3D vertex-base unstructured finite volume model with bi-harmonic dissipation for turbulent wind flow around a group of cooling towers

S.R. Sabbagh-Yazdi, N.E. Mastorakis, and M. Torbati

***Abstract*—** In this paper, a three dimensional turbulent air flow solver is introduced and its accuracy for the solution incompressible flow equations in conservative form. The equation of continuity is simultaneously solved with the three equations of motion in a coupled manner by application of the pseudo compressibility technique for the steady state problems. The set of flow equations in Cartezian coordinate system is combined with a SGS (Sub-Grid Scale) eddy viscosity turbulence model. The discrete form of the three-dimensional flow equations are formulated using the vertex-base overlapping Finite Volume Method for unstructured mesh of tetrahedral cells. In order to damp out the numerical oscillations during the explicit solution procedure, a formulation for bi-harmonic artificial dissipation is used which suits the unstructured meshes of tetrahedral cells. Using unstructured meshes provides the merit of accurate geometrical modeling of the curved boundaries of the cooling towers. In order to demonstrate the performance of the developed model, the velocity and pressure fields due to wind flow around a group of three cooling towers with arbitrary arrangements, is presented. The computed results are presented in terms of color coded maps of pressure and velocity fields as well as velocity vectors on boundary surfaces of the solution domain.

***Keywords*—**Wind Flow Simulations, SGS Turbulent Viscosity Model, Tetrahedral Unstructured Mesh, 3D Vertex-base Overlapping Finite Volume Method, Cooling Towers Interference

I. INTRODUCTION

Unexpected pressure loads on such tall structures with thin walls that their geometrical features make them vulnerable against storm may cause disastrous loading condition. Hence, evaluation of the wind induced pressure load on cooling tower shells is one of the major concerns of the structural engineers.

The cooling towers neighborhood may considerably change the pressure load distributions on their shell, and, wind flow through particular arrangement of a group of cooling towers may produce unexpected pressure fields, which may cause disastrous structural loading condition.

There are some codes of practice for evaluation of wind induced external pressure on the particular structures like natural draught cooling towers of power plants. Most of the codes of practice consider a coefficient that may be applied to increase the pressure load all over the single cooling tower shell. This constant coefficient may be used either for the effect of grouping the cooling tower (with particular arrangements) on wind induced external pressure suggested for a single tower. For a group of cooling towers, the interference factor is usually defined for a certain regular arrangements and number. However, modeling of final design is recommended by most of the codes of practice.

The production of high performance digital computers and development of efficient numerical modeling techniques have ended to the development of powerful Computational Fluid Dynamics (CFD) models for solution of problems concerning fluid flow. Hence, the computer simulation of complicated flow cases has become one of the interesting areas of the research works by development of efficient and accurate numerical methods suitable for the complex solution domain. The control over properties and behavior of fluid flow and relative parameters are the advantages offered by CFD witch make it suitable for the simulation of the applied problems.

Several works on numerical simulation of steady and time dependent flow around circular cylinder using various turbulent models are reported in the literature. Murakami reviewed successful researches on numerical modeling of flow past 2D cylinders and CFD analysis of wind flow [1]. Recently, some research satisfactory numerical simulations of the flow around circular cylinders using Sub-Grid Scale turbulent model is reported by Salvett [2].

After recent success of the first author on accurate solution of two and three dimensional incompressible inviscid flow cases using unstructured meshes [3] and [4], and then acceptable solution of two dimensional turbulent flow

Manuscript received May 12, 2007; Revised received January 28, 2008.

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).

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

Mehdi Torbati is BSc graduate of Civil Engineering Department of K.N. Toosi University of Technology, 1346 Valiasr St. Tehran, IRAN (Torbati.Mehdi@gmail.com).

Bahman Haghighi is Head of Structural Design Office of BOLAND-PAYEH Co.No.3, 21th Street, Argentina Square, Tehran IRAN (B_Haghighi@BolandPayeh.com)

problems [5], motivates the present completion of the software for simulating three dimensional incompressible turbulent flow cases. In the software developed in this work, the governing equations for incompressible wind flow are solved on unstructured finite volumes. By application of the pseudo compressibility technique, the equation of continuity can be simultaneously solved with the equations of motion in a coupled manner for the steady state problems. This technique helps coupling the pressure and the velocity fields during the explicit computation procedure of the incompressible flow problems. The Sub-Grid Scale model is used to compute the turbulent eddy viscosity coefficient in diffusion terms of the momentum equations. The discrete form of the three-dimensional flow equations are formulated using the Vertex-base Overlapping Finite Volume for unstructured mesh of triangles. In order to damp out the numerical oscillations during the explicit solution procedure, a formulation for bi-harmonic artificial dissipation is used which suits the unstructured meshes of tetrahedral cells.

In this paper, the development of a three-dimensional Vertex-base Overlapping Finite Volume solver to turbulent wind flow (named NASIR: Numerical Analyzer for Scientific and Industrial Requirements) is described. The ability of the developed software is demonstrated for simulation of three dimensional wind flow around a set of three cooling towers down stream of some large buildings in KAZERUN power station (in south part of IRAN) is presented. The results are demonstrated using color coded maps of velocity and pressure fields.

