Sindh Univ. Res. Jour. (Sci. Ser.) Vol. 48 (1) 109-112 (2016)

SINDH UNIVERSITY RESEARCH JOURNAL (SCIENCE SERIES)

Time–Dependent Flow Through Asymmetric Contraction and Expansion Channel

A. A. MAHESSAR, Z. A. K. BALOCH*, A. L. QURESHI, M. A. SOLANGI**, A. BALOCH++***

US Pakistan Center for Advanced Studies in Water (USPCAS-W), IWREM, M.U.E.T, Jamshoro, Pakistan.

Received 9th June 2015 and Revised 22rd December 2015

Abstract: A computation of flow through asymmetric contraction and expansion channel is presented in the paper. A two–dimensional model is reported for computing incompressible flow of Newtonian fluid adopting Navier–Stokes equations using semi–implicit Taylor–Galerkin finite element algorithm. The flow is expected at downstream of asymmetric expansion section is gradually/rapidly varied flow and generating vortices on both sides of channel wall. This approach allows to explore very complex hydrodynamics phenomena like computation and analysis of unsteady/transient free–surface flows present hydraulic structures. The different inertial (Reynolds number) values are used for computing of flow structure by streamline contours, velocity and pressure values. It is observed that, by enhancing inertial values, the wake on 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 solving complex geometrical hydrodynamic problems.

Keywords: 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 simulation of unsteady flows, many two– and three–dimensional hydrodynamic numerical models, both commercial software as well as in house computer programs, have been developed. Recent advancement in numerical method, in the field of hydraulic research, prediction of the solutions are one of the most exciting and dynamic topics (Jia, 2001-2). 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 3-D unsteady flow is cumbersome, difficult and time consuming than the computation of one- and two-dimentional flows, which are less complicated (Fennema, 1990).

Explicit as well as implicit approach within the framework of finite difference method had been employed by various researchers to compute unsteady two-dimentional flow and flood wave propagation. As explicit methods need very small time–steps, so the convergence of numerical algorithms are very slow; however, implicit methods do not require small time steps. As 2-D model provide more pragmatic description of the flow comparatively 1-D 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 the two dimensional modelling of the flood wave propagation has been considered (Garcia-Navarro, e*t al.,* 2000, Bradford, 2002).

For regular hydrodynamic shape of problems, the finite difference method (FDM) is straight forward and involves less computation overhead. There are some limitations that is inflexible to deal with complex boundaries. Whereas, finite element method (FEM) discretise a complex computational field into plain tessellations through structural and/or unstructured formation and relatively more flexible to deal with complex boundary conditions (Qureshi, 2013).

The flow behaviour depends on various parameters, such as viscous and inertial forces. The dominating one is the non–dimensional Reynolds Number (Re). When 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. Vortex shedding is an unsteady flow phenomenon that occurs frequently behind obstacles of different shapes (Bosch, 1996). A two–dimensional flow past square obstacle for Newtonian fluid was simulated by Baloch and Qureshi (2006) for different Re values calculating velocity field and pressure values using Taylor–Galerkin algorithm.

The numerical model, semi-implicit Taylor-Galerkin/ Pressure-Correction algorithm originally developed by (Townsend, 1987; Baloch, 1994),

++Corresponding author: A. Baloch, email: ahsanullah.baloch@isra.edu.pk
*Department of Petroleum Engineering, University of Stavanger, Norway.
**Department of Basic Sciences and Related Studies, Mehran University of Engineering and Technology, Jamshoro, Pakistan.
***Department of Computer Science, ISRA University, Hyderabad, Pakistan

subsequently applied for predicting sediment concentration in alluvial channels by Qureshi and Baloch (2013) suggested that the scheme is robust. For unsteady flow, 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

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

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

Where u (x, t) is local component of velocity field in longitudinal axial (x) direction, v (y, t) islocal 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 a kinematic viscosity of the fluid 'ν' (= μ/ρ): a ratio of dynamic viscosity of the fluid 'μ' and fluid density '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: $x=L_c x^*$, $t=\frac{L_c}{U_c}t^*$, $\mathbf{u}=U_c\mathbf{u}^*$ and $p=\rho U_c^2 p^*$. Where as, 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:

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

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

3. NUMERICAL ALGORITHM

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 (Mahessar, et *al.,* 2015). 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 equations (4 and 5) has been realized in weak form:

Step 1:

At half time step (n + $\frac{1}{2}$) level, calculate velocity field $U^{n+\frac{1}{2}}$ 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 second order accurate U utilizing 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, M is mass matrix. For details matrices, reader is referred to (Mahessar, e*t al.*, 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. The total number of finite elements are 9360, whereas, vertex nodes are 4863 and total degree–of–freedoms are 43033. The finite element utilized in the simulation is illustrated below in **(Fig.2)**.

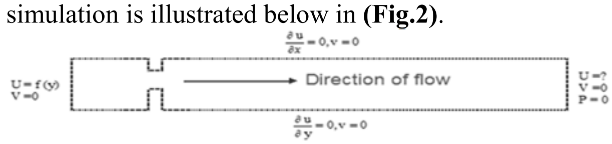

Fig.1: Schematic diagram of a two–dimensional contraction/expansion channel

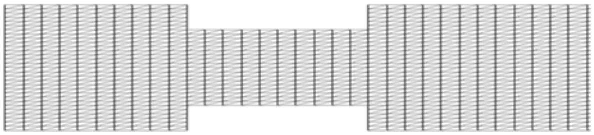

Fig.2: Finite Element Mesh of computational domain

5. NUMERICAL RESULTS AND DISCUSSIONS

5.1 Contours and Vortex Shedding

A semi–implicit finite–element scheme is adopted as discussed above. Algorithm is based on Taylor–Galerkin with combination of second order pressure–correction method. For primitive variable solution process, time step is fixed ($\Delta t = 0.01$). At the end of solution process, from the values of velocities, the stream function has been computed as a postprocessor based on the Galerkin formulation using iterative method. The flow structure is presented through streamlines. The predicted numerical results are displayed through streamline patterns showing core flow structure and recirculation/vortices in the downstream of domain to demonstrate the influence of fluid inertia. To illustrate the time–depend effects, at different non–dimensional time units, flow structure is presented in the following figures. In core flow structure and in both upper and lower recirculating region, streamlines are drawn at equal spaced, this is due to large variation in eddies; where, it is hard to choose comparative space in streamlines. Therefore, in recirculating region, streamlines are selected separately for each Reynolds number.

5.2 Mesh Convergance

For mesh convergence, two structural meshes are employed, first one is coarse mesh, while, other mesh has been developed from first mesh. In all simulation cases second refined mesh has been used. Time–dependent solution process is monitored through computing minimum and maximum increment error norms, i.e., least square L_2 and L_∞. The stream functions have been computed using the predicted velocity values at different dimensionless time units for low Re value of 10, 15, 25 and 50. The streamline patterns were drawn **(Figs 3 to 6)**. The 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 up to small 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.

Initially, at low value of inertia, an embryo vortex centre appear in the vicinity of upper wall of channel. As time–marching process proceeds further, this embryo vortex centre strengthen and form asymmetric twin vortices develop on both sides of channel with single vortex centres up to non–dimensional time units of fifty (50). Increasing inertia, a more complex nature of streamline pattern has been anticipated with the formation of two salient corner vortices, followed by an intense enhancement of twin vortex centres, which engulfs the salient corner vortex forming a long vortex that continues to grow span out with increasing inertial valuesand occupy larger area of downstream and also an intense vortex enhancement has been observed.

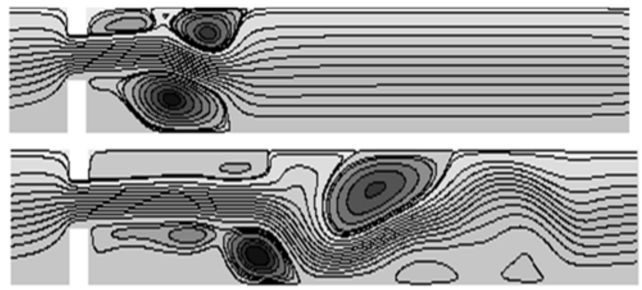

Fig.3: Streamline pattern, at Re=10, for non–dimensional time units t=100 and 300.

