



On the Unsteady Diverging Flow through Open Channel

A. A. MAHESSAR*, Z. A.K. BALOCH**, A. L. QURESHI, S. M. KORI, A. BALOCH^{††***}

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

Received 9th June 2015 and Revised 22nd March 2016

Abstract: A computational FE model was constituted for investigating of unsteady and non-uniform flows in open channels. Unsteady flow in open channels is complex by variation in hydraulic properties and complex geometry of channels and also the flood plains. The computation of such flow phenomena in open channels through numerical model is due to present sophisticated numerical solvers and enhanced modern computers. In this paper, unsteady flow conditions in sudden expansion channel has been investigated and computed using Navier Stokes techniques and semi-implicit Taylor Galerkin in finite element scheme. It has been observed that velocity profile in sudden expansion channel is increased with increase of Reynolds number, hence, higher in front and lower enlarged side. The efficiency and accuracy can be ascertained from simulated results of the presented model which represents; the presented model can be applied for addressing irregular geometry problems and computing open channel flow.

Keywords: 2-D FEM, Velocity Profile, Sudden Expansion Channel,

1 **INTRODUCTION**

The numerical analysis of an unsteady flow in open channels is interesting computation field. Hence, the flow problems have attracted to investigators for computing the flow conditions in open channels. The contraction and expansion channel flows display the ideal class flow which includes stream and re-circulating flows for validating the accuracy of two-dimensional and three dimensional algorithms (Baloch, 1996).

The sufficient computer power availability, together with modern tools for computation of Fluid Dynamic (CFD) analysis has opened the vista to computation of flow problems on increasing complexity. Hence practical application of free surface flows simulation indicates most fascinating problem. Inherent complexities in the calculation of unsteady flows and non-regularity of computational boundaries of domains present challenges to researchers (Larese, 2007).

Complexities of floodplains and arbitrarily geometrical structures within floodplains, the numerical algorithm for simulation of two-dimensional free surface flow is able to handle water bodies of arbitrary shapes and using flexible ranges in spatial and temporal scales to ascertain exact requirements at different times and locations (Zhang, *et al.*, 2004).

The two-dimensional algorithm gives real description of flow condition than one-dimensional approach and simple computation than three-dimensional approach. The finite elements (FE)

technique is more accurate than the method of characteristics while problems are two-dimensional or three-dimensional nature (Tou, 1991).

The finite difference (FD) technique has been used in past almost all of numerical models, it has observed that FDM requires using small time and space steps, which leads to decrease its efficiency. However, FE technique is more efficient, accurate and is flexible in handling irregular geometry shape, and boundary conditions. It is very important and vigorous to solve complex CFD problems, a lots of efforts are required for the computation of complex flow field problems (Mahessar, *et al.*, 2016 and 2015).

The complexity of flow behaviour inside complex geometries has fascinated both experimental and theoretical studies. Dimensionless governing equations depends on only one dimensionless parameter, i.e., the Reynolds Number (Re) represents fluid inertia or material parameters. The flow may reach steady-state, when the value of Re is low. Contrary, flow becomes unsteady when Re is higher and complexity enhance, and may develop shedding. As changing material parameters of value of inertia (Re), enhancing, eddies weaken in shape regularity. Influence of turbulence become more noticeable (Johnson, *et al.* 1994).

Sudden enlargement flows become the focus of many experimental studies that have been demonstrated in the 2-D case, size of re-circulating flow region increases linearly with increasing Reynolds number (Durst, *et al.* 1974).

^{††}Corresponding author: A. Baloch, email: ahsanullah.baloch@isra.edu.pk

* Sindh Barrages Improvement Project, Irrigation Department, Government of Sindh, Hyderabad. amahessar@yahoo.com

**Department of Petroleum Engineering, University of Stavanger, Norway,

***Department of Computer Science, Isra University, Hala Nakah, Hyderabad, Sindh.

The velocity and pressure values were computed employing the Taylor-Galerkin algorithm with different Reynolds number (Re) values. It was observed that the wake is increasing with increasing of Reynolds number on the downstream side of square obstacle (Baloch and Qureshi, 2006).

Two dimensional finite element model was constructed in order to compute Newtonian fluid flow employing Navier–Stokes equations using Taylor–Galerkin algorithm. This approach allows for computing hydrodynamic phenomena and exploring unsteady and non-uniform flow patterns in open channels with expansion and contraction (Mahessar, 2015).