II. Governing Equation

In this paper, The Navier-Stokes equations for an incompressible fluid combined with a sub grid scale (SGS) turbulence viscosity model are used for the flow around circular cylinder. Using the concept of artificial compressibility for coupling velocity and pressure fields [6], the non-dimensional form of the governing equations in Cartesian coordinates can be written as:

$$\frac{\partial W}{\partial t}+P\left(\frac{\partial E_c}{\partial x}+\frac{\partial F_c}{\partial y}+\frac{\partial G_c}{\partial z}\right)=P\left(\frac{\partial E_d}{\partial x}+\frac{\partial F_d}{\partial y}+\frac{\partial G_d}{\partial z}\right) \quad (1)$$

Where,

$$W=\begin{pmatrix}0\\u\\v\\w\end{pmatrix},\quad P=\begin{pmatrix}\beta^2&0&0&0\\0&1&0&0\\0&0&1&0\\0&0&0&1\end{pmatrix}$$

$$F_c=\begin{pmatrix}u\\u^2+p/\rho_0\\uv\\uw\end{pmatrix},\ G_c=\begin{pmatrix}v\\uv\\v^2+p/\rho_0\\vw\end{pmatrix},\ H_c=\begin{pmatrix}w\\uw\\vw\\w^2+p/\rho_0\end{pmatrix}$$

$$F_d=\begin{pmatrix}0\\ \nu_T\frac{\partial u}{\partial x}\\ \nu_T\frac{\partial v}{\partial x}\\ \nu_T\frac{\partial w}{\partial x}\end{pmatrix},\ G_d=\begin{pmatrix}0\\ \nu_T\frac{\partial u}{\partial y}\\ \nu_T\frac{\partial v}{\partial y}\\ \nu_T\frac{\partial w}{\partial y}\end{pmatrix},\ H_d=\begin{pmatrix}0\\ \nu_T\frac{\partial u}{\partial z}\\ \nu_T\frac{\partial v}{\partial z}\\ \nu_T\frac{\partial w}{\partial z}\end{pmatrix}$$

W represents the conserved variables while, F^c, G^c and H^c are the components of convective flux vector and F^v, G^v and H^v are the components of viscous flux vector of W in non-dimensional coordinates x and y, respectively. Components of velocity u, v and w as well as pressure p, are four dependent variables. ν_T is the summation of kinetic viscosity ν and eddy viscosity defined as [1] and [2]:

$$\nu_{SGS}=(C_s\Delta)^2[1/2\,\bar{s}_{ij}\bar{s}_{ij}]^{1/2},\ i,j=1,2,3 \quad (2)$$

where $\bar{s}_{ij}=\frac{\partial\bar{u}_i}{\partial x_j}+\frac{\partial\bar{u}_j}{\partial x_i}$

The variables of above equations are converted to non-dimensional form by dividing x, y and z by L, a reference length u, v and w by U_o, upstream wind velocity, and p by ρU_0^2.

The parameter β is introduced using the analogy to the speed of sound in equation of state of compressible flow. Application of this pseudo compressible transient term converts the elliptic system of incompressible flow equations into a set of hyperbolic type equations [6]. Ideally, the value of the pseudo compressibility is to be chosen so that the speed of the introduced waves approaches that of the incompressible flow. This, however, introduces a problem of contaminating the accuracy of the numerical algorithm, as well as affecting the stability property. On the other hand, if the pseudo compressibility parameter is chosen such that these waves travel too slowly, then the variation of the pressure field accompanying these waves is very slow. Therefore, a method of controlling the speed of pressure waves is a key to the success of this approach. The theory for the method of pseudo compressibility technique is presented in the literature [7].

Some algorithms have used constant value of pseudo compressibility parameter and some workers have developed sophisticated algorithms for solving mixed incompressible and compressible problems [8]. However, the value of the parameter may be considered as a function of local velocity using following formula proposed [9]:

$$\beta^2=\ Maximum\ \ (\beta^2_{\min}\ \ or\ \ C|U^2|) \quad (3)$$

In order to prevent numerical difficulties in the region of very small velocities (ie, in the vicinity of stagnation points), the parameter $\beta^2_{\min}$ is considered in the range of 0.1 to 0.3, and optimum C is suggested between 1 and 5 [10].

The method of the pseudo compressibility can also be used to solve unsteady problems. For this propose, by considering additional transient term. Before advancing in time, the pressure must be iterated until a divergence free velocity field is obtained within a desired accuracy. The approach in solving a time-accurate problem has absorbed considerable attentions [11]. In present paper, the primary interest is to develop a method of obtaining steady-state solutions

III. Finite Volume Formulation

The governing equations can be changed to discrete form for the unstructured meshes by the application of the Vertex-base Overlapping Finite Volume Method. This method ends up with the following formulation after multi-plying the vector form of the governing equations by linear shape function of tetrahedral cells and integration by part [12, 13]:

$$W_i^{n+1} = W_i^n - \frac{\Delta t}{\Omega_i} P \sum_{k=1}^{N_{faces}} [\overline{E}_c\,(\Delta S)_x + \overline{F}_c\,(\Delta S)_y + \overline{G}_c\,(\Delta S)_z]_k^n + \frac{\Delta t}{\Omega_i} P \sum_{k=1}^{N_{faces}} [\overline{E}_d\,(\Delta S)_x + \overline{F}_d\,(\Delta S)_y + \overline{G}_d\,(\Delta S)_z]_k^n \quad (4)$$

Where, W_i represents conserved variables at the center of control volume that can be calculated as (Fig.1):

$$\Omega_i = \sum_{k=1}^{N_{faces}} \overline{x}_k (\Delta S)_x \quad (5)$$

in which, the vector of surrounding faces is defined as(Fig.2):

$$(\Delta S)_k = (\Delta S)_x \hat{i} + (\Delta S)_y \hat{j} + (\Delta S)_z \hat{k} \quad (6)$$

Hence, $\left|\overline{\Delta S}\right| = \sqrt{(y_{21}z_{31} - z_{21}y_{31})^2 + (z_{21}x_{31} - x_{21}z_{31})^2 + (x_{21}y_{31} - y_{21}x_{31})^2}$

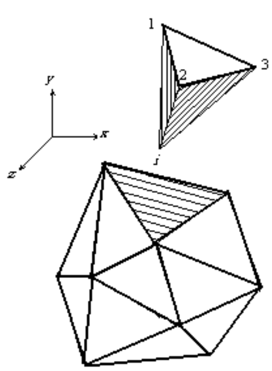

Fig. 1. Control Volume Ω_i formed by tetrahedral cells

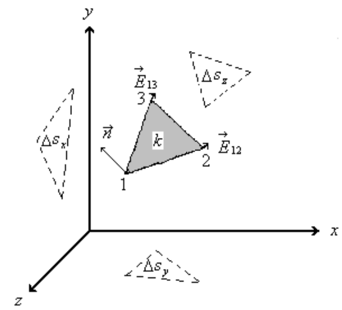

Fig. 2. Components of a face $(\Delta S)_k$ of a control Volume Ω_i

In the above formulation, F^c, G^c and H^c are the mean values of convective fluxes at the control volume boundary faces and F^v, G^v and H^v are the mean values of viscous fluxes which are computed at each tetrahedral cell.

