. Engg., 6(4): 335–342.

Townsend, P., and M. F. Webster, (1987). “An algorithm for three-dimensional transient simulation of non-Newtonian fluid flows”, *In: Theory and Applications, Proc. of Numeta Conf., Num. Meth.,* NUMETA, 02, T12, 1-11, Nijhoff.