Numerical techniques are used not only for development of the interface, but also for exploring of the pressure and velocity characterizing the flow itself. The employment of finite element standard Galerkin scheme requires for diverse discrete finite element spaces for pressure and velocities (Brezzi, et al. 1991) in order to ensure stability, many different stabilized methods have been proposed to allow to use the same spaces for pressure and velocities. To explore algorithmic aspects of this technique when applied to the equations of Navier–Stokes another goal of the present work.

2. TWO-DIMENSIONAL MODEL AND NUMERICAL SCHEME

The hydrodynamic equations have been applied for the construction of a two-dimensional model. Let Ω is two-dimensional domain of \mathfrak{R}^2 and Γ is its boundary with spatial x and temporal t coordinate. The flow of a Newtonian is governed by the non-dimensional

continuity and Navier–Stokes formulations in two-dimensions varied unsteady flow can be derived along equations have been described as under:

$$\text{Continuity Equation: } \nabla \cdot \mathbf{u} = 0 \quad (1)$$

$$\text{Momentum Equation: } \frac{\partial \mathbf{u}}{\partial t} = \frac{1}{\text{Re}} \nabla^2 \mathbf{u} - \mathbf{u} \cdot \nabla \mathbf{u} - \nabla p \quad (2)$$

Where, \mathbf{u} is the velocity vector, p is pressure, t is time, μ is dynamic fluid viscosity, ρ is fluid density, ∇ is gradient operator and ∇^2 is Laplacian operator, where additionally $\text{Re} = \frac{\rho U_c L_c}{\mu}$ is Reynolds number, U_c and

L_c are the characteristic velocity (inlet velocity) and length (size of the cross-section of channel), respectively. For initial conditions are specified, the value of \mathbf{u} at the initial time: $\mathbf{u}(\mathbf{x}, 0) = 0$. The boundary conditions are taken essential condition, fully developed axial velocity profile is imposed at inlet with maximum velocity $U_{\max} = 0.9$ and vanishing cross flow, traction free at outlet, and along the remaining boundaries vanishing tangential tractions and normal velocity components are imposed. The no-slip boundary condition is imposed at the contraction section. The inclusion upstream and/or downstream by the specification of p at a single point often as $p = 0$. For above governing system of equations, fully-discrete system in the weak formulations implementing the numerical scheme over following fractional steps are written, computation of $\mathbf{u}^{1+1/2}$, \mathbf{u}^* , \mathbf{u}^{n+1} , p^{n+1} , using given \mathbf{u}^* and p^* values, as:

Step-1: Predict the Velocity U at half time step $(n+1/2)$ level using following equation:

$$\left(\frac{2M}{\Delta t} + \frac{S_u}{2R_e} \right) \left(U_j^{n+1/2} - U_j^n \right) = \left[\frac{-S_u}{R_e} U_j - N(U) U_j + PL_k + \mathbf{b} \cdot \mathbf{t} \right]^n \quad (3a)$$

Step-2: Exercising the above information, compute U at full time $(n+1)$ level applying the following formulation:

$$\left(\frac{M}{\Delta t} + \frac{S_u}{2R_e} \right) \left(U_j^* - U_j^n \right) = \left[\frac{-S_u}{R_e} U_j + LP_k + \mathbf{b} \cdot \mathbf{t} \right]^n - N(U) U_j^{n+1/2} \quad (3b)$$

Step-3: For full time $(n+1)$ step, estimate pressure difference ‘ P ’ from Poisson equation using non-divergence free velocity U^* and pressure P^n value:

$$\mathbf{K} \left(P_k^{n+1} - P_k^n \right) = \frac{-2}{\Delta t} \mathbf{L} U_j^* \quad (3c)$$

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

$$\frac{2M}{\Delta t} \left(U_j^{n+1} - U^* \right) = \mathbf{L}^T \left(P_k^{n+1} - P_k^n \right) \quad (3d)$$

Where M is mass matrix, S is Momentum Diffusion matrix, $N(U)$ is convection matrix, b.t. is *Boundary terms*, K is *pressure Stiffness matrix*, P is *Pressure Gradient matrix and superscript T* represents transpose of matrix. Analysis of solution process with respect to its accuracy and stability is monitored by calculating increment error norms L_2 and L_∞ tolerating to order of 10^{-6} for time marching steady-state convergence.