Formulation 4 can be written in the following form:

$$\frac{\Delta W_i}{(\Delta t)_i} = \frac{1}{\Omega_i}.P_i.\left[D(W_i) - C(W_i)\right]_k^n \quad (7)$$

Where, $C(W)_i$ and $D(W)_i$ are convective and diffusive operators at node i, respectively.

The convective is defined as:

$$C(W)_i = \sum_{k=1}^{N_{faces}} [\overline{E}_c (\Delta S)_x + \overline{F}_c (\Delta S)_y + \overline{G}_c (\Delta S)_z]_k^n \quad (8)$$

That can be written as,

$$C(W)_i = \sum_{k=1}^{N} \begin{pmatrix} (\Delta S)_x(\overline{u}) + (\Delta S)_y(\overline{v}) + (\Delta S)_z(\overline{w}) \\ (\Delta S)_x(\overline{uu} + \overline{p}) + (\Delta S)_y(\overline{uv}) + (\Delta S)_z(\overline{uw}) \\ (\Delta S)_x(\overline{vu}) + (\Delta S)_y(\overline{vv} + \overline{p}) + (\Delta S)_z(\overline{vw}) \\ (\Delta S)_x(\overline{wu}) + (\Delta S)_y(\overline{wv}) + (\Delta S)_z(\overline{ww} + \overline{p}) \end{pmatrix}$$

The diffusive (viscous + turbulent) operator is defined as,

$$D(W)_i = \sum_{k=1}^{N_{faces}} [\overline{E}_d (\Delta S)_x + \overline{F}_d (\Delta S)_y + \overline{G}_d (\Delta S)_z]_k^n \quad (9)$$

That can be written as,

$$D(W)_i = 3/2 \sum_{k=1}^{N} \begin{pmatrix} 0 \\ \{(\Delta S)_x \sum_{j=1}^{M} \left[\overline{u}(\Delta S)_x\right]_j / \Lambda + (\Delta S)_y \sum_{j=1}^{M} \left[\overline{u}(\Delta S)_y\right]_j / \Lambda + (\Delta S)_z \sum_{j=1}^{M} \left[\overline{u}(\Delta S)_z\right]_j / \Lambda\} \nu_{SGS} \\ \{(\Delta S)_x \sum_{j=1}^{M} \left[\overline{v}(\Delta S)_x\right]_j / \Lambda + (\Delta S)_y \sum_{j=1}^{M} \left[\overline{v}(\Delta S)_y\right]_j / \Lambda + (\Delta S)_z (\sum_{j=1}^{M} \left[\overline{v}(\Delta S)_z\right]_j / \Lambda\} \nu_{SGS} \\ \{(\Delta S)_x \sum_{j=1}^{M} \left[\overline{w}(\Delta S)_x\right]_j / \Lambda + (\Delta S)_y \sum_{j=1}^{M} \left[\overline{w}(\Delta S)_y\right]_j / \Lambda + (\Delta S)_z \sum_{j=1}^{M} \left[\overline{w}(\Delta S)_z\right]_j / \Lambda\} \nu_{SGS} \end{pmatrix}$$

In this study, assuming negligible viscosity for the air, the Sub-Grid Scale model is used for computation of the turbulence eddy viscosity $\nu_{SGS} = (C_s\Delta)^2[1/2\,\bar{s}_{ij}\bar{s}_{ij}]^{1/2}$, in which:

$$\bar{S}_{ij} \approx \frac{1}{\Lambda}\sum_{1}^{4}[\bar{u}_i(\Delta S)_i + \bar{u}_j(\Delta S)_j] \qquad (10)$$

Where, are for the two-dimensional computation in this paper. The Sub-Grid Scale model is used for definition of ν_{SGS}, where Δ is the area of a triangular cell and the $C_s=0.15$ are used. In order to damp unwanted numerical oscillations associated with the explicit solution of the above algebraic equation a fourth order (Bi-Harmonic) artificial dissipation term $A_4(W_i)$ is added to the convective, $C(W_i)$ and viscous, $D(W_i)$ terms. Hence, the general formulation should be modified as;

$$\frac{\Delta W_i}{(\Delta t)_i} + \frac{1}{\Omega_i}.P_i.\left[C(W_i) + A(W_i) - D(W_i)\right]_k^n = 0 \qquad (11)$$

The artificial dissipation term, is formed by using the Bi-Harmonic operator as follow [3];

$$A_4(W_i) = \varepsilon_4 \sum_{j=1}^{N_{faces}} \lambda_{ij}(\nabla^2 W_j - \nabla^2 W_i) \qquad (12)$$

Here, $\lambda_{ij} = \min(\lambda_i, \lambda_j)$, is a scaling factors. This formulation is adopted using the local maximum value of the spectral radii Jacobian matrix of the governing equations and the size of the mesh spacing as:

$$\lambda_i = \sum_{k=1}^{Ne}\left|U_k.(\Delta S)_k\right| + \sqrt{\left(\left|U_k.(\Delta S)_k\right|\right)^2{}_k + (\Delta S)_k^2} \qquad (13)$$

In above equation, the Laplacian operator at every node *i*, is computed using all the variables *W* at neighboring nodes *j*.

$$\nabla^2 W_i = \sum_{j=1}^{Nedge}(W_j - W_i) \qquad (14)$$

Similar to the most numerical formulations, this formulation is somehow mesh-dependent. For obtaining the accurate results, the minimization of the coefficient ε is the key point in the application of the artificial dissipation term on the specific mesh ($1/256 \le \varepsilon \le 3/256$) [3].

