Simulation of Flow Pattern in Open Channels with Sudden Expansions

[1]J. Mamizadeh and [2]S.A. Ayyoubzadeh
[1]Department of Water Engineering, Ilam University, Iran
[2]Department of Water Structure, Tarbiat Modares University, Iran

Abstract: Three-dimensional flows in sudden expansions of rectangular channels have been investigated. In this research, CFD simulation model (FLUENT6.2) is used. The k-ε turbulence model is used to simulate turbulence. Desired canal with coarse, medium and fine mesh size is selected for simulation. The results in canal with sudden expansion showed that the vortex formed by coarse mesh size and number of iteration more than 400 is asymmetric and for small and medium mesh size is symmetric. In order to analyze experimentally and numerically more closely, the pattern of flow in a flume with sudden expansion is investigated. Results showed that, asymmetrical flow pattern can be viewed with some minor changes such as roughness coefficient in any part of flume.

Keywords: Flow pattern, sudden expansion, symmetric flow

INTRODUCTION

Bifurcation phenomena or circular flow are frequently encountered in fluid mechanics and hydraulics. Some examples include transonic inviscid flows over airfoils and cylinders, incompressible and compressible viscous flows over airfoils and incompressible viscous flows through sudden expansions. The latter is the subject of the present study. A sudden expansion creates a separation of the boundary layer from the wall, which results in significant pressure loss. The magnitude of the pressure loss depends upon the Reynolds number and the geometric expansion ratio. In most cases, it is desired to minimize pressure loss in a piping system, since the pressure loss correlates to lost energy.

Many experimenters have been observed that the flow in symmetric expansions is asymmetric. Some of these situations are documented in Graber (1982, 2006) who gave an explanation of such phenomenon. He provided a predictive method that agreed well with the experimental observations and then extended it theoretically to analyze proposed corrective measures. Asymmetric flow patterns may occur in perfectly symmetric abrupt expansions, in which the main flow deflects and attaches arbitrarily to one wall of the expansion. The adverse effects include lengthening of the distance to full-width flow in expansions, inefficient operation of screens in screen canals and increasing the potential for vortexing in pump intakes. Cherdron *et al.* (1978) studied the detailed description of the velocity characteristics of the asymmetric flows which form in symmetric, two-dimensional, plane, sudden-expansion

geometries by flow visualization and laser-Doppler anemometry. The flow and geometry boundary conditions which give rise to asymmetric flow are indicated and the reason for the phenomenon is shown to lie in disturbances generated at the edge of the expansion and amplified in the shear layers. Zhou (1995), develop a depth average mathematical model and test it in a canal with sudden expansion. Result showed that the flow rate, flow lines and vortex Formed on both sides are asymmetric. Aloui and Souhar (2000), presents an experimental study of recirculating flows downstream of a flat duct sudden expansion. Results of measurements of average and RMS quantities of the pressure P and the axial velocity U exhibit the asymmetry of the flow behind the sudden expansion as in the two-dimensional case. Escudier *et al.* (2002), reported an experimental investigation of turbulent flow through a plane sudden expansion. Not only was the mean flow found to be strongly asymmetric, but integration of the mean axial velocity profiles revealed significant departures from two dimensionality along the centerplane of the expansion duct. The numerical analysis of the turbulent fluid flow through an axi-symmetric sudden expansion passage has been carried out by Roy *et al.* (2010). The recirculation bubble generated due to the sudden expansion of the passage is observed to reduce in size and strength with the increase in the Reynolds number. But the size and strength of the recirculation bubble increases with the increase in the expansion ratio. Theoretical studies have been made by Roul and Dash (2011) to determine the pressure drops caused by abrupt flow area expansion/contraction in small circular pipes for two-phase flow of air and water mixtures at room

Corresponding Author: J. Mamizadeh, [1]Department of Water Engineering, Ilam University, Iran

temperature and near atmospheric pressure. The pressure drop is determined by extrapolating the computed pressure profiles upstream and downstream of the expansion/contraction. Based on the numerical results as well as experimental data, correlations are developed for two-phase flow pressure drops caused by the flow area contraction as well as expansion. The above review research indicates that few studies done on flow pattern in sudden and gradual expansions and therefore evaluated in this study. In this research, flow pattern in sudden expansions studied with laboratory work and CFD simulation model.