4. PROBLEM SPECIFICATION:

The computational domain contains 12m-long and 0.25m to 0.5m wide sudden expansion channel. A non-symmetrical 2-D channel where in incompressible fluid across the flow direction. The flow is expected to vary rapidly at downstream along the sudden expansion channel. The schematic diagram is shown in (Fig.1). A finite element mesh denoted by having 8380elements, 17331 total nodes, vortex node 4471 and 56464 DOF (Degrees of Freedom).display in (Fig.2)

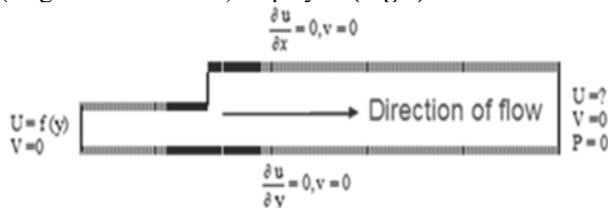


Fig.1: Schematic diagram of a sudden expansion 2-D channel

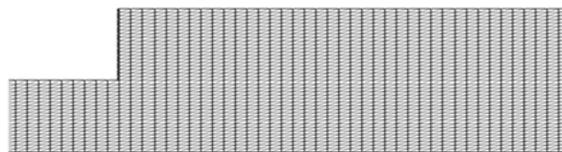


Fig. 2: Finite element mesh of computational domain

5. NUMERICAL RESULTS AND DISCUSSIONS

The Figure 3demonstrates that velocity vectors in the sudden enlargement channels. The velocity in the sudden expansion channel at contraction portion as well as in straight direction is higher in the enlargement portion. The velocity vector profile shows that velocity flow from contraction portion to sudden enlargement fluxes higher in straight direction and varies with expansion channel from inlet to out boundary of channels.

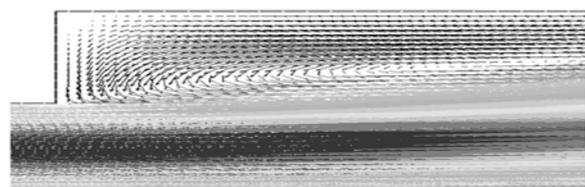


Fig.3: Vector velocity profilein suddenexpansion two-dimensional channel

The legends are shown along with figures 4 to 6 which represent the velocity profile along the sudden enlargement channel. The maximum velocity is 0.9 m/s, the velocity is varying from 0 to 0.9 m/s. Hence, the legends of figures display flow conditions within the channel.

The velocity profile is computed with different Reynolds Number (Re); all computation has been carried out within same time unit 12.The velocity profile in sudden enlargement channel has been computed using Re 500 as shown in Figure 4. It is observed that the velocity at contraction portion is much higher as comparison the sudden enlargement channel. The velocity behaviour in the expansion channel reveals that at inlet section of enlargement channel is higher which is decreasing gradually at outlet boundary of channel but at corner of inlet section of enlargement is very low as corner outlet boundary of channel.

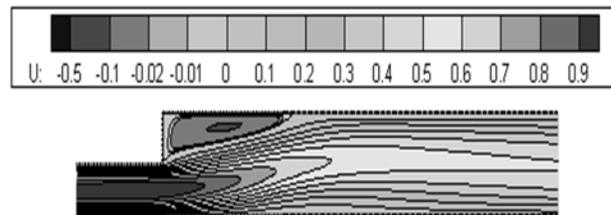


Fig. 4: Contours of axial velocity (m/s)in sudden expansion channel (Re=500, t=12)

The velocity profile shown in the following figures5 and 6, which represents that velocity in the sudden enlarged channel is increasing with increasing Re value. In this regard, Re value 700 is used further investigating and it is observed that the velocity shoots from contraction portion into sudden expansion section but initially velocity is shoots in the straight channel direction towards of outlet boundary and gradually lower velocity in the expansion portion of channel. This developed phenomenon in sudden enlargement channel which leads exact images real field problems.

