Multi-layer simulation of circulations in an artificial shallow lake

S.R. Sabbagh-Yazdi, N.E. Mastorakis, and H. Arabi

***Abstract*—** In the current study a multi layers numerical model is introduced for modeling shallow water flows. The model numerically solves conservative equations of continuity and motions in order to compute water depth and velocity patterns in each layer. The model can consider the elevation variation of upper layers and bed and wall geometric complexities and resistances. It can also consider the effect of wind as well as evaporation and rain on surface layer. The governing equations are discretized using vertex base overlapping finite volume method in triangular unstructured meshes. For stabilizing the explicit solution process, artificial viscosity formulations are adopted for the unstructured meshes in such a way that preserves the accuracy of numerical results. The accuracy of results of present multi-layer flow solver is assessed by simulating wind induced flow in a circular basin and comparison of computed results with the results of previous research works. The application of the model to a real world environmental problem is presented as well.

***Keywords*—**Vertex Base Finite Volume Method, Multi-Layer Algorithm, Conservative Shallow Water Equations, Flow in Lake with Complex Geometry

I. Introduction

In artificial and natural lakes, the effect of wind on the water surface, external and internal parameters like wind, evaporation and rain, inflows and outflows, bed and wall resistances to form the streamlines is considerable. Prediction of flow patterns plays an important role in improvement of the design and planning for to preserve the ecological system and environment. Thus, before finalizing the design of an artificial lake, performing modeling tests for prediction of flow patterns is an essential task. The availability of powerful computers and advances in mathematical simulation technologies motivate the use of numerical solution of the fluid flow problems as an efficient and accurate means of the modeling.

One of the basic assumptions in mathematical modeling of shallow water flows is considering hydrostatic pressure distribution. This assumption is reasonable for the flow in lakes that their horizontal dimensions are considerably larger than water depth. Using the hydrostatic pressure assumption and integration of the incompressible continuity and momentum equations end up with two-dimensional depth averaged flow equations known as shallow water equations (SWE).

Interaction of wind effects and geometrical features of the lakes (i.e. bathymetry and coastal irregularities) plays important roles in formation of wind induced flow patterns. Therefore, development of a finite volume SWE solver with an algorithm suitable for unstructured meshes is considered in this work.

For desired modeling algorithm, the computational work load may increase due to following reasons. First, the computational work load for the solution of the equations of the layers which is linearly increases by increasing the number of layers. Second, direct referencing to the element connectivity and coordinated definition of the nodes of unstructured meshes stored in the computer memory. Third, the non-linear reconstruction method (for transferring computed values at cell centers to nodal points at cell boundaries), which is an accuracy requirement in the most finite volume schemes.

In order to reduce the computational work load, a reconstruction free vertex base overlapping finite volume algorithm is utilized in the present multi layer SWE solver. The solution domain is discretized by application of Deluaney method which easily converts a flow domain with irregular shape into unstructured mesh of triangles. In order to damp out unwanted numerical oscillations and stabilize the explicit solution without degradation of the results, the artificial viscosity terms are added to the discrete formulation. The blend of Laplasian and Biharmonic operators are constructed in a form suitable for unstructured triangular meshes which do not introduce unwanted physical damping to the solution results.

The performance of the model is demonstrated for simulating a circular basin flow and accuracy of results checked with the available data. Then, developed multi-layer flow solver is applied to a real world problem and its abilities of the model to deal with a case with irregular geometrical features and complicated flow and boundary conditions are presented, and its capability to serve as a powerful means of modeling for environmental studies and researches are demonstrated.

Manuscript received June 30, 2007: Revised version received Sept. 19, 2007.

Saeed-Reza Sabbagh-Yazdi is Associate Professor Civil Engineering Department of K.N. Toosi University of Technology, 1346 Valiasr St. Tehran, IRAN (phone: +9821-88521-644; fax: +9821-8877-9476; e-mail: SYazdi@kntu.ac.ir, web site: http://sahand.kntu.ac.ir/~syazdi/).

Nikos E. Mastorakis, is Professor of Military Institutes of University Education (ASEI) Hellenic Naval Academy, Terma Chatzikyriakou 18539,Piraues, GREECE.

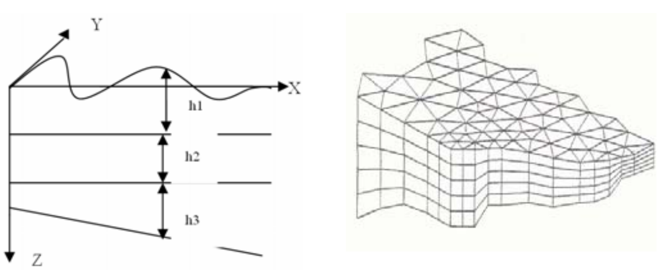
Hamed Arabi is MSc. Graduate in Civil Engineering (e-mail: Hamed_ Arapi1 @ yahoo. Com).

II. Multilayer Strategy

In some cases of the real world lake flow problems (i.e. shallow lakes with considerable bed roughness which are subject to wind on water surface) vertical variations of in horizontal velocity components are significant. In such cases, although the vertical momentum is mostly due to hydrostatic pressure, consideration of vertical gradients of the horizontal momentum is an essential task. This is particularly important if modeling the time variation of some parameters like heat, salt and density and other materials which may be solved or suspended in the water are interested. The multilayer solution of SWE is one of the techniques which can be used for numerical solution of such cases. Therefore, the layer averaged equations of continuity and motions may be chosen as the mathematical model. If the effect of momentum in vertical direction is negligible, the pressure distribution in each layer can be approximated by hydrostatic pressure.

Some of the works on numerical multi layer simulation of the lakes flow could be referred to Ezar & Mellor in 1993 which used $k-\varepsilon$ turbulence model in multi layer form for analyzing the lakes flow [1]. Mellor et al. in 1998 analyzed the two dimensional model in sigma coordinate considering the effects of advection and diffusion, density and pressure gradients considered [2].

