Proceedings of the 9th WSEAS International Conference on Applied Mathematics, Istanbul, Turkey, May 27-29, 2006 (pp556-565)

Comparison of Central and Upwind Flux Averaging in Overlapping Finite Volume Methods for Simulation of Super-Critical Flow with Shock Waves

Saeed Reza Sabbagh Yazdi
Assistant Professor, Department of Civil Engineering, K.N.Toosi.University of Technology, Tehran, Iran
Email:SYazdi@KNTU.ac.ir

ABSTRACT

This paper presents the comparison between the numerical results of central and upwind results of a computational model which efficiently simulate super-critical free surface flow in channels with non-parallel side walls. This model computes water depth and velocity components in horizontal plane using shallow water continuity and equations motion in horizontal plane. This model evaluates two-dimensional velocity patterns and shockwaves. The governing equations are discretized utilizing over lapping cell vertex finite volume method on triangular unstructured meshes. The numerical oscillations of explicit solution procedure are damped out using either artificial viscosity scheme or upwind averaging fluxes at control volume boundary edges. The algorithm of evaluation of the fluxes at edges and artificial dissipation terms at nodes is adopted for unstructured meshes. Using both techniques, no additional dissipation is introduced to the computed flow and shockwaves are simulated accurately. The efficiency of the model is improved using edge base data structure for addressing unstructured mesh details. The accuracy of results is assed by simulating super-critical flow in two chute canal with expanded and contracted walls and using comparison of between the computation results with the reported experimental measurements.

Keywords: "Numerical Simulation", "Upwind Flux", "Finite Volume Method", "Super-Critical Flow"

INTRODUCTION

In order to calculated flow parameters such as velocity, depth and Froude number, and to check cavitations phenomena, It is necessary to predict flow patterns in chutes. When the width of channel is expanded or contracted some steady shock waves are generated and propagates downstream in the hydraulic system. The position and direction of these waves remains constant for steady flows. The height and speed of the shock waves are important parameters in design of side walls along channels and the self aeration flow characteristics.

Many researches were reported in this regard, using various schemes of Finite element method (FEM), Finite difference method (FDM), the characteristics method; and the Finite volume method (FVM) “Zhao et al. (1996)”.

In this paper, the shallow water equations are considered for mathematical modeling of two-dimensional flows. Numerical computation of two-dimensional super-critical flow using finite volume method is performed using two methods of central and upwind flux averaging at the overlapping finite volume boundary edges. The central scheme uses artificial viscosity technique for shock capturing and damping numerical oscillations of explicit solution while with application of upwind averaging no additional terms are used. The numerical model utilized for present computations are originally developed by S.R. Sabbagh-Yazdi and here its accuracy is assessed for super-critical flow “Sabbagh and Mohammad zade (2004)”.

1. GOVERNING EQUATION

The Shallow-water equations describe unsteady gradually varied flow in open channels. The equations are obtained from integrating the Navier-Stokes equation over the channel depth “Sanders (2001) , Gharmy and Steffer (2002)”. Some assumptions to extract shallow water equations are; hydrostatic pressure distribution, incompressible flow (water), distribution of velocity in vertical direction is uniform. These equations are suitable for solving two-dimensional flow over mild slope beds.

Shallow water continuity equations equation is:

$$\frac{\partial(h)}{\partial t}+\frac{\partial(hu)}{\partial x}+\frac{\partial(hv)}{\partial y}=0$$

[1]

And two equations of motion in two horizontal directions are:

$$\frac{\partial}{\partial t}(hu)+\frac{\partial}{\partial x}(hu^2)+\frac{\partial}{\partial y}(huv)=-\frac{1}{2}gh\frac{\partial}{\partial x}(h+z)-\frac{\tau_{bx}}{\rho_0} \quad [2]$$

$$\frac{\partial}{\partial t}(hv)+\frac{\partial}{\partial y}(hv^2)+\frac{\partial}{\partial x}(huv)=-\frac{1}{2}gh\frac{\partial}{\partial y}(h+z)-\frac{\tau_{by}}{\rho_0} \quad [3]$$

Where t = time, x and y = Cartesian coordinates, h= flow depth, u and v = velocity vectores in x and y coordinates , z = bed elevation and g = gravity acceleration

Global forces are computed from following equation:

$$-\frac{\tau_{bx}}{\rho}=c_f u\sqrt{u^2+v^2}$$

[4] $$-\frac{\tau_{by}}{\rho}=c_f v\sqrt{u^2+v^2} \quad [5]$$

c_f is bed friction coefficient. In order to computed the bed stress can use roughness coefficient of Manning, Chezy, Darcy Weisbach or equivalent sand roughness.