MATERIALS AND METHODS

FLUENT is a Computational Fluid Dynamics (CFD) software package to simulate fluid flow problems. It uses the finite-volume method to solve the governing equations for a fluid. It provides the capability to use different physical models such as incompressible or compressible, inviscid or viscous, laminar or turbulent, etc. Geometry and grid generation is done using GAMBIT which is the preprocessor bundled with FLUENT. This software uses a control-volume-based technique to convert a general scalar transport equation to an algebraic equation that can be solved numerically. Pressure-velocity coupling is achieved by using five algorithms: SIMPLE, SIMPLEC, PISO, Coupled and Fractional Step. In the case of incompressible turbulent flow and averaged in time, conservation of mass (1) And momentum conservation (2) Expressed as follows:

$$\frac{\partial \overline{u}_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial \overline{u}_i}{\partial t} + \overline{u}_j \frac{\partial \overline{u}_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_i} + g_i + \frac{\partial}{\partial x_j}(v \frac{\partial \overline{u}_i}{\partial x_j} - \overline{u'_i u'_j}) \tag{2}$$

where, u_i = velocity component in x_i direction, υ = kinematic viscosity, ρ = density, g_i = gravity acceleration component in x_i direction, p = pressure and $-\overline{u'_i u'_j}$ = Reynolds stress. This equation contains four main components of the unknown velocity in three directions and pressure. Also includes six components of unknown Reynolds stress equation. Thus, system of equations is not closed and turbulence model must be used. $k - \varepsilon$ turbulence model is used in this study. In the other part of research, flow pattern in sudden expansions studied with laboratory flume in tarbiat modares university-iran (Mamizadeh, 2009).

RESULTS AND DISCUSSION

Simulated flow pattern in sudden expansion: This part of research, compare the result of flow pattern in a sudden expansion (90 degree angle from centerline of canal) canal with FLUENT model and developed mathematical model by Zhou (1995). The entrance canal is 1.4 m wide and 14 m long and the expanded canal is 2.8 m wide and 14 m long. There is no bed slope in the canal. The upstream entrance velocities are u = 0.63 m/s, v = 0 and the flow depth y = 0.2 m as the downstream boundary condition is considered (Fig. 1).

In accordance to Fig. 2, the velocity inlet along with inlet turbulence intensity and hydraulic depth are the upstream boundary condition and outflow is the downstream boundary condition. Boundary condition for bottom and side wall of canal is wall. Symmetry boundary condition is used for top of the canal. Hydraulic flow field details table is shown in Table 1. Turbulent and under critical flow is established in canal. Due to the turbulent flow, turbulence models have been used. SIMPLE algorithm for coupling the velocity field and pressure is used.
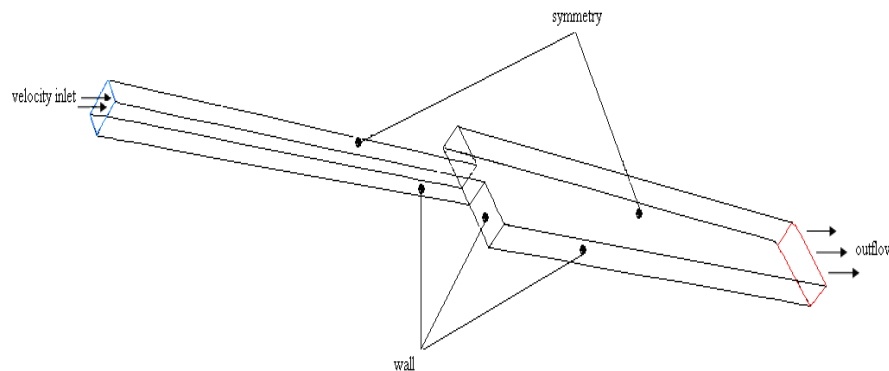

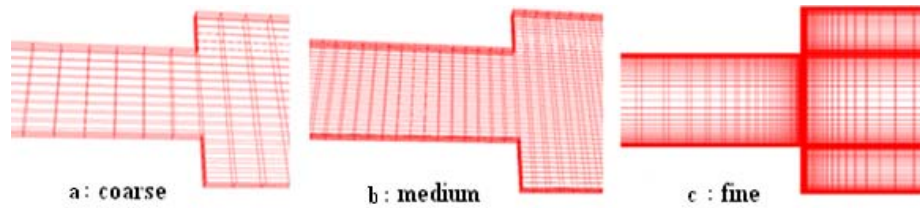Fig. 1: Canal for simulation (Zhou, 1995)