In general, two types of layering technique are commonly used in the literature. The first one is known as sigma coordinate system and the second is variable-layers system. In sigma coordinate system the depth of the flow is divided to constant number layers and in each step the depth of flow is divided to the fixed number of layers (Fig.1a). In variable layer system, thickness of the layers is fixed, but regarding to solving situation the number of layers may be changed (Fig.1b). Although the variable layers system is more simple than the sigma coordinate system, it is would associate with some problems for the cases with complicated bed topography and irregularities coastal boundaries.



a) sigma coordinate system b) variable layers system

Fig. 1. Two kinds of layering technique in multilayer method

In this paper because of the characteristics of the application problem desired in this work (topography, bed slop and boundary fractures), the sigma coordinate system is used and a constant number of layers is defined for multilayer solution [3].

The effects of bed slope and upper layers weights are considered in the hydrostatic pressure term of the momentum equations of the layers. The effects of wind stress and bed global friction are considered in the momentum equations [4].

Solution method of multi layer system has two basic stages. First, model run for adequate convergence and forming the water layer surface in depth averaged single layer mode. Second, the multi layer solution is performed, in which, the depth of water on each point is divided to required number of layers, and afterwards, the boundary condition and body forces of the governing equations are imposed at each layer.

III. Governing Equations

With assumption of water incompressibility the mathematical multi layer model contains equations of continuity and motions. The layer average continuity equation for the k^{th} layer can be written in vector form as follow:

$$\frac{\partial h_k}{\partial t}+\frac{\partial h_k u_k}{\partial x}+\frac{\partial h_k v_k}{\partial y}=\pm(q_z)_k \tag{1}$$

where t is time, x and y are Cartesian coordinates, h is water depth, u and v are velocity components in x and y directions. In this equation, first term shows the changing of flow mass and the second and third terms show the transported mass of flow in x and y directions. q_z is the net balance of water volume exchange in vertical direction between the k^{th} layer and its neighboring media.

The horizontal x and y directions layer averaged momentum equations for the k^{th} layer could be written as follow:

$$\frac{\partial(h_k v_k)}{\partial t}+\frac{\partial(h_k v_k u_k)}{\partial x}+\frac{\partial(h_k v_k^2)}{\partial x}+g h_k\frac{\partial \zeta}{\partial y}= \frac{\tau_{by}}{\rho}+\left[\frac{\partial}{\partial x}h_k\left((T_{yx})_k\right)+\frac{\partial}{\partial y}h_k\left((T_{yy})_k\right)\right] \tag{2}$$

$$\frac{\partial(h_k v_k)}{\partial t}+\frac{\partial(h_k v_k u_k)}{\partial x}+\frac{\partial(h_k v_k{}^2)}{\partial y}+g h_k\frac{\partial \zeta}{\partial y}= =\frac{\tau_{ky}}{\rho}+\left[\frac{\partial}{\partial x}h_k\left((T_{yx})_k\right)+\frac{\partial}{\partial y}h_k\left((T_{yy})_k\right)\right] \tag{3}$$

Here, h_k is thickness of each layer and ζ water surface level which includes the effect of upper layer weight on desired layer [5]. τ_{kx},τ_{ky} representing the x and y directions global stresses of the k^{th} layer, including the effect wind stresses τ_{tx},τ_{ty} on the upper layer, τ_{bx},τ_{by} representing the effect of the bed roughness in the lower layer, τ_{lx},τ_{ly} representing the effect of the interference surfaces between the layers. These stresses can be defined from the flowing relations [6].

The interface friction between layers k and $k+1$ must be determined especially when the density or viscosity of layers are different from each other, otherwise the shear stress between the layers is negligible. Considering the following equation, the effect of friction between layers can be defined.

$$\tau_{lx} = C_l (u_{k+1} - u_k)\left|(\sqrt{u^2+v^2})_{k+1} - (\sqrt{u^2+v^2})_k\right| \quad (4)$$

$$\tau_{ly} = C_l (v_{k+1} - v_k)\left|(\sqrt{u^2+v^2})_{k+1} - (\sqrt{u^2+v^2})_k\right| \quad (5)$$

where, C_l is friction coefficient of the two layers, u_k and u_{k+1} are horizontal velocity components in x direction and v_k and v_{k+1} are horizontal velocity components in y direction.

Alternatively, for the cases that the densities or viscosities of the layers are not different, the effects of wind and bed roughness can be interpolated between the momentum equations of all the layers.

The wind stresses that should be added to the above mentioned x and y directions global stresses of the top layer are defined as,

$$\tau_{tx} = \rho_a C_\omega W_x |W_{10}| \quad (6)$$

$$\tau_{ty} = \rho_a C_\omega W_y |W_{10}| \quad (7)$$

where,. W_{10} is the wind speed at 10 m above the water surface and C_w is the drag coefficient which can be obtained from following graph.

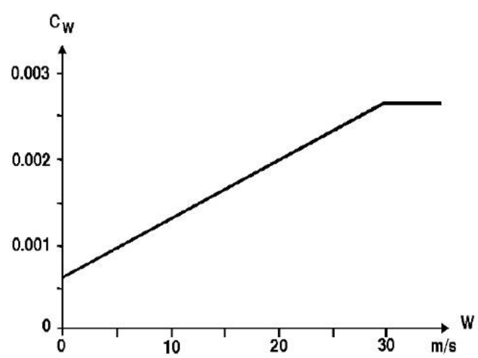


Fig. 2. Relationship between wind speed & drag coefficient [3]

The global friction due to bed roughness can be attained from the following relations:

$$\tau_{bx} = C_f u \sqrt{\left(u^2+v^2\right)_k} \quad (8)$$

$$\tau_{by} = C_f v \sqrt{\left(u^2+v^2\right)_k} \quad (9)$$

where, C_f is the effective global dissipative coefficient. In multi layer cases, the global effect of bed friction that must be added to all the layers may be calculated using following coefficient of friction,

$$C_f = \left[\frac{1}{K}\log(\frac{30Z_b}{k_s})\right]^{-2} \quad (10)$$

where, k_s is the bed friction coefficient, k is Von-Karman coefficient which equals to 0.4 and Z_b is distance from bed.