In the above decribed formulations, superscripts *n* and *n+1* show *n*th and the *n+1*th time stages. *Δt* is the time step (proportional to the minimum mesh spacing) applied between time stages *n* and *n+1*. In present study, a three-stage Runge-Kutta scheme is used for stabilizing the computational process by damping high frequency errors, which this in turn, relaxes CFL condition [14].

Due to the variations in sizes unstructured control volumes calculations, the allowable time step for computation of dynamic problems for the entire mesh is limited to the minimum associated with the smallest control volume of the domain. However, the large variation in grid size for the unstructured mesh will slow down the computations. In present work, the local time step of each control volume is used for computation of static problems. In this technique to accelerate the convergence to steady state conditions, the computation of each control volume can advance using a pseudo time step which is calculated for its own control volume. The use of local time stepping greatly enhances the convergence rate [15].

In the formulation of $C(W)_i$ $D(W)_i$ and $A_4(W_i)$, the faces are referred to during the computation procedure. Therefore, it would be convenient to use the face-base data structure for definition of unstructured meshes. In this case, using the face-base computational algorithm reduces the number of addressing to the memory, and therefore, provides 50% saving in computational CPU time [3].

Two types of boundary conditions are applied in this work; flow and solid wall boundary conditions. The flow boundary condition is developed from the similarity to the one dimensional characteristic theory for the first order wave equations. From this theory, the prorogation directions are defined according to the sign of the system waves of the incompressible convection dominated equations (Eigenvalues of the Jacobian matrix of the continuity and motion equations). The values of related quantities are imposed wherever the characteristics enter the computational domain. Conversely, at boundaries where characteristics leave the domain, nodal value of related quantities are determined from the interior solution domain. For incompressible flow, at the inflow boundaries free stream, values of *u*, *v* and *w* are imposed and *p* is extrapolated from inside domain nodes, and at the outflow boundaries free stream, *p* is imposed and *u*, *v* and *w* are extrapolated from inside domain nodes [14].

IV. Verification Case

In order to assess the changes of pressure distribution on the circular cylinder with standard geometrical feature, the flow solver is applied to solve the turbulent flow on an unstructured tetrahedral mesh (Fig.3).

In this work, No-slipping condition is considered at the solid wall nodes by setting zero normal and tangential components of computed velocities at wall nodes. At inflow boundaries unit free stream velocity and at outflow boundaries unit pressure is imposed. The free stream flow parameters (outflow pressure and inflow velocity) are set at every computational node as initial conditions.