Fig. 2: (a) Coarse mesh: 9726 node, (b) medium mesh: 68651 node, (c) fine mesh: 270291 node
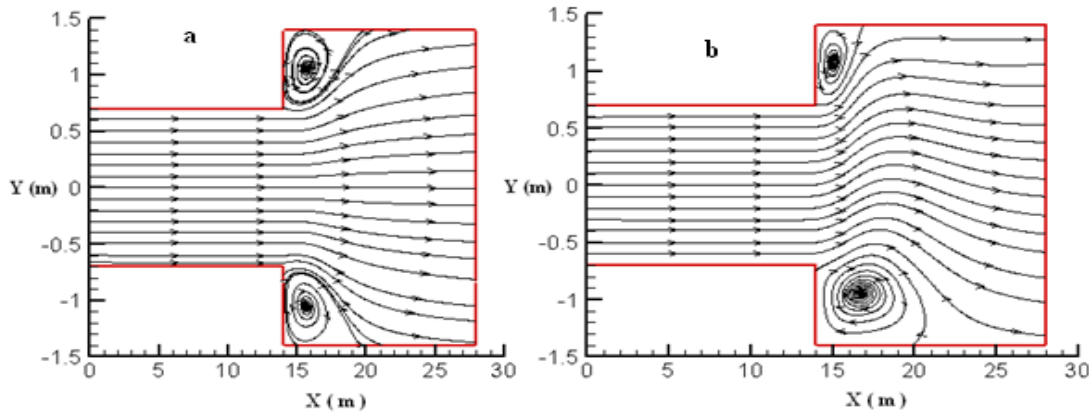

Fig. 3: Flow pattern in canal coarse mesh size: (a) after 400 iteration, (b) after 2000 iteration

Table 1: Hydraulic properties of canal

Turbulence intensity	Hydraulic depth (m)	Re	Fr	V (m/s)	Q (m3/s)
0.03	0.2	126000	0.45	0.63	0.1764

Desired canal with coarse, medium and fine mesh size is selected for simulation. Coarse and medium mesh size are $\Delta x = \Delta y = 0.28$ m and $\Delta x = \Delta y = 0.14$ m . Fine mesh size was carried out in compliance with first node spacing criteria from the wall. The first node spacing is calculated for $k - \varepsilon$ and RSM equal to 0.00136 m. Figure 2 shows the three types of coarse, medium and fine mesh size for problem solving.

Figure 3 showed the results of the coarse grid model after 400 and 2000 iteration. Flow pattern formed after 400 iteration in the Fig. 4a consists of two symmetrical vortex on both sides expansion and a high flow velocity in the central line will be gradually slowed down in x direction. Figure 4b showed that after 2000 Iteration, the flow diverted to one side and an asymmetric flow pattern occurred. Asymmetric flow pattern in this case are due to coarse mesh and also added errors for higher iterations. It should be noted that these results is similar to Zhou (1995) results.

Figure 4 showed the results of model for the medium and fine mesh. Flow pattern consists of two symmetrical vortex on both sides expansion and a high

flow velocity in the central line will be gradually slowed down in x direction.

Figure 5 showed comparing the results of model for coarse, medium and fine mesh. Longitudinal and transverse axis represents the canal width and velocity magnitude respectively. During the 9 m of canal (Fig. 5a) Results do not differ much, but in sudden expansion place for example 15 m (Fig. 5b), coarse and medium mesh size do differ much because of wall function criteria and the accuracy of their answers is slightly low. Also coarse mesh size showed the higher velocity magnitude in the centerline of canal rather than medium and fine mesh size.