The effect of global turbulent diffusion in x and y directions are computed using the following formulation [7].

$$T_{xx} = \frac{\partial}{\partial x}\left(2\nu_t h \frac{\partial u}{\partial x}\right) \quad (11)$$

$$T_{xy} = \frac{\partial}{\partial y}\left(\nu_t h \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right)\right) \quad (12)$$

$$T_{yx} = \frac{\partial}{\partial x}\left(\nu_t h \left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}\right)\right) \quad (13)$$

$$T_{yy} = \frac{\partial}{\partial y}\left(2\nu_t h \frac{\partial v}{\partial y}\right) \quad (14)$$

where ν_t is the horizontal eddy viscosity parameter that can be computed with algebraic, Sub-Grid Scale (SGS), $k-\varepsilon$ models. In the present work, the widely used depth-averaged parabolic turbulent model is applied, in which the eddy viscosity parameter is computed algebraic formulation $\nu_t = \theta h U_*$. In this formulation the bed friction velocity is defined as $U_* = [C_f (u^2+v^2)]^{0.5}$ and the empirical coefficient θ is advised around 0.1 [8].

IV. Finite Volume Formulation

Here, the vertex base overlapping finite volume method is applied for converting the governing equations of the layers into discrete form on unstructured triangular meshes. In order to discrete the solution domain into triangular sub-domains Deluaney Triangulation method is applied [9]. In the applied solution algorithm, at the outset, the computational control volumes Ω are formed by gathering the triangles on every node. Then, the governing equations are integrated over each control volume. Application of the Green's theorem to the integrated equations results in:

$$\int_\Omega \frac{dW}{dt} d\Omega + \oint_\Gamma \left(\left(E\Delta y - F\Delta x\right) + \left(G\Delta y - H\Delta x\right)\right) = \int_\Omega S d\Omega \quad (15)$$

where,

$$W = \begin{pmatrix} h_k \\ h_k u_k \\ h_k v_k \end{pmatrix}, \qquad E = \begin{pmatrix} h_k u_k \\ h_k u_k^2 \\ h_k u_k v_k \end{pmatrix},$$

$$F = \begin{pmatrix} h_k v_k \\ h_k u_k v_k \\ h_k v_k^2 \end{pmatrix},$$

$$G=\begin{pmatrix}0\\ h_k\nu_{Th}\dfrac{\partial u_k}{\partial x}\\ h_k\nu_{Th}\dfrac{\partial v_k}{\partial x}\end{pmatrix},\quad H=\begin{pmatrix}0\\ h_k\nu_{Th}\dfrac{\partial u_k}{\partial y}\\ h_k\nu_{Th}\dfrac{\partial v_k}{\partial y}\end{pmatrix}$$

$$,\ S=\begin{pmatrix}q_{zk}\\ -gh_k\dfrac{\partial\xi}{\partial x}-\dfrac{\tau_{kx}}{\rho_w}\\ -gh_k\dfrac{\partial\xi}{\partial y}-\dfrac{\tau_{ky}}{\rho_w}\end{pmatrix}$$

And Ω and Γ are the area and perimeter of the control volume formed by gathering triangular cells sharing a computational node, respectively.

In order to solve the value of the unknown vector W at the central node of the control volumes, the above integral equation can be transformed to the following algebraic formulation.

$$W_i^{n+1}=W_i^n-\frac{\Delta_t}{\Omega_i}\left[\sum_{i=1}^{m}\left((\overline{E}\Delta y-\overline{F}\Delta x)+(\overline{G}\Delta y-\overline{H}\Delta x)\right)\right]^n+S^n \quad (18)$$

where, W_i^{n+1} is the value of W_i^n to be computed after Δt in the i^{th} node. The parameters $\overline{E}$, $\overline{F}$, $\overline{G}$ and $\overline{H}$ are the averaged values of the fluxes at boundary edges of the control volume Ω [8].

In the explicit solution of the convection dominated problems (i.e. flow with negligible friction and turbulent effects) lack of some enough physical dissipation may give rise to some numerical oscillations. This problem, which endangers the stability of the computations, usually appears near regions with high gradient of flow parameters. For damping out these numerical noises artificial dissipation terms can be added to the derived formulations. The formulation of these additional terms should suit unstructured nature of the mesh data. The dissipation terms must damp out the numerical errors without degradation of the accuracy. Although for the flows with gradual variations independent variables the fourth order term (Biharmonic operator) produces enough dissipations, near the high gradient region it is necessary to add second order operators (Laplacian operator) to the dissipation formulation. In order to prevent unwanted dissipation in the smooth flow regions, the Laplacian operator can be multiplied by a water elevation switch. Hence, the artificial dissipation operators are introduced as,

$$\left(\nabla^4 W_i\right)_k=\varepsilon_4\sum_{j=1}^{Ne}\left[\lambda_{ij}\left(\nabla^2W_j-\nabla^2W_i\right)\right]_k \quad (19)$$
$$\left(\nabla^2 W_i\right)_k=\varepsilon_2\sum_{j=1}^{Ne}\left(W_j-W_i\right)_k$$

Where, λ is scaling factor, ε_4 and ε_2 are the artificial dissipation coefficients ($1/256<\varepsilon_4<3/256$ and $0.2<\varepsilon_2<0.3$), which must be tuned to minimum value required for the stability of computations. The scaling factor λ_{ij} is computed using maximum nodal values of Eigen values of Jacobian matrix at the edges connected to the centre node of the control volume. λ is evaluating as follow.

$$\lambda_k=\left[\left|\vec{U}.\hat{n}\right|+\sqrt{U^2+C^2\left(\Delta x^2+\Delta y^2\right)}\right]_k \quad (20)$$

Where, $C=\sqrt{gh_k}$ is celerity, $\vec{U}$ is averaged computed velocity, and $\hat{n}$ is normal vector at of edges. Note that, the computations of artificial dissipation operators are performed over the edges connected to the central node of the control volume **Error! Reference source not found.**(Sabbagh-Yazdi S.R. & Mohamad Zadeh Qomi M, 2003).