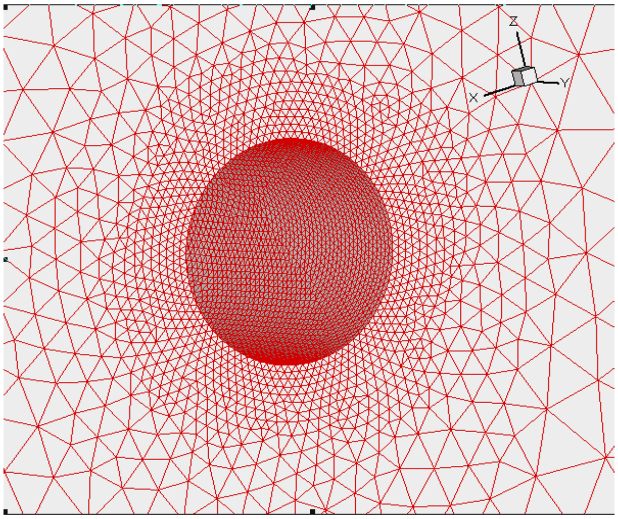

Fig. 3, Computational domain of the verification problem

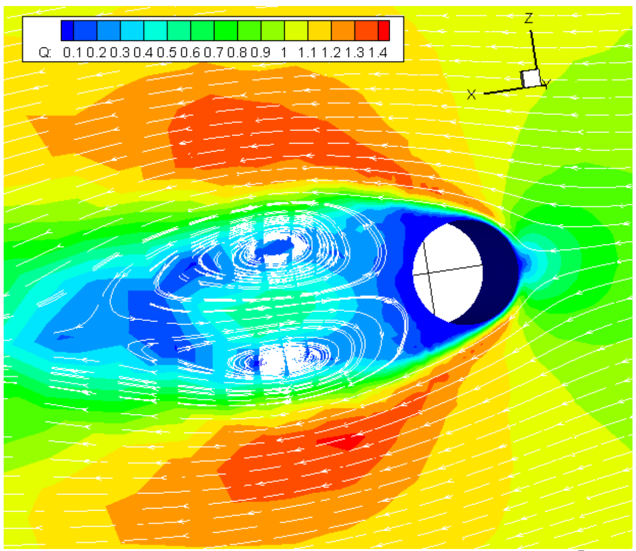

Fig.5, Computed velocity contours for Re=4.5×10^5

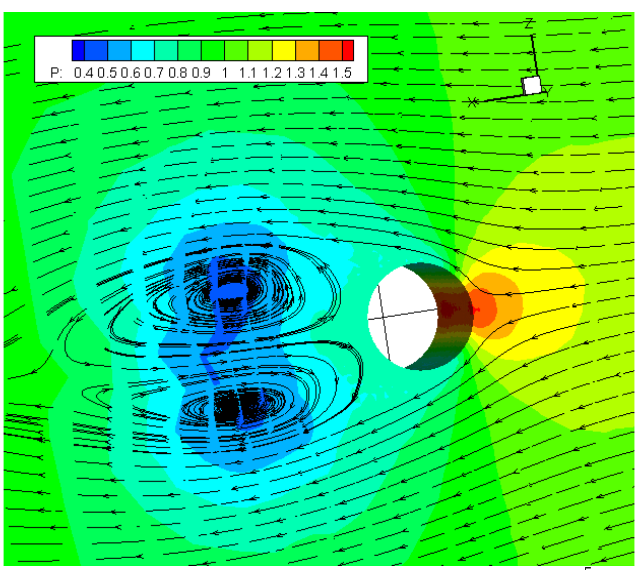

Fig.6, Computed pressure contours for Re=4.5×10^5

The computed results of supercritical Reynolds number (Re=4.5×10^5) flow around the cylinder wall are plotted in terms of velocity and pressure fields in (Fig. 5) and (Fig. 6) , respectively.

Accuracy of the developed turbulent three dimensional flow solver on unstructured meshes is examined by with the two dimensional results of similar work using SGS turbulent eddy viscosity model [5] (Fig7). The results of the two-dimensional model are verified by comparison with experimental measurements in Peking University [16]. As can be seen, similar to the two dimensional solution (Fig.7), computed stream lines present two symmetric vortexes behind the three dimensional cylinder (Fig.5). Therefore, it can be stated that, despite of fully unstructured nature of the utilized mesh the results of the developed three dimensional flow solver can be symmetric if proper techniques are utilized.

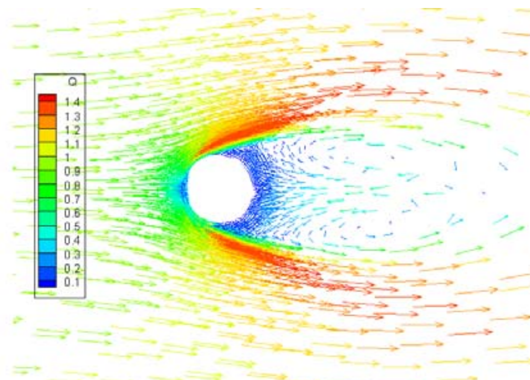

Fig.7, Velocity vectors at Re=4.5×10^5 computed by two-dimensional verified flow sore

V. Application of the Model

The performance of the developed flow-solver is examined by solving turbulent wind flow around a set of three cooling towers with irregular locations (Fig.8).

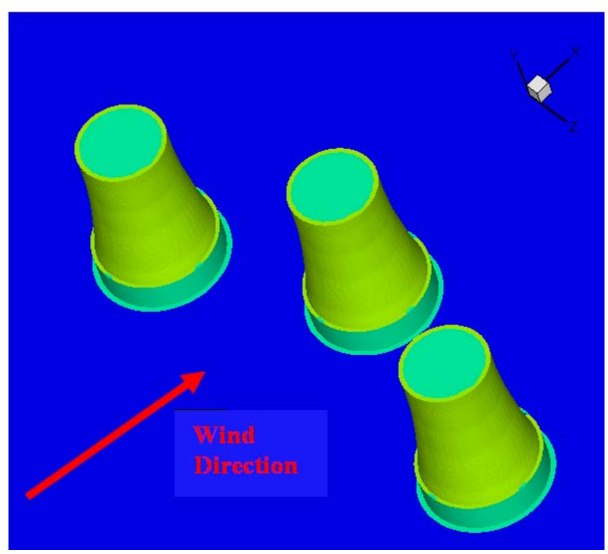

Fig. 8, A group of cooling towers with arbitrary arrangement

In order to assess the effects of cooling towers-building interference the flow solver is applied to solve the wind induced flow patterns on two different meshes which geometrically represent; first single cooling tower and second three neighboring cooling towers with large building at upstream. To generate unstructured mesh of tetrahedral around the objects in the wind flow field, geometry of the far-field flow boundaries as well as surfaces of ground and structures are digitally modeled in the first stage. Then, inside the surface boundaries is filled with unstructured tetrahedral cell.

In this work, it is assumed that the louvers in cooling tower(s) base wall are closed. Therefore, the air flow through top opening of the tower(s) is considered negligible and impermeable free slip condition is imposed at the top plane of the tower(s). Free slip condition is considered at the solid wall nodes by setting zero normal components of computed velocities at wall nodes. At inflow boundaries unit free stream velocity and at outflow boundaries unit pressure is imposed. The free stream flow parameters (outflow pressure and inflow velocity) are set at every computational node as initial conditions.

A. *Turbulent Wind Flow around Single Cooling Tower*

The solid boundary surfaces of the unstructured tetrahedral mesh for the single cooling tower are colored with the vertical elevation surfaces (Fig.9).

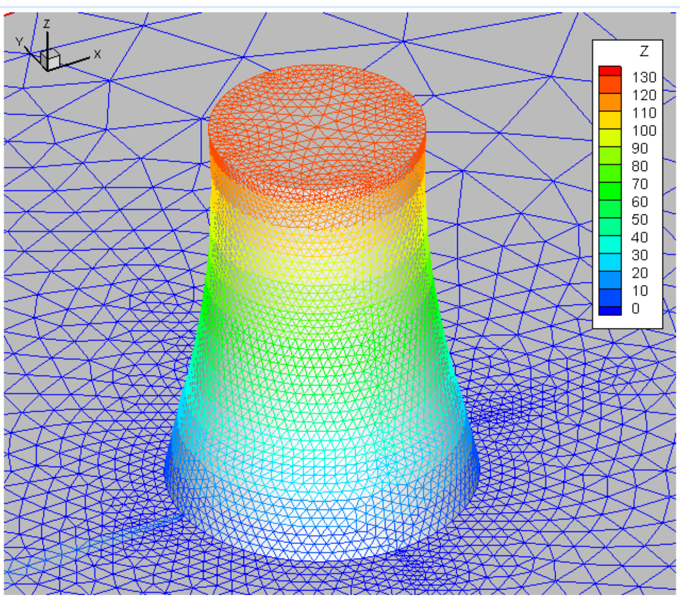

Fig.9, Unstructured tetrahedral mesh for single cooling tower

The upstream and downstream views of computed dimensionless pressure contours are plotted on the impermeable boundary surfaces (ground surface, solid wall cooling tower shell and top inviscid surface) and presented in the following figures (Fig.10).