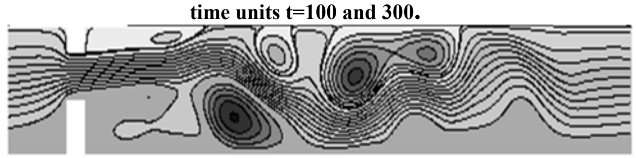

Fig.4: Streamline pattern, at Re=15, for non–dimensional time unit 300.

While, at Re = 10, up to non–dimensional time units t=50, flow structure display the vortex shading anunequal single salient corner vortices developed on both sides. In Fig–3 it is demonstrated that at time units t = 100, vortices breaks away in several vortex centres and this wake move further downstream and grow stronger. Whereas at t = 300, lower vortex is diffused and during the process of vortex shedding, the relative positions of vortex alter and symmetry of vortex shading breakups.

This phenomena of asymmetry vortex shading augmented further with increasing Re value from 15 to 25. If the computations are continued for Reynolds number, the recirculating region propagate and develop further at some significant region of the expansion plane.

Fig.5: Streamline pattern, at Re=25, for non–dimensional time unit 300.

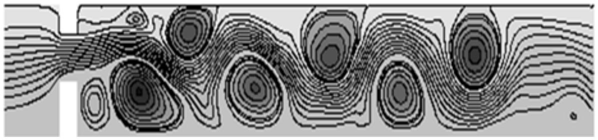

Fig.6: Streamline pattern, at Re=50, for non–dimensional time unit 300.

At relatively higher Reynolds number of fifty (=50) at non–dimensional time units of t = 300, flow becomes prominently unsteady and one can see meandering effects in downstream of the flow geometry. Unfortunately, this would require a large downstream domain and continue increase in the number of nodes in x-direction to ensure that outflow boundary far away and maintain sufficiently far downstream of long eddy.

6. CONCLUSIONS

The water surface flow behaviour predicted by the proposed model simulations describes that the generation of vortices at downstream of expansion region as well as along the channel. The capability of the model has to correctly simulate time-dependent free surface flows. The simulated results encourages that the model can be utilized for computing flow behaviour in field conditions.

For physical quantities, two-dimensional model has provided appropriate evidence in both domains, i.e., space and time, such as flow structure, free surface profile, and wave front dynamics. Therefore, proposed model, in the vicinity of field region and further more, can efficiently and effectively be applied. For mesh convergence, it is observed that in the downstream expansion plane the flow structure is comparatively plane; whilst, very small change in the values of excess pressure drop, recirculating flow rate and vortex reattachment length.

AKNOWLEGEMENT

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

REFERENCES:

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

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., R. (1996), "Simulation of Vortex Shedding past a Square Cylinder near Wall", Intl. J. Heat and Fluid Flow, 17: 267-275, Elsevier Science Inc.

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

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

Garcia-Navarro, P., P. Brufau, E. M. V-azquez-Cendon, (2000) "Zero mass errorusing unsteady wetting-drying conditions in shallow flows over dry irregular to pography"', Int. J. Num. Meth. Fluids, 45: 1047-1082.

Jia, Y., S. Y. Wang, (2001-2), "CCHE2D Verification and Validation Tests Documentation", Technical Report No. NCCHE-TR.

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

Tou, S. K. W., (1991), "A 2-D Finite Element Model for Unsteady Free-Surface Flow and its Experimental

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

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

Mahessar, A.A., Z.A.K. Baloch, A. L. Qureshi, and A. Baloch, (2015), "Transient Analysis of Two–Dimensional Free Surface Flow through Hydraulic Structure", Sindh Uni. Res. J. (Sci. Series), 47(4): 713-716. http://sujo.usindh.edu.pk/index.php/SURJ/article/view/1713/1552

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

Townsend, P., M. F. Webster, (1987), "An algorithm for three-dimens. transient simulation of non-Newtonian fluid flows", Theory and Appl., Proc. of Numeta Conf., Num. Meth., NUMETA, 2(T12) 1-11, Nijhoff.

Verification, Computer-Aided Civil and Infrastructure Engineering, 6(4): 335–342.