In the multi layer simulation of the lake flow, particular considerations are taken into account for imposing flow boundary conditions. For example inflow to a reservoir and free surface spillway should be considered at the upper layer and bottom outlet of a reservoir is imposed at the lower layer. All the inflow and outflow volumes may be considered as source and sinks of the continuity equation at desired nodal points.

Free-slip velocity condition at coastal walls can be imposed where no flow passes through the flow boundaries. These boundaries can be used to reduce the computational domain assuming that the thickness of near wall boundary layer is considerably small in comparison with the horizontal dimensions of the problem. At these boundaries the component of the velocities normal are set to zero. Therefore, tangential computed velocities are kept using free slip condition at wall boundaries [8].

V. Verification of the Model

Numerical solution of flow in a circular basin is chosen as a test case to evaluate the model performance in simulating the effect of wind flow in a shallow lake (Fig.3).

This test is a circular basin that Krunburg in 1992 design it as shallow basin case. The circular basin has a radius of 192 meter and the depth of 0.6 meter. A steady wind directly passes from east to west of this circle and its speed is 10 meter per second. The drag coefficient is nearly equal to 0.002 [10].

In present work, the water depth is divided into three layers. The unstructured triangular meshes which models three dimensional bottom surfaces of top, middle and bottom layers are shown in figure 4.

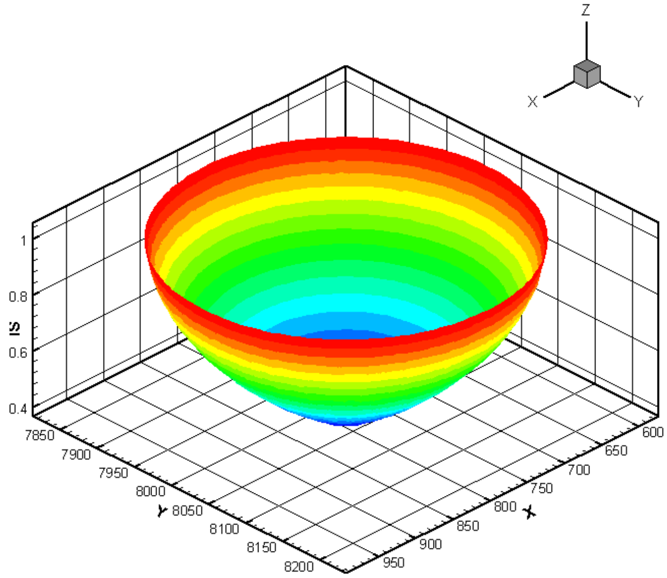


Fig. 3. Topology of the of the shallow basin

(Dimensions in meter)

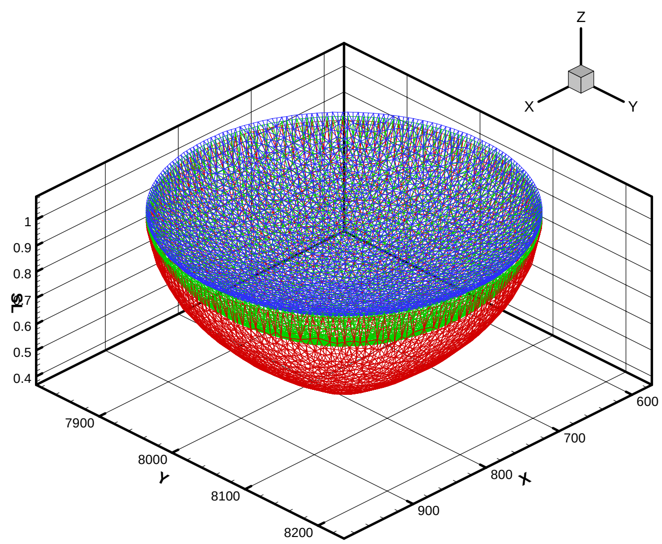


Fig. 4. Three unstructured triangular mesh layers of the basin

(Top, middle and bottom layers are colored with blue, green and red, respectively)

As there is no internal or external flow through the basin, thereafter just wind stress effects are considered in this case. Still water is assumed as the initial condition. Then, a constant wind with speed of 10 m/sec considered blowing 10 meters over the basin. This west to east direction wind formed two circulating flows in the basin. In the following figures the computed flow pattern at each layer are plotted in the form of stream lines (Fig.5) and velocity vectors (Fig.6).