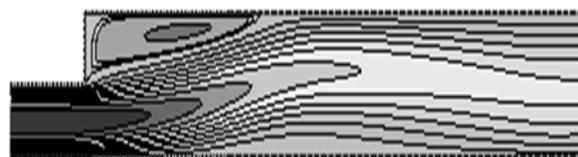


Fig. 5: Contours of axial velocity (m/s) in sudden expansion channel (Re=700, t= 12)

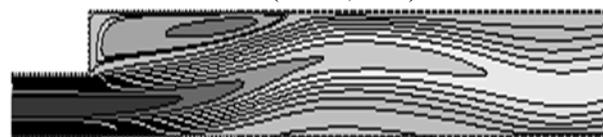


Fig. 6: Contours of axial velocity (m/s) in sudden expansion channel (Re=1000, t= 12)

The figures demonstrate that behaviour of velocity in this channel which represents the real velocity profile in the hydraulic channels. The legends of figures display that velocity is varying from negative value to a positive velocity (i.e. in flow direction) of 0.9 m/s. The figures show that the velocity is higher in the straight direction of channel and low at side of enlargement channel. The negative velocity is observed in the upper enlargement portion up to value of -0.5 m/s. It is also observed that maximum velocity is developed in straight direction which shoots to outlet boundary of channel and low velocity side of enlargement section which cause of silt deposition on the flank of expansion especially at corner where velocity is negligible. It has been observed that due to maximum velocity at straight direction of channel which may cause of scouring the bed of channel and disposition of sediment load at side of enlargement section due to low velocity. The results of this numerical model demonstrate nice, hence it can be applied for solution of river training problems i.e., controlling scouring, bank erosion and approach velocity to hydraulic structure for avoiding any damage and failure of hydraulic structure on rivers/channels.

6. CONCLUSIONS

The computed results of presented numerical model represent the simulation of real field problem solution. The numerical results show that the developed model is more accurate, stable and efficient in computing two-dimensional numerical problems.

Good agreement is obtained between the numerical solutions. Primitive variable finite element algorithm employed in the simulations is based on a time-stepping scheme explained above is observed that scheme is robust in predicting a range of unsteady complex flows.

The results that have been presented in this research work fall into a number of categories. The varying velocity profile in contraction and sudden enlargement section of channel with high and low velocity due to different shape and boundary conditions. This presented model is nice tool for addressing the river training and hydraulic problems in open channels.

7. ACKNOWLEDGMENT

The authors are thankful to Mehran University of Engineering and Technology, Jamshoro for providing the opportunity for conducting this research work. Second author would like to acknowledge the Department of Petroleum Engineering, University of Stavanger, Norway.

REFERENCES:

- Baloch, A. (1994), "Numerical Simulation of Complex Flows of Non-Newtonian Fluids", Ph.D. 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.
- Brezzi, F. and M Fortin, (1991) "Mixed and hybrid finite element methods", Springer Series in Computational Mathematics, Springer Verlag, New York.
- Johnson, C. (1994), "A New Paradigm for Adaptive Finite Element Methods, In the Mathematics of Finite Elements and Application", (Whiteman, J.R., Ed.) John Wiley and Sons, Inc., New York.
- Larese A., (2007), "Validation of the particle finite element method (PFEM) for simulation of free surface flows", Engineering Computations: International Journal for Computer-Aided Engineering and Software, 25(4).
- Mahessar, A. A., Z. A. K. Baloch, A. L. Qureshi, A. Baloch, and M. A. Solangi (2016), "Time Dependent Flow through Asymmetric Contraction and Expansion Channel", Sindh Uni. Res. J. (Sci. Ser.), 48(1): 109-112, <http://sujo.usindh.edu.pk/index.php/SURJ/article/view/2158>.
- 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 Flowthrough Hydraulic Structure", Sindh Uni. Res. J. (Sci. Series), 47(4): 713-716, <http://sujo.usindh.edu.pk/index.php/SURJ/article/view/1713/1552>.
- Tou, S. K. W., (1991), "A 2-D Finite Element Model for Unsteady Free-Surface Flow and its Experimental Verification" Computer-Aided Civil and Infrastructure Engineering, 6(4): 335-342.
- Zhang, X. H., (2004), "An Approach to Inundation Simulation in Large River Basins Using the Triangle Finite Difference Method", Journal of Environmental Informatics 3(1): 51-63.