The results on the single cooling tower are plotted in terms of stream lines and contours color coded by computed dimensionless velocity and pressure fields, respectively. The flow variables are converted to non-dimensional form by dividing x and y, z by L, a reference length u, v and w by U_o, upstream wind velocity, and p by $\rho_o U_o^2$.

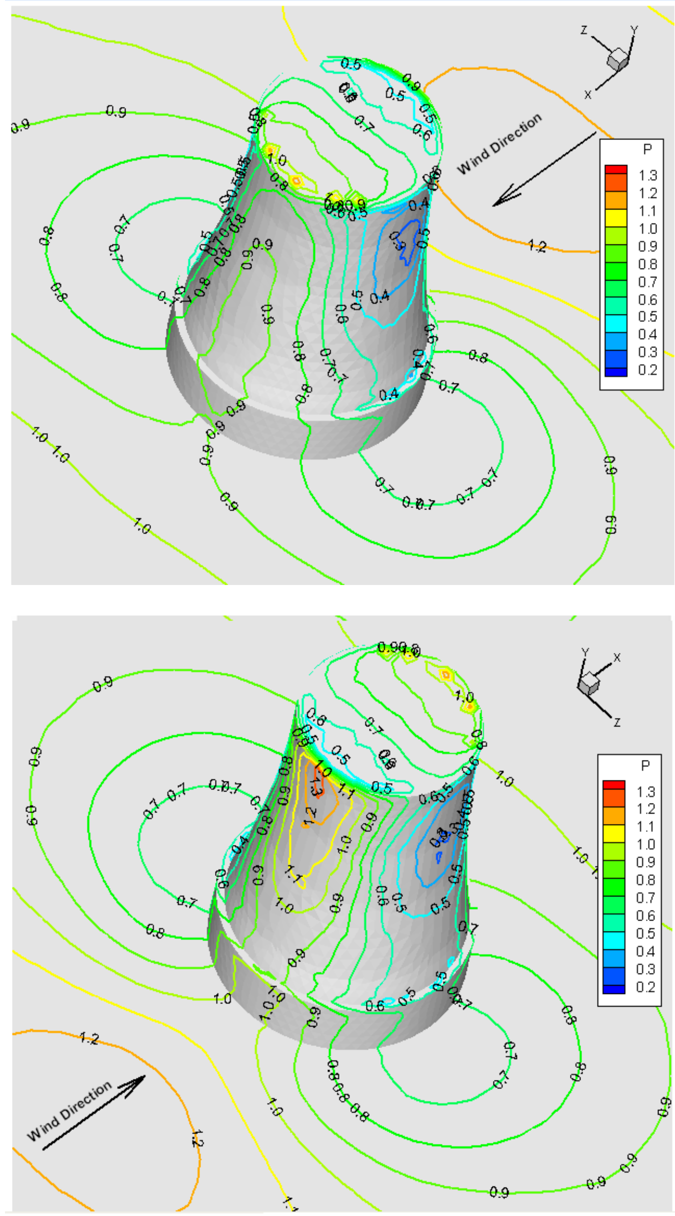

Fig.10, Upstream and downstream views of computed dimensionless pressure contours for single cooling tower

In order to give an inside view to the velocity field around the tower walls, the color coded stream lines on some horizontal planes with different elevations are plotted in the following (Fig.11).

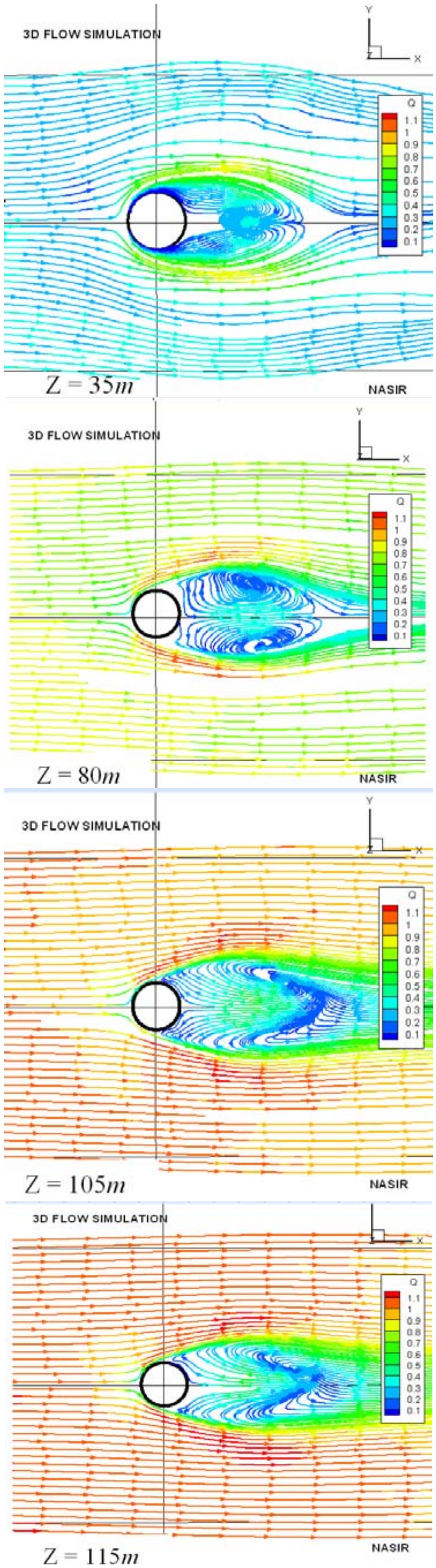

Fig.11, Upstream and downstream views of computed dimensionless pressure contours for single cooling tower