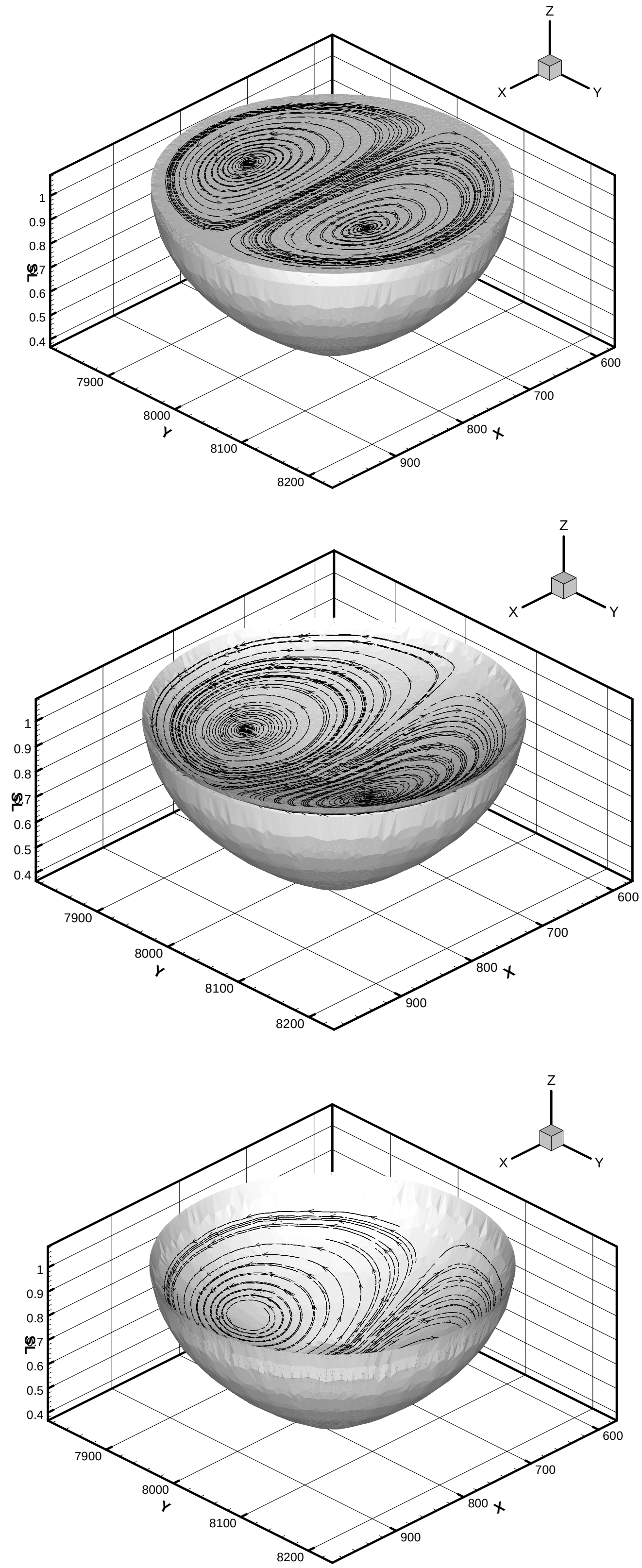


Fig. 5. Computed stream lines on surfaces of three layers

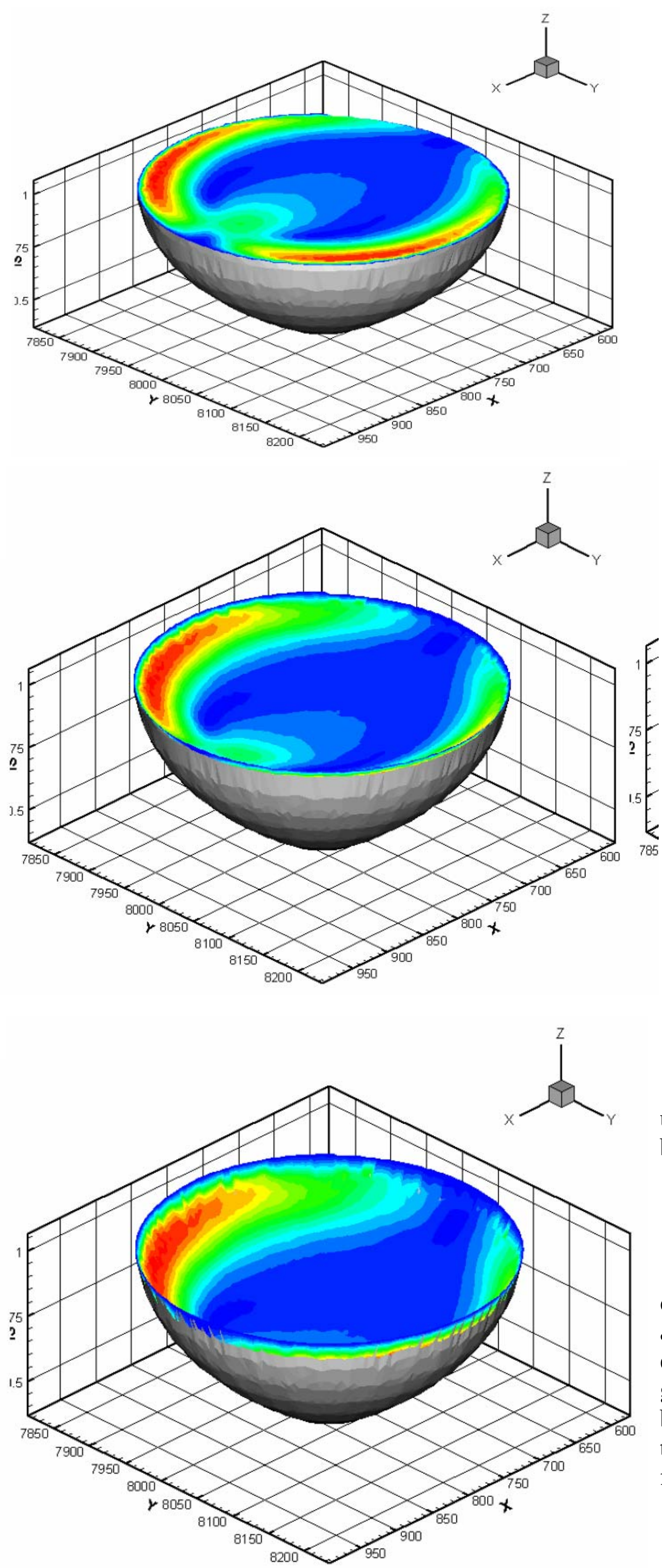


Fig. 6. Computed velocity contours on surfaces of three layers

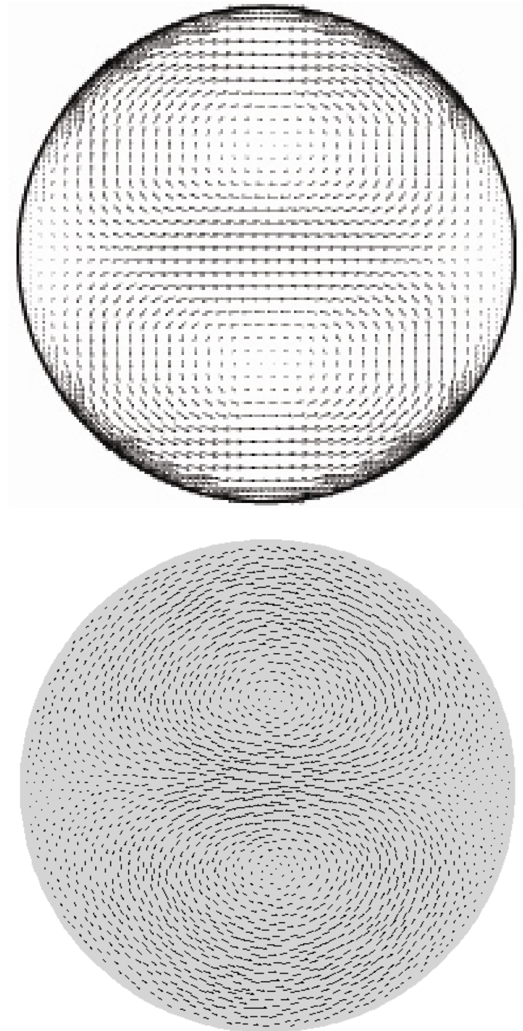


Fig. 7. Comparison between velocity vectors of the reference (top) and results of the present model on upper layer (bottom)

Figure 7 presents the results of the present model on unstructured mesh which is very similar to the results reported by previous workers on structured mesh [10].

VI. Application of the Model

Multilayer simulation flow in CHITGAR Lake is considered as an application of the developed model. This artificial lake is designed to be created by construction of an embankment dam and impermeable lining of the natural ground at west side of Tehran (Iran). The water which would be supplied from the KAN River which may be stored during the winter and spring seasons and must gradually be released from the bottom outlet (Fig.8).

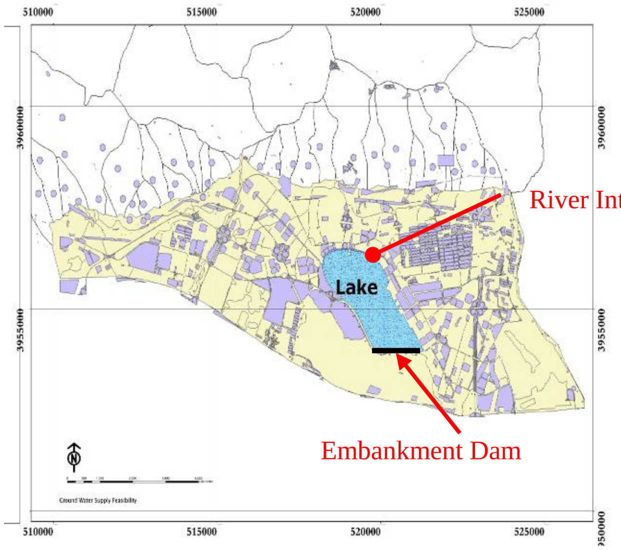


Fig.8. Location map of CHITGAR artificial lake

(at west side of Tehran, Iran)

Due to dry nature and warm weather of the region, evaporation from the water surface and weak circulation, respectively, may endanger quantity and quality of the pounded water. Hence, providing feasible intake/outlet programs that may best fit with geometrical features and climatologically situation of the lake is the key point for design control of the case.