2. NUMERICAL SOLUTION MODEL

In this paper, in order to discretize the shallow water equations the overlapping cell vertex finite volumes which are formed by some triangles meeting at each nodal point (Figure 1) are applied. The two-dimensional governing equations in the conservative form can be written as :

$$\frac{\partial Q}{\partial t}+\frac{\partial F_x}{\partial x}+\frac{\partial F_y}{\partial y}=\frac{\partial G_x}{\partial x}+\frac{\partial G_Y}{\partial y}+S \quad [6]$$

Integrating over each control volume Ω using the Finite Volume Method, the discrete form of equations takes the general form of :

$$Q^{n+1}=Q^n-\frac{\Delta t}{\Omega}\sum^{m}[(\overline{F}_X\Delta x-\overline{F}_y\Delta y)_m+(\overline{G}_x\Delta x-\overline{G}_y\Delta y)_m]-S_i\Delta t \quad [7]$$

Here $\overline{F}_x$ and $\overline{F}_y$ are the convective fluxes while $\overline{G}_x$ and $\overline{G}_y$ are hydrostatic pressure fluxes. The value of dependent variables Q at nodes can be used for computation of average fluxes at control volume boundary edges as:

$$F_x=F_x(Q) \quad , \quad F_y=F_y(Q) \quad [8]$$

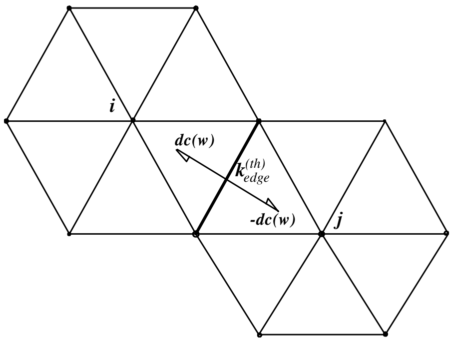


Figure 1. Two overlapping control volumes at the left and right nodes of computational edge

The above mention fluxes are usually computed by averaging of fluxes at associated nodes of control volume boundary edges. Two averaging methods are described in following sections. The first method which uses two end nodes of boundary edges of the control volumes for flux averaging procedure is identical to central schemes (for regular meshes) while the second method which uses the aforementioned node plus an upstream node is an upwind scheme.

3. THE CENTERAL FLUX AVERAGING

It can be shown that if two end nodes of control volume boundary edges are used for computation of average fluxes of relation "Younus and Chaudhry (1996)".

$$\overline{F} = (F_1 + F_2)/2 \qquad [9]$$

the resulted formulation approaches the central differencing scheme for meshes with relatively regular cell sizes. Therefore, application of this averaging method may give rise to numerical oscillation and would destroy the stability and accuracy of explicit computation procedure. To stabilize the explicit solution of hyperbolic equations adding artificial viscosity to the numerical formulation is one of the efficient techniques. The Laplacian operator ($\nabla^2 Q$) with depth switch S_h provides a suitable mechanism for damping superior numerical oscillation near shock waves.

$$\nabla^2 Q = S_h \, \varepsilon_2 \lambda_{ij} \sum_{j=1}^{Ne} (Q_j - Q_i) \qquad [10]$$

However, for the regions with smooth variations of flow variables application of Biharmonic operator ($\nabla^4 Q$) may be beneficial "Sabbagh and Mohammad zade (2003)".

$$\nabla^4 Q = \varepsilon_4 \lambda_{ij} \sum_{j=1}^{Ne} (\nabla^2 Q_j - \nabla^2 Q_i) \qquad [11]$$

λ_{ij} is a scaling factor and is computed using the maximum value of the spectral radii of every node i. The coefficient of the artificial dissipation term ε_2 and ε_4 should be tuned to minimum value required for stabilizing the solution procedure. The details of dissipation operator designed for the explicit solution of the shallow water equation on unstructured triangular meshes are described at reference paper utilized for numerical modeling of sub-critical flow "Sabbagh-Yazdi et al (2004)".

4. THE UPWINDING FLUX AVERAGING

In previous section the value of fluxes is calculate at the center of each boundary edge of control volumes. Here in order to make upwind characteristics, three point of cells attached to the boundary edges of control volume are utilized to compute the fluxes required for finite volume calculations.

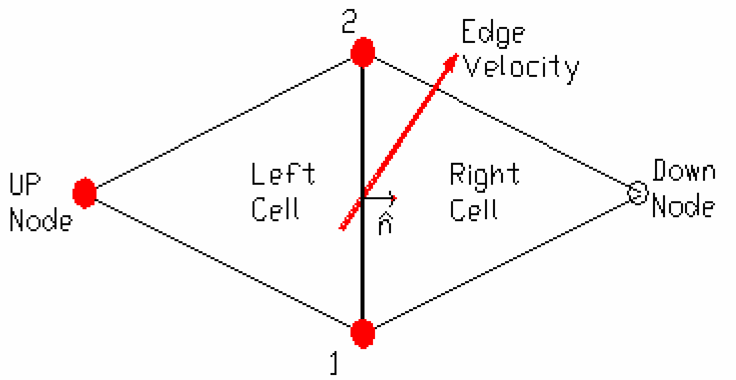


Figure 2. Two triangular cells from two neighboring control volumes sharing an edge