B. *Turbulent Wind Flow around Three Cooling Towers*

For the case of three cooling towers with irregular locations down stream of some large buildings the triangular mesh of boundary surfaces of volume unstructured tetrahedral mesh are colored with the vertical elevation surfaces (Fig.12)

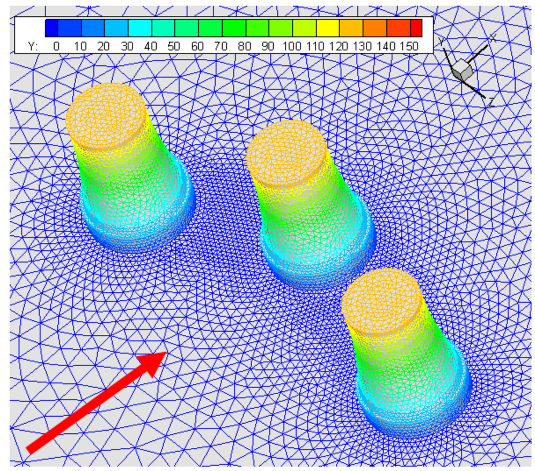

Fig.12, Unstructured tetrahedral mesh for a group of three cooling towers

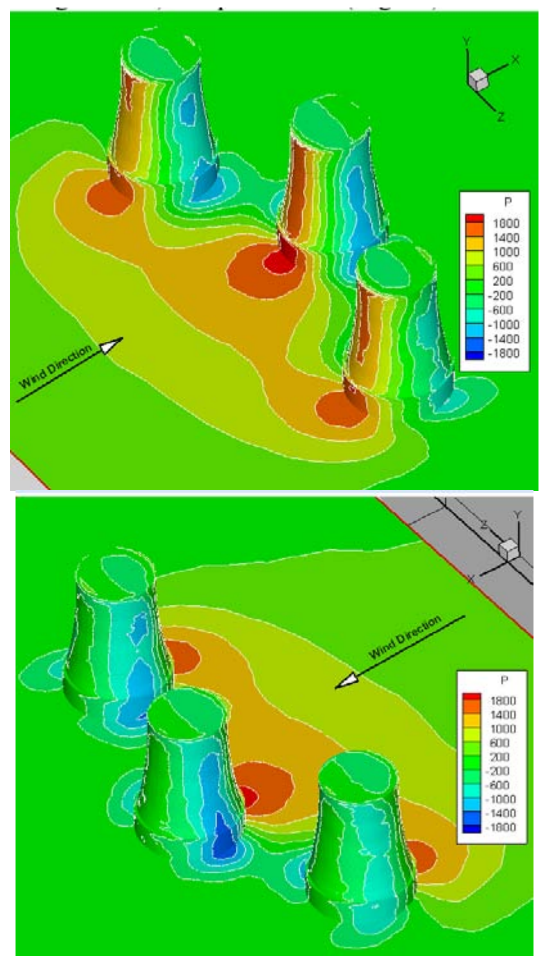

Fig.13, Upstream and downstream views of computed dimensionless pressure contours for three cooling towers

The developed flow solver is applied for computation of pressure field around cooling towers due to upstream velocity of 27.5 m/s at 10 m above the ground level and downstream zero pressure.

In the following figures, the upstream and downstream views of color coded maps of computed pressure are plotted on the impermeable boundary surfaces (ground surface, solid wall surfaces of cooling towers shell and top inviscid surface of the cooling towers) are presented (Fig.11).

The color coded maps of computed pressure on two vertical perpendicular planes are presented for three cooling towers in the following figures, (Fig.14), (Fig.15) and (Fig.16). As can be seen, due to interference effects, there are some difference in pressure distributions around the cooling towers.

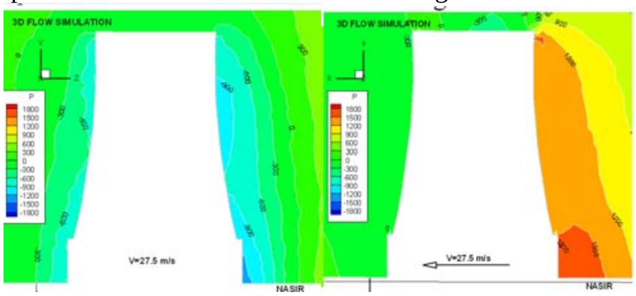

Fig.14, Computed pressure on two vertical perpendicular planes passing through the centre of the right cooling tower

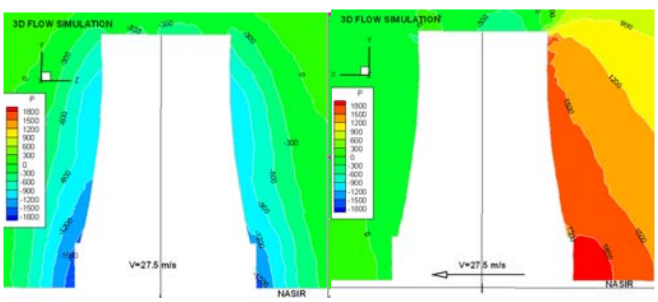

Fig.15, Computed pressure on two vertical perpendicular planes passing through the centre of the central cooling tower

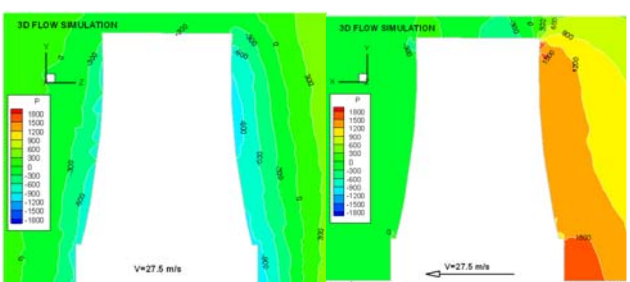

Fig.16, Computed pressure on two vertical perpendicular planes passing through the centre of the left cooling tower

As can be seen due to neighboring effects the flow parameters on three towers with upstream buildings not only are not symmetric and the pressure field is very different from the case of a single cooling tower, but also it is different on each of the three towers.