It worth noting that the vertical length scale of the lake (H_{max}=16*m*) is considerably smaller than its vertical length scale (L_{max}=2.5*km*). Hence, the natural transport of fresh water to all positions far from the river intake point is the most important issue for preserving the uniform quality of the lake. Hence, the horizontal circulations are much important than the vertical convection, and, prediction of horizontal circulations due to in/out flows as well as wind effects one some of the main concerned issues during the design process (Fig.9).

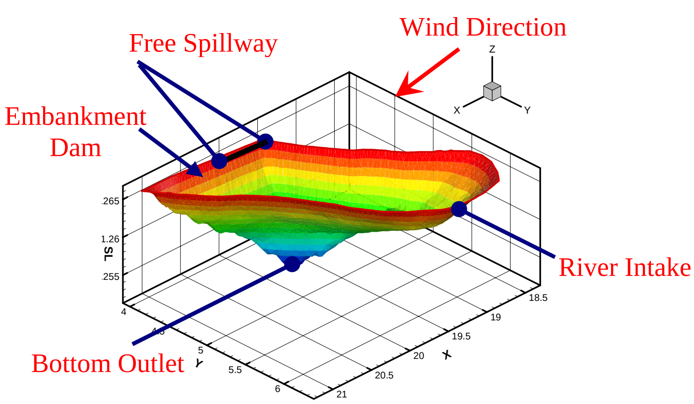


Fig.9. Geometrical features of CHITGAR artificial lake

(Dimensions in kilometers)

In order to simulate the circulating flow in this artificial lake that would provide mixing and refreshment of the water, the effects of major factors are considered. The wind effect on the water surface, inflow from river intake as well as outflow from bottom outlet and evaporations from the water surface are considered as the main factors that may cause major circulations in this lake. Due to negligible vertical velocity components, the effects of vertical momentums are ignored and hydrostatic pressure distribution in the lake is assumed. Therefore, layer averaged simulation of the case is performed using a three layers model (Fig.10).