The value of dependent variables at three nodes (including two nodes at both ends the desired edge and upstream node.) can be used for computation of average fluxes at boundary edges as:

$$\overline{F} = \theta F_{up} + \frac{1-\theta}{2}(F_1 + F_2) \tag{10}$$

Where, the weighting coefficient θ can be chosen between 0. to 0.333. On regular meshes, above mentioned formulation with $\theta = 0$ may produce numerical results similar to second order central differencing scheme. However, if one uses $0.33 < \theta$, the scheme will produce numerical results similar to the first order schemes. Using the values of $0 < \theta < 0.33$ would result a simple oscillation free upwind scheme (with the accuracy somehow between first and second order) and hence, there would be no need to add any extra numerical dissipation to the computed residuals.

The upstream node may be considered proportional to the direction of computed value of normal velocity $\overline{U}_k$ at the edge by using following formulations. The up and down nodes can be distinguished using the sign of the dot product of unit normal and velocity vector of the edge ($\overline{U}_k . \hat{n}$, where $\hat{n} = (\Delta y\hat{i} - \Delta x\hat{j}) / \Delta s$ is unit normal vector of the edge).

In order to compute upstream convective flux, F_{UP}, the dependent variable $Q_{UP} = Q(Up\ Node)$ may be used. The upstream node may be considered proportional to the direction of computed value of normal velocity $\overline{V}_e = \overline{u}_i + \overline{v}_j$ at the edge by using following formulations. The up and down nodes can be distinguished using the sign of the dot product of $V_n = \overline{V}_e . \hat{n}$, where $\hat{n} = (\Delta y\hat{i} - \Delta x\hat{j}) / \Delta s$ unit normal vector of the edge.

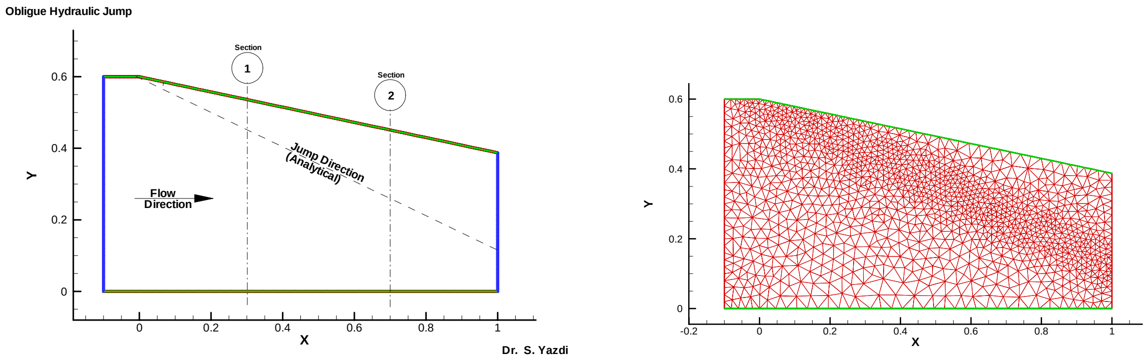
Using this simple upwind scheme will produce oscillation free results, and hence, there would be no need to add any extra numerical dissipation to the computed residuals.

5. OBLIQUE HYDRAULIC JUMP

In this section, the quality of the result of two schemes of central and upwind averaging in the model are assessed by comparison between the numerically simulated flow parameters of oblique hydraulic jump in a canal with a contraction in the left side wall of the canal. For present flow simulation the details of the test case are considered similar to the case utilized by previous researchers “Jimenez et al. (1998)”.