Flow pattern simulation in laboratory flume with sudden expansion: In order to examine more closely the flow pattern in the sudden expansion, a laboratory flume is constructed and a series of experiments were performed on it. Then flume is studied with fluent numerical model. The flume consists of two sections. Section 1 (River section) have 5 m Length, 16 cm width, 30 cm depth and no bed slope. Section 2
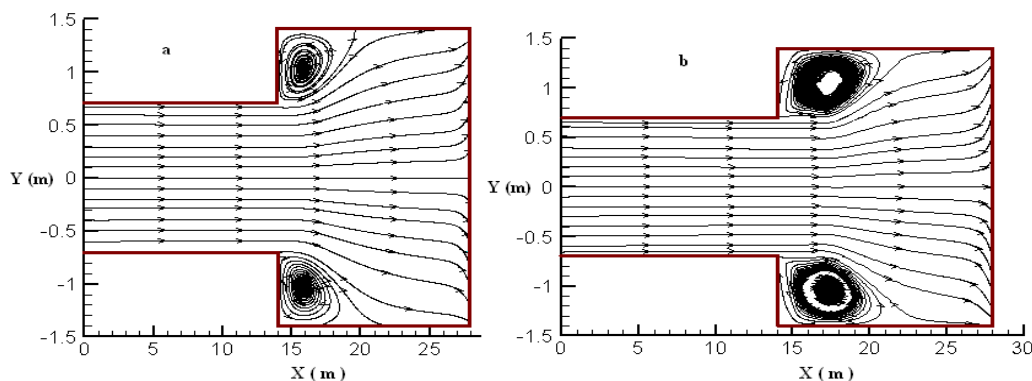
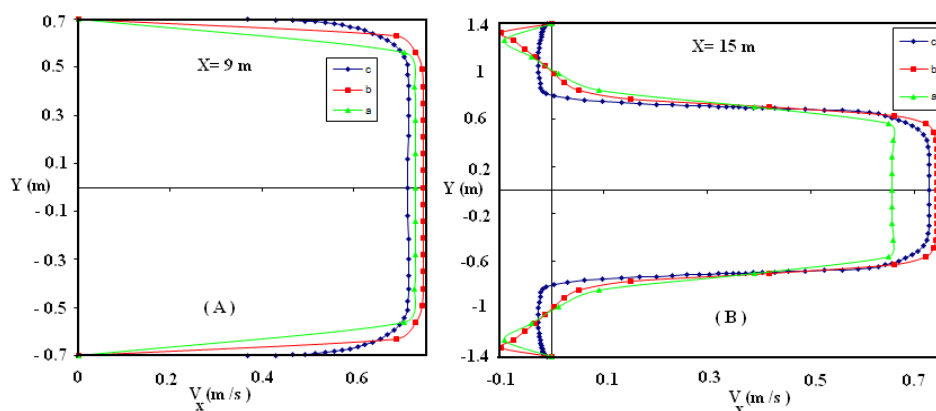Fig. 4: Flow pattern in canal: (a) medium mesh size, (b) fine mesh size

Fig. 5: Result for simulation (a) coarse mesh, (b) medium mesh, (c) fine mesh

Table 2: Hydraulic properties of flume

Turbulence intensity	Hydraulic depth (m)	Re	Fr	V (m/s)	Q (L/s)
0.03	0.3	28125	0.055	0.094	4.5
0.03	0.3	56250	0.110	0.178	9.0

(reservoir section) have 10 m length, 96 cm width and longitudinal slope 2%. At the end of flume a weir located with 45 cm elevation. Hydraulic flow field details give in Table 2. Turbulent and under critical flow in a canal is established. Due to the turbulent flow, k-ε and RSM turbulence model is used. SIMPLE algorithm for coupling the velocity field and pressure is used.

In this case, input velocity perpendicular to the boundary is used for upstream boundary condition and outflow for downstream boundary condition. In order to calculate the flow field, turbulence intensity and hydraulic diameter in upstream 0.3 and 5% is applied, respectively. Wall boundary condition for the walls and the symmetry boundary condition for top of the canal are applied. According to the first node Calculation (4 mm from bottom of canal), mesh near the wall are small and growth 20% in two adjacent cells farther from wall. Figure 6 shows the result of turbulence model in which flow field are symmetrical at different levels of depth, longitudinal and transverse direction.

Simulated flow pattern simulation with non-uniform roughness and geometry: Flow pattern formed in the laboratory flume with sudden expansion, using the numerical model was symmetrical and was asymmetric with the experimental observations. In the other hand, numerical model and experimental results did not match together. So in this part non-uniformity of roughness and geometry on the flow pattern is studied. In order to evaluate the effect of non-uniform wall roughness on the flow field, the roughness coefficient of flume in the left part is selected 50% higher than right part of the flume. For evaluation of non uniform
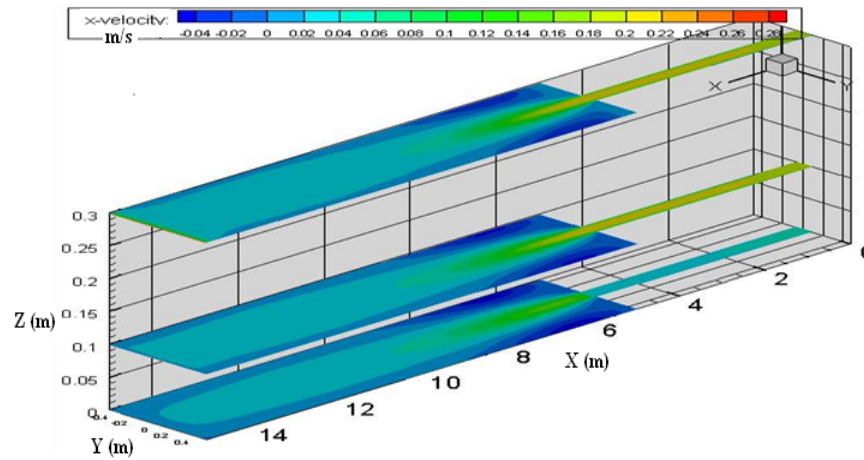
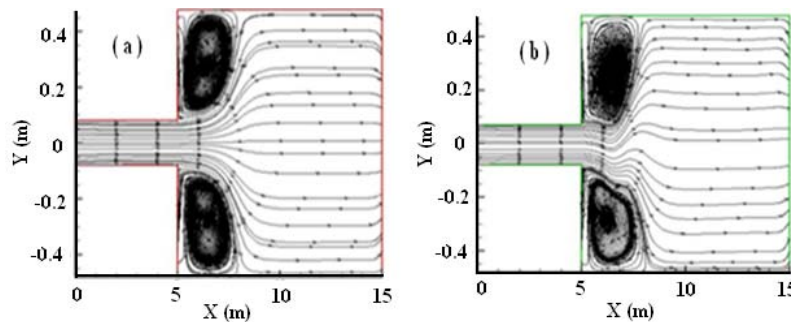Fig. 6: X velocity in flume