VI. Conclusion

Using the above described explicit vertex-base overlapping Finite Volume formulation, there is no need to use the reconstruction method, and hence, it is computationally superior to the Cell Centre Finite Volume Methods [17]. Furthermore, unlike the Vertex-base overlapping Finite Element Methods, the explicit nature of the formulations paves the way for matrix free computations procedure.

This wall function free software is successfully used for investigation of SGS turbulent eddy viscosity model on computation of wind pressure around a circular cylinder at supercritical Reynolds number (Re=4.5×105). Then, the developed turbulent solver model is applied to the cases with complicated geometries. Turbulent flow around a set of irregularly arranged cooling tower downstream of some tall buildings is chosen as a challenging application case. From the computed results, it can be stated that complicated physical conditions around a geometrically complex object can accurately modeled using the presented flow solver.

The computed results of the developed model show that, there are major differences in computed velocity patterns and pressure fields on the wall surface of the cooling tower due to neighboring of a number of cooling towers will result considerable changes in computed flow parameters. The computational results show that the interference effects due to neighboring of the cooling towers not only changes the flow patterns from the symmetric condition to irregular condition but also considerable differences in the minimum and maximum values of the flow parameters on three towers may appear.

The computed results prove that the computer simulation can serve as a powerful means for investigation of the complex engineer problems.

VII. Conclusion

Thanks to BOLANDPAYEH Engineering Co. for supporting this research work.

References

[1] S. Murakami and A. Mochida, *On turbulent vortex shedding flow past 2D square cylinder predicted by CFD*, Journal of Wind Engineering Industrial Aerodynamics. 54/55, 1995, pp 191-211.

[2] Salvatici E. and Salvetti M.V. *large eddy simulations of the flow around a circular cylinder: effects of grid resolution and sub-grid scale modeling*, Journal of Wind and Structures, Vol. 6, No. 6, 2003, pp 419-436.

[3] S.R. Sabbagh-Yazdi and A. Hadian, *Accuracy Assessment of Solving Pseudo Compressible Euler Equations on Unstructured Finite Volumes* ANZIAMJ Vol. 46, No. C, 2004, Available at: http:// anziamj.austms.org.au/V46/CTAC2004/yazd

[4] S.R. Sabbagh-Yazdi, M. Torbati, F. Meysami-Azad, and B. Haghighi, Computer Simulation of Changes in the Wind Pressure Due to Cooling Towers-Buildings Interference, *WSEAS Transactions on Mathematics*, Vol6, No1, 2007, pp. 205-214.

[5] S.R. Sabbagh-Yazdi, N.E. Mastorakis and F. Meysami, Wind flow pressure load simulation around storage tanks using SGS turbulent model, *International Journal of Mechanics*, Issue 3, Vol.1, 2007, pp. 39-44.

[6] A.,Chorin, *A Numerical Method for Solving Incompressible Viscous Flow Problems"*, *Journal of Computational Physics*, Vol. 2, 1967, pp 12-26.

[7] J.L. Chang and D. Kwak, *On the Method of Pseudo Compressibility for Numerically Solving Incompressible Flow*, AIAA 84-0252, 22nd Aerospace Science Meeting and Exhibition, Reno, 1984.

[8] Turkel E., *Preconditioning Methods for Solving the Incompressible and Low Speed Compressible Equations*, ICASE Report 86-14, 1986.

[9] Dreyer. J., *Finite Volume Solution to the Steady Incompressible Euler Equation on Unstructured Triangular Meshes*, M.Sc. Thesis, MAE Dept., Princeton University, 1990.

[10] A.,Rizzi, and L.,Eriksson, *Computation of Inviscid Incompressible Flow with Rotation*, Journal of Fluid mechanic, Vol. 153, 1985, pp 275-312.

[11] A.,Belov, L.,Martinelli and A.,Jameson, *A New Implicit Algorithm with Multi-grid for Unsteady Incompressible Flow Calculations*, AIAA 95-0049, 33rd Aerospace Science Meeting and Exhibition, Reno, 1995.

[12] L.A. Sykes, *Development of a Two-Dimensional Navier-stokes Algorithm for Unstructured Triangular Grids*, ARA Report 80, April 1990.

[13] R.S. Sabbagh-Yazdi, *Simulation of the Incompressible Flow Using the Artificicial Compressibility Method*, Ph.D Thesis , University of Wales, Swansea , 1997

[14] Jameson A., Schmidt W. and Turkle E. , *Numerical Solution of the Euler Equations by Finite Volume Method using Runge Kutta Time Stepping Schemes* , AIAA Paper 81-1259, June 1981.

[15] Jameson A., Baker T. J. and Weatherill N. P. , *Calculation of Inviscid Transonic Flow over a Complete Aircraft*, AIAA Paper 86-0103, Jan. 1986.

[16] Zh. Gu, *On interference between two circular cylinders at supercritical Reynolds number*, Journal of Wind Engineering and Industrial Aerodynamics 62, 1996, pp. 175-190.

[17] M. Aftosmis, D. Gaitonde, and T. Sean-Tavares *On the Accuracy, Stability, and Monotonicity of Various Reconstruction Algorithms for Unstructured Meshes*, AIAA 32nd Aerospa.ce Sciences Meeting and Exhibit, Reno, NV, US, AIAA-94-0415, 1994.

First Author's biography may be found in following site:
http://sahand.kntu.ac.ir/~syazdi/