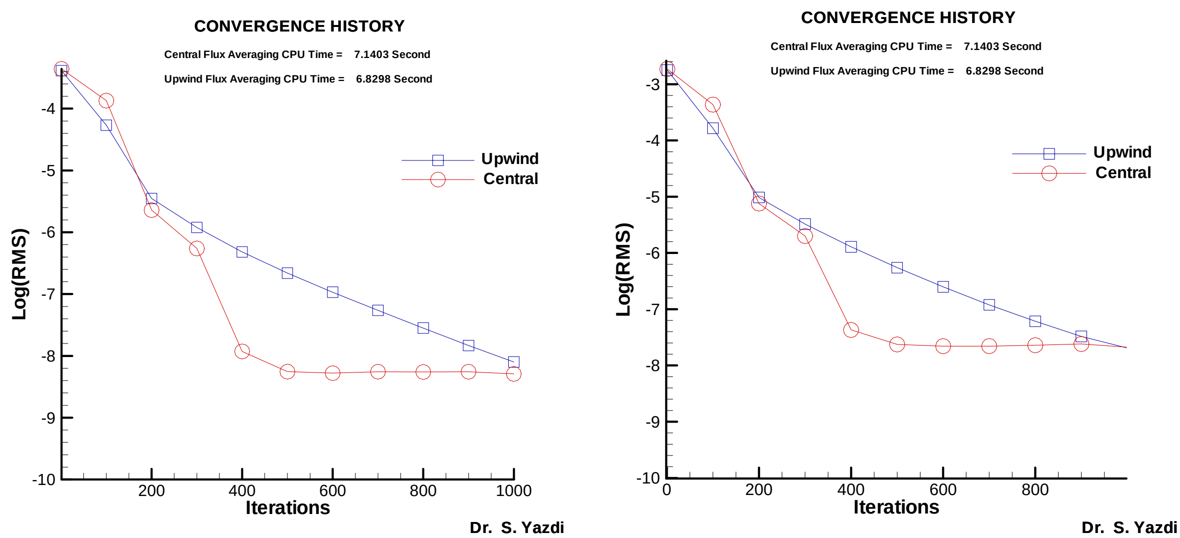
The canal with certain aspect ratio (B/h1>6, L/h1>11.8 where h1=0.1) is used to generate unstructured triangular mesh. The canal is contracted by diverting the left side wall of the canal by a constant angle of 12 degree (Figure 3). The method Delaunay triangulation is used to generate the unstructured mesh. This method provides the ability of local refinement of the mesh by considering points and line sources “Weatherill et al. (1994)”. Since the angle of the hydraulic jump from the analytical solution is known (β=25.505 degree), the mesh is fined along a line with similar angle. The finalized unstructured mesh for present numerical simulation contains 1376 nodes, 2585 cells and 3960 edges (Figure 3).

The upstream Froude number is considered as 4. Therefore, the upstream flow depth and velocity are considered as *0.1m* and *3.982m/s*. Figure 4 present the convergence behavior of two flux averaging schemes. Figure 5 demonstrates computed depths from central and upwind flux averaging schemes. For the computation of this case (using both schemes) on a Pentium IV (*2.4 MHz*) Personal Computer the *CFL=3* are used and CPU time consumption were measured as *7.14* and *6.82* seconds for central and upwind flux averaging schemes, respectively. It worth noting that according to the analytical solution for $\beta = 25.505$ degree the depth and velocity after jump can be calculated using the ratios of *h2/h1=1.987* and *v2/v1 = 0.9282*, respectively. In figure 6, the comparison between the reported analytical depth and numerically simulated water surface (using central and upwind flux averaging) along some cross sections of the canal are plotted.



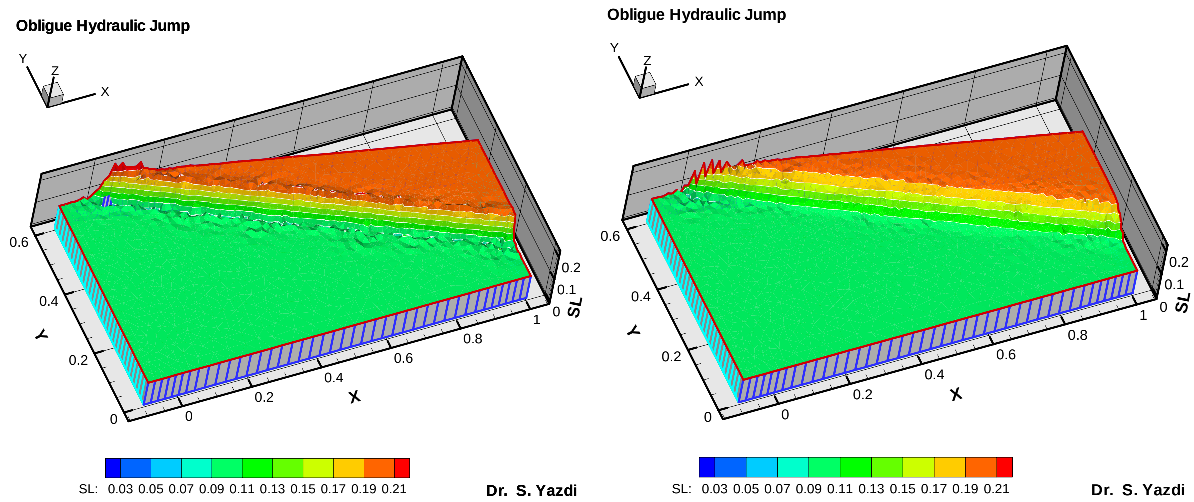
a) Geometry *b) Mesh*

Figure 3. Geometry of the canal and Unstructured triangular mesh for the oblique jump (dimensions in meter)