Fig. 7: Stream line for surface layer of flume (a) uniform roughness, (b) non uniform roughness
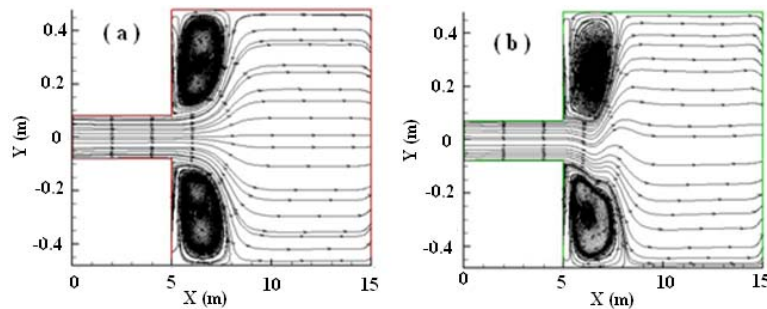

Fig. 8: Stream line for surface layer of flume (a) uniform geometry, (b) non uniform geometry

geometry, the width of canal measured from the centerline in the left part is considered 5% less than right part of the flume. Figure 7 and 8 show the streamline in the surface layer of flow for these two conditions. In the case of same roughness and geometry, river flows into the reservoir completely symmetric, while for the change of the roughness and geometry is deflected to the right. Increased roughness of the canal and reservoir, cause more resistance against the flow and the resulting flow is deflected to the right part.

CONCLUSION

In this Research, flow pattern in the sudden expansion are investigated. In this condition for medium and fine mesh size, the flow pattern was symmetrical. In the case of coarse mesh size, increasing the number of iterations cause the instability of flow and symmetric flow pattern. Also, the flow pattern in a laboratory flume with sudden expansion is studied. Symmetric flow pattern is obtained in these conditions. Finally, for study the effect of roughness and geometry

on the flow pattern, the wall of flume is changed and the asymmetry flow observed in this case. this result showed that asymmetric flow pattern could be observed due to minor changes in flume.

REFERENCES

Aloui, F. and M. Souhar, 2000. Experimental study of turbulent asymmetric flow in a flat duct symmetric sudden expansion. J. Fluids Eng., 122(1): 174-177.

Cherdron, W., F. Durst and H. Whitelaw, 1978. Asymmetric flows and instabilities in symmetric ducts with sudden expansions. J. Fluid Mech., 84: 13-31.

Escudier, M., P. Oliveira and R. Poole, 2002. Turbulent flow through a plane sudden expansion of modest aspect ratio. J. Phys. Fluids, 14(10), DOI: org/10.1063/1.1504711.

Graber, D., 1982. Asymmetric flow in symmetric expansions. J. Hydr. Div., 108: 133-140.

Graber, D., 2006. Asymmetric flow in symmetric supercritical expansions. J. Hydr. Eng., 132: 207-213.

Mamizadeh, J., 2009. The effect of entrance diverging angle and hydraulic-sediment properties on pattern of delta progress in dam reservoirs. Ph.D. thesis, Faculty of Agriculture Tarbiat Modares University, Iran.

Roul, M.K. and S.K. Dash, 2011. Two-phase pressure drop caused by sudden flow area contraction/expansion in small circular pipes. Int. J. Numer. Meth. Fl., 66(11): 1420-1446.

Roy, V., S. Majumder and D. Sanyal, 2010. Analysis of the turbulent fluid flow in an axi-symmetric sudden expansion. Int. J. Eng. Sci. Technol., 2(6): 1569-1574.

Zhou, G., 1995. Velocity depth coupling in shallow water flows. J. Hydr. Eng., 121: 717-724.