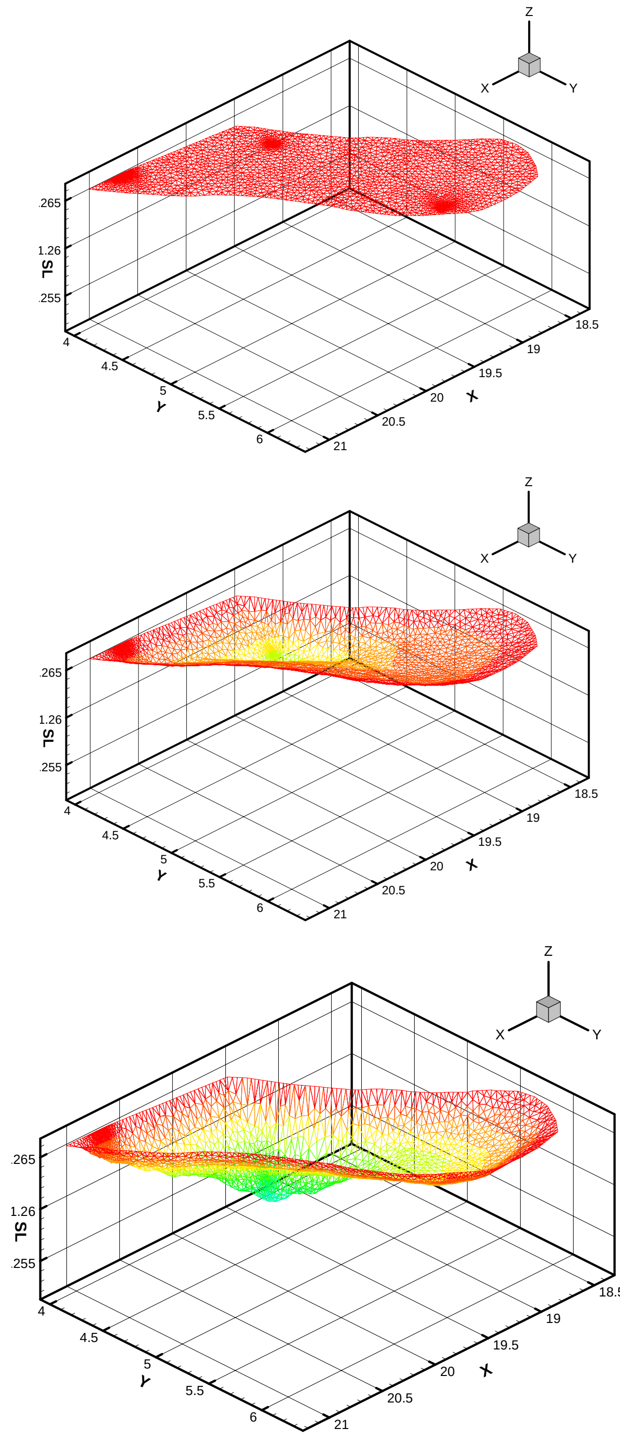


Fig.10. Unstructured triangular meshes of three layers in CHITGAR artificial lake

Considering the interaction of fresh water inflow from river intake and outflow from the bottom outlet of the lake as well as evaporation and wind effects on the water surface, the horizontal circulations due to geometrical features of the lakes are computed. The computed circulations by the present multilayer finite volume solver are plotted in form of stream traces (Fig.11).

However, for the assessment of the effects of the predicted circulations, not only the direction of the flow streams but also the power of the currents in terms of velocity magnitudes should be investigated. The color coded maps of computed velocity magnitudes are shown on the surfaces of the upper, middle and bottom layers (Fig.12).