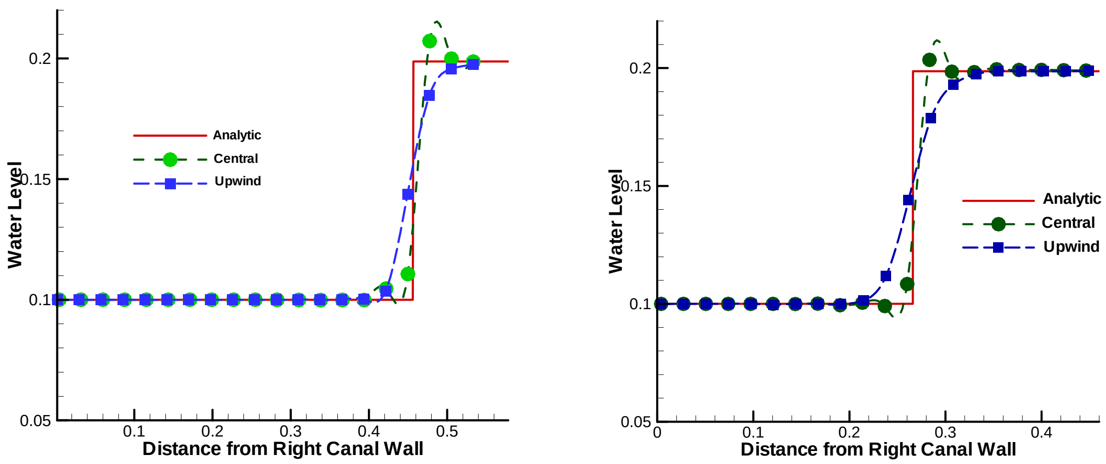
a) Depth *b) Velocity*

Figure 4. Convergence history of depth and velocity values for two flux averaging schemes (Oblique Jump)



a) Central flux averaging scheme *b) Upwind flux averaging scheme*

Figure 5. Computed water surface contours in oblique jump canal



a) Section 1 *b) Section 2*

Figure 6. Comparison of computed water surface with the analytical solution in two sections

6. NUMERICAL APPLICATIONS

In this section, in order to evaluate the computational results of two central and upwinding schemes to simulate free surface flow in channel; data of the results of the model are compared with a experimental test measurements of the previous workers on super-critical flow in two canals with contracting and expanding width “Kruger et al. (1998), Younus et al (1994)”. In these canals, flow regimes are super-critical, thus two parameters of flow (depth and velocity) are imposed at the upstream of the canals.

In following super-critical flow cases, depth and velocity of the flow in upstream are imposed at upstream flow boundary. Therefore, the flow parameters are computed at downstream of the channel by numerical model.

6.1. CONTRACTED CANAL WITHOUT FRICTION

In this part, the quality of the result of two schemes of central and upwind averaging in the model are assessed by simulation of flow in a contracted canal. The dimension of the canal is present at figure 7. The upstream flow depth and velocity are considered as 0.05m and 2.8m/s in the experimental set up report “Kruger et al. (1998)”.

The utilized unstructured mesh for present numerical simulation contains 2340 nodes, 4417 cells and 6756 edges (Figure 7).

Figure 8 present the convergence behavior of two different schemes of flux averaging. Figure 9 demonstrates computed depths from central and upwind flux averaging schemes. Figure 10 presents the comparison between the reported measured depth and numerically simulated water surface along the central are at the canal using central and upwind flux averaging.
For the computation of this case (using both schemes) on a Pentium IV (2.4 MHz) Personal Computer the CFL=3 are used and CPU time consumption were measured 23.08 and 22.26 seconds for central and upwind schemes, respectively.