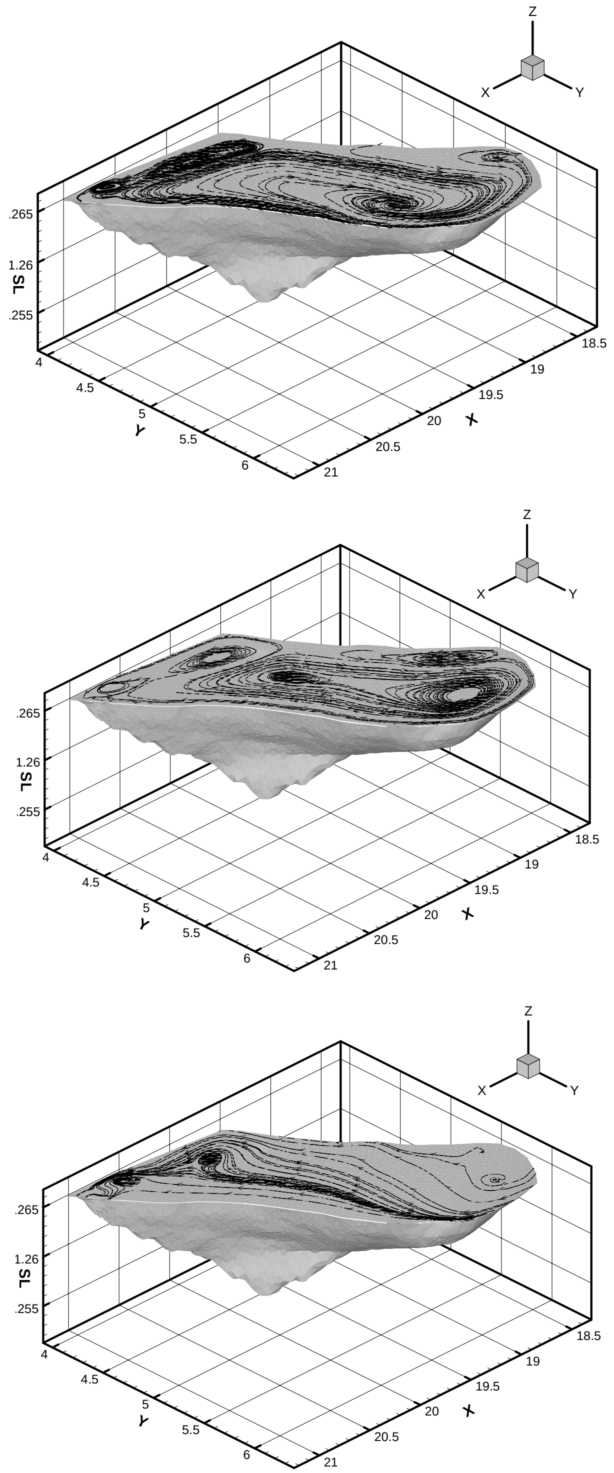


Fig.11. Stream traces on the three layers in CHITGAR artificial lake

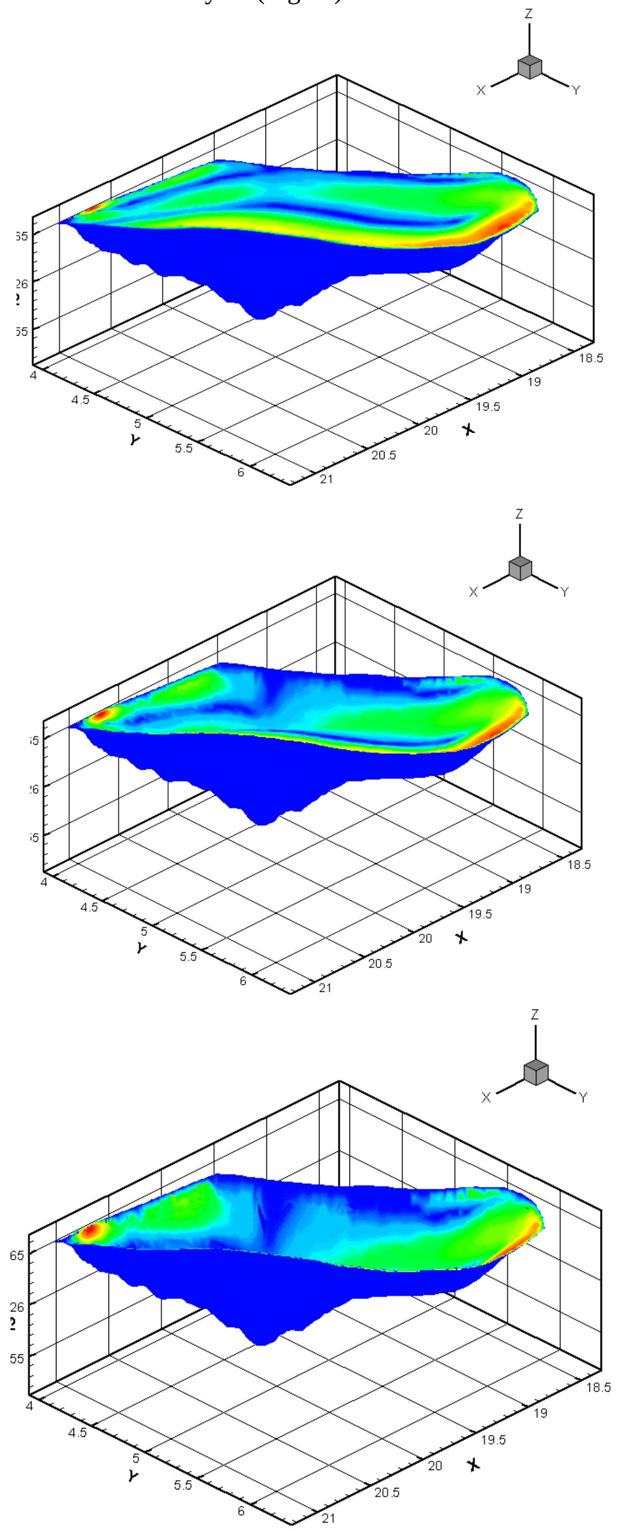


Fig.12. Color coded maps of velocity values on the three layers in CHITGAR artificial lake

VII. Conclusion

In this work, a multi-layer algorithm for vertex base overlapping finite volume solution of conservative equations of continuity and motions on unstructured meshes are presented. The layer averaged equations solved on triangular unstructured meshes using vertex base overlapping finite volume method. In this algorithm the effects wind stress is successfully imposed on water surface (top layer) and the hydrostatic pressure and global stresses are distributed over the layers.

In order to validate the results, a circular basin was chosen and the results of present model were compared with the available results of previous reported research works. Accurately and oscillation free numerical results are obtained for the flows with wide range variation in depth by application of artificial dissipation operators.

The ability of the present multi-layer flow solver to simulate horizontal circulations due to simultaneous wind and evaporation effects on free surface of the water as well as inflow from river intake and outflow from bottom outlet is investigated by its application to a real world large shallow artificial lake which is under design in Iran.

The developed model can deal with the cases with irregular geometrical features and complicated flow and boundary conditions. Since required convection diffusion equations and their source/sink terms can be easily added to the model, it may be used as a powerful tool for environmental studies and researches.

References

[1] T. Ezar, G.L. Mellor, *Model Simulated Changes in Transport. Meridional Heat & Costal Sea Level*, Department of Physics, Memorial University of New Found land, Canada, 1993

[2] G.L. Mellor, L.Y. Oey, *American Metrological Society*. Journal of Atmospheric & Oceanic Technology 15, 1998, pp11-22

[3] Reference Manual of MIKE 3/21 FM, *Estuarine and Coastal Hydraulics and Oceanography, Hydrodynamic Module*, DHI Water & Enviromental, Agem, Denmark

[4] User Guide for POM, Mellor, G., *A 3D Primitive Equation Numerical Ocean Model*, 2002,

[5] User Documentation Coherens, Release 8.4, *A Coupled Hydro dynamical Ecological Model for Regional & Shelf Seas*, 1999.

[6] M.B. Abbot, *Computational Hydraulic*. Pitman Publishing Limited, 1980.

[7] C.B. Vreugdenhil, *Numerical Methods for Shallow-Water Flow*. Kluwer Academic Publishers, 1994.

[8] S.R. Sabbagh-Yazdi, M. Zounemat-Kermani and A. Kermani, *Solution of depth-averaged tidal currents in Persian Gulf on unstructured overlapping finite volume*, International journal for numerical methods in fluids, Vol(55), 2007 , pp. 81-101.

[9] J.F. Thompson, B.K. Soni, N.P. Weatherill, *Hand Book of Grid Generation*. CRC Press, 1999.

[10] C. Kranenburg, *Wind-Driven Chaotic Advection in a Shallow Mode Lake*. Journal of Hydraulics Research 30(1), 1992, pp29-47