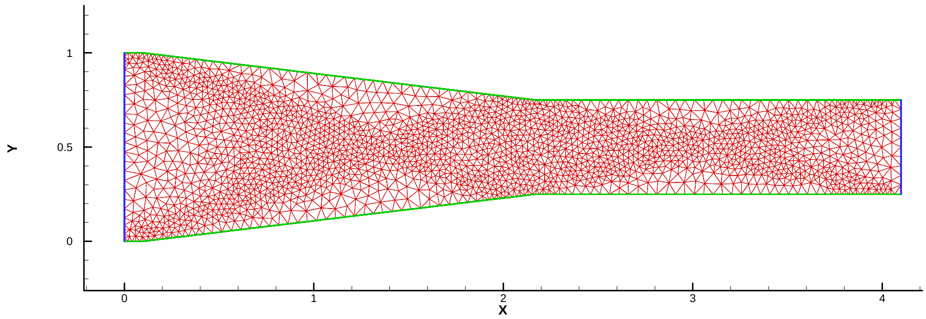
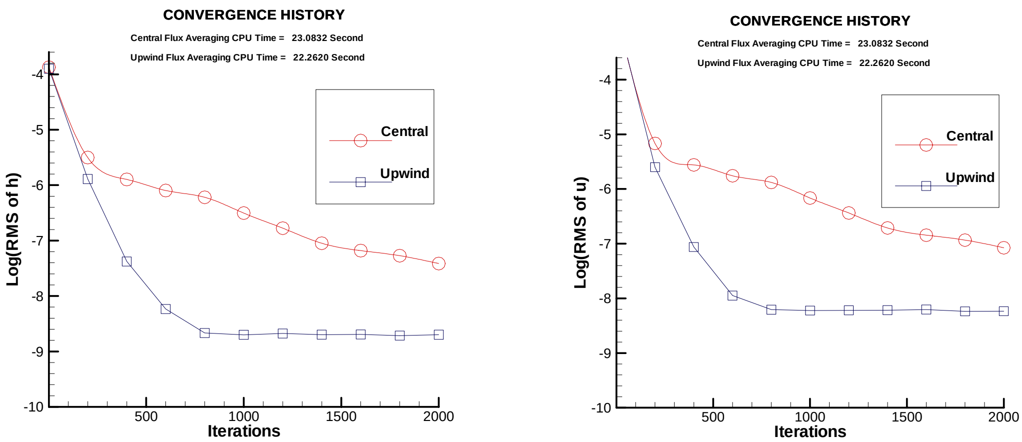
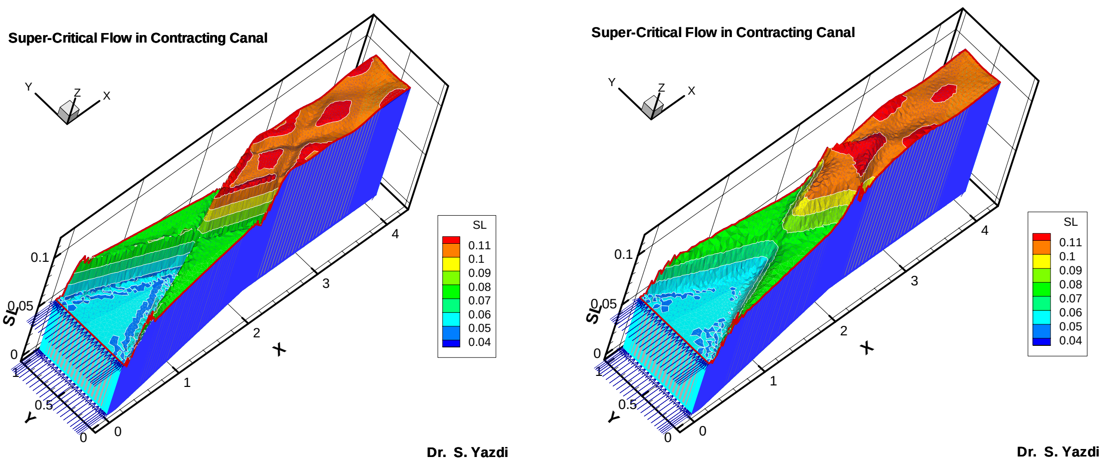


Figure 7. Unstructured triangular mesh for contracted canal (dimensions in meter)



a) Depth *b) Velocity*

Figure 8. Convergence history of depth and velocity values for two flux averaging schemes (contracted canal)



a) Central flux averaging scheme *b) Upwind flux averaging scheme*

Figure 9. Computed water surface contours in contracted canal for upwind flux averaging scheme

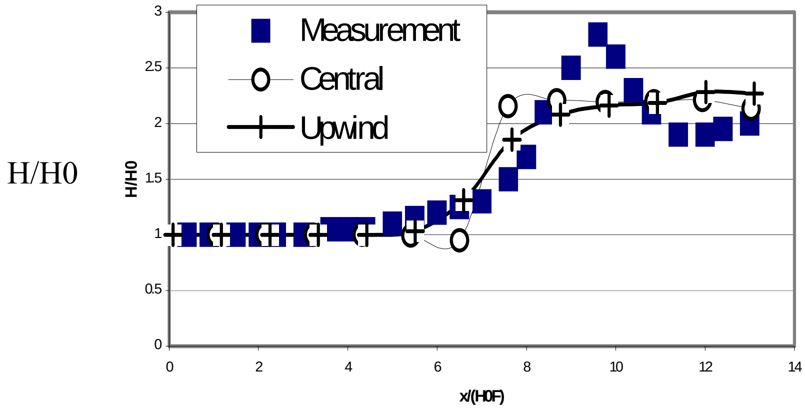


Figure 10. Water surface plots along center of contracted canal

6.2. EXPANDING CANAL WITH FRICTION

In order to verify the accuracy of the super-critical flow simulation in a canal with expanded walls a canal that its geometry is presented in figure 11 is chosen. The bed surface Chezy coefficient is reported as 70 and the inflow velocity and the upstream depth are measured as 1.4 *m/s* and 0.049 *m*. respectively according to the previous experimental work on the similar canal " Younus and Chaudhry (1994)".

For numerical simulation of this case an unstructured mesh containing 3071 nodes, 8952 edges and 5882 cells is utilized (Figure 11). Figure12, shows the convergence history flow depths for central and upwind flux averaging schemes. Figure 13, present some sample results of two abovementioned schemes of the finite volume model. In figure 14, the results of computed water surfaces along the center of the canal resulted from both schemes are plotted and compared with the experimental data.

It worth nothing that, the computation of the case (using both schemes) were performed by considering CFL=3 and CPU time consumption were measured 46.87 and 44.87 seconds on a Pentium IV (2.4 MHz) Personal Computer, for central and upwind schemes, respectively.

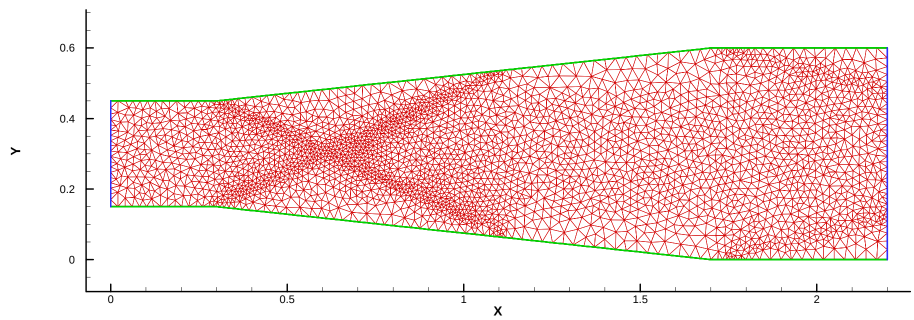
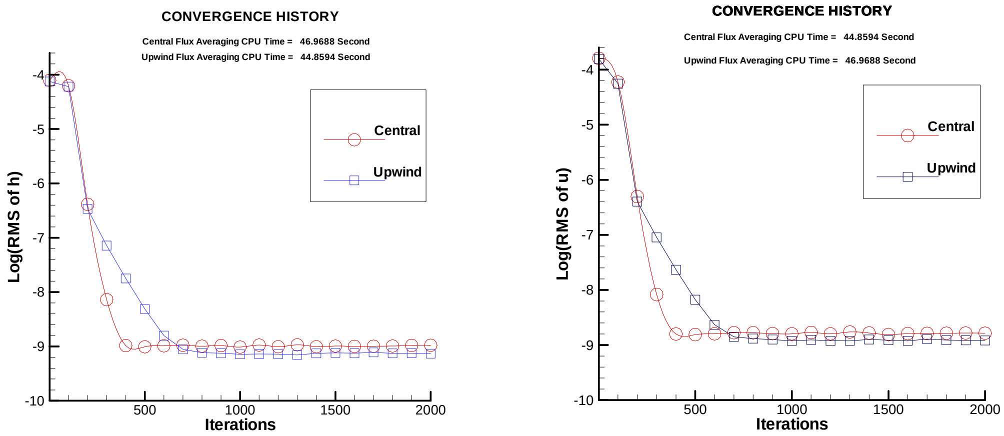
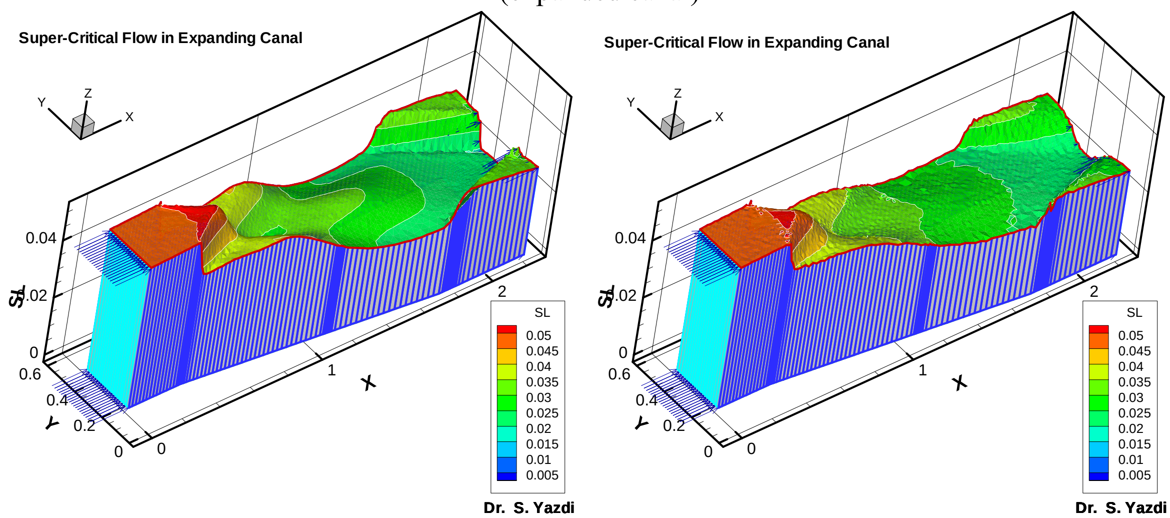


Figure 11. Unstructured triangular mesh for expanded canal (dimension in meter)



a) Depth *b)Velocity*

Figure 12. Convergence history of depth and velocity values for two schemes of flux averaging (expanded canal)



a) Central flux averaging scheme *b) Upwind flux averaging scheme*

Figure 13. Computed water surface contours in expanded canal

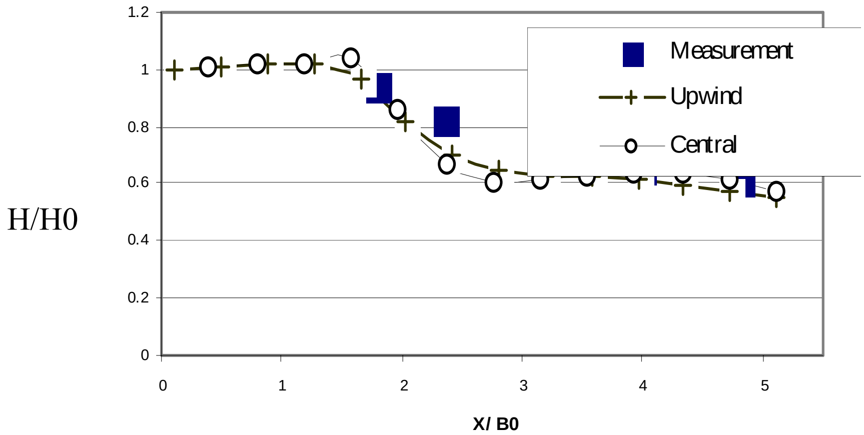


Figure 14. Computed depths and experimental measurements along center of expanded canal

7. CONCLUSION

Using depth averaged equations of continuity and motions are used as mathematical model for numerical simulation of two-dimensional super-critical free-surface flow in channels with non-parallel walls. The overlapping cell vertex finite volumes formed by gathering triangles connected to each computational node are used to discretize the governing equations. In this paper, the effects of non-parallel (contracted and expanded) walls are numerically investigated and shock waves similar to experimental observations are computed. The numerical oscillations due to explicit procedure of computations are damped out by application of two different methods of central and upwind flux averaging. The central averaging scheme needs adding the artificial dissipation terms while the upwind averaging without any additional formulation provides relatively accurate results via robust and stable solution procedure. For two-dimensional flow problems the numerical model successfully simulated the super-critical flow in contracted or expanded canals and the comparison with reported experimental works presents accepted agreement.

8. ACKNOLEDGEMENT

The finical support of this research work is provided by Water Resources Management Organization (Iranian Ministry of Energy) under contract No. DAM2-81171.

REFERENCES

1. Ghamry.Haitham, Steffler.Peter. (2002),"Two Dimensional Vertically Averaged and Moment Equations Rapidly Varied Flows". *Journal of Hydraulic Research*, Vol.40, No.5.
2. Jimenez O. F. & Chaudhry M. H. (1988), "Computation of Supercritical Free-Surface Flows", Journal of Hydraulic Engineering, Vol. 114, No.4, pp. 377-395.
3. Kruger, S . M, Burgisser& P.Rutschmann. (1998)," Advances in Calculating Supercritical Flows in Spillway Contractions ", *Hydro-informatics 98*,Bavic & Larsen Balkema ,Rotterdam ,ISBN 90 5410 983 1.
4. Sabbagh Yazdi. R. S. & Mohammad Zadeh Qomi . M.(2004),"Finite Volume Solution of Two-Dimensional Convection Dominated Sub-Critical Flow Using Unstructured Triangular Meshes", *Intrenational Journal of Civil Engineering* , Volume 2 , No. 3.
5. Sabbagh Yazdi. R. S. & Mohammad Zadeh Qomi . M. (2003),"Using 2D Unstructured Mesh for Numerical Simulation of Free Surface Flow in Meandering Canal", *Proceeding of Shallow Flow* TU Delft Netherlands.
6. Sanders, B. (2001),"High-Resolution and Non-Oscillatory Solution of the St.Venant Equations in Non-Rectangular and Non-Prismatic Channels.", *Journal of Hydraulic Research*, Vol.39,No.3.
7. Younus M. & Chaudhry M. H. (1994)," A Depth-Averaged k-ε Model for the Computation of Free Surface Flow", *Journal of Hydraulic Research*, Vol.32, No.3, pp. 415-444.
8. Zhao D.H., Shen H. W., Lai J.S., Tabios G.Q.(1996),"Approximate Riemann Solvers in FVM for 2D Hydraulic Shock Wave Modeling", *Journal of Hydraulic Research*, P. 692-702.
9. Weatherill N. P. , Marchant M. J., Hassan O. and Marcum D. , " Grid Adaptation Using a Distribution at Sources Applied to Invicid Compressible Flow Simulation", Int. J. for Num. Meth. in Fluids, 19 , pp. 739-794, 1994.