Wind flow pressure load simulation around storage tanks using SGS turbulent model

S.R. Sabbagh-Yazdi, N.E. Mastorakis, and F. Meysami

Abstract**—**The equation of continuity is simultaneously solved with the two equations of motion in a coupled manner by application of the pseudo compressibility technique for the steady state problems. The set of two dimensional for the incompressible fluid is combined with a SGS (Sub-Grid Scale) eddy viscosity turbulence model. The discrete form of the two-dimensional flow equations are formulated using the Galerkin Finite Volume Method for unstructured mesh of triangular cells. Using unstructured meshes provides the merit of accurate geometrical modeling of the curved boundaries of the tanks. Satisfactory results are obtained by the use of proper boundary conditions. The accuracy of the model for the solution flow around circular cylinder at supercritical Reynolds number is assessed by comparison of computed results with experimental coefficient of pressure measurements. Then, the model is applied to simulate the changes in the pressure distributions due to the wind flow on two storage tanks in tandem arrangement.

Keywords**—**Wind Pressure Load, SGS Turbulent Viscosity Model, Storage Tanks Interference, Triangle Unstructured Mesh, 2D Galerkin Finite Volume Method

I. INTRODUCTION

The availability of high performance digital computers and development of efficient numerical models techniques have accelerated the use of Computational Fluid Dynamics. The control over properties and behavior of fluid flow and relative parameters are the advantages offered by CFD which make it suitable for the simulation of the applied problems. Consequently, the computer simulation of complicated flow cases has become one of the challenging areas of the research works.

The interaction of neighboring tanks may considerably change the pressure filed on a storage tank. More over, wind flow through particular arrangement of storage tanks may produce unexpected pressure fields, which may cause disastrous structural loading condition. Therefore, modeling of final design is recommended by most of the codes of practice to evaluate actual pressure loads on them.

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].

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 two-dimensional flow equations are formulated using the Galerkin Finite Volume for unstructured mesh of triangles. Using unstructured meshes provides great flexibility for modeling the flow in geometrically complex domains.

The ability of the developed Galerkin finite volume solver, is applies to simulate wind flow at supercritical Reynolds number ($Re=4.5\times10^5$) on the pressure distribution on circular cylinder is presented and discussed. Then, as an application of the developed model, computation of pressure distribution of two circular tanks with different diameter in tandem arrangement at supercritical Reynolds number (1.43×10^8) is performed and the results are discussed.

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. The non-dimensional form of the governing equations in Cartesian coordinates can be written as:

Manuscript received May 29, 2007: Revised version received July 29, 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).

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

Farzad Meysami is MSc. Graduate of Civil Engineering Department of K.N. Toosi University of Technology, 1346 Valiasr St. Tehran, IRAN

$$\frac{\partial W}{\partial t}+(\frac{\partial F^{c}}{\partial x}+\frac{\partial G^{c}}{\partial y})+(\frac{\partial F^{v}}{\partial x}+\frac{\partial G^{v}}{\partial y})=0 \qquad (1)$$

Where,

$$W=\begin{pmatrix}\dfrac{p/\rho_0}{\beta^2}\\ u\\ v\end{pmatrix},$$

$$F^{c}=\begin{pmatrix}u\\ u^2+p/\rho_0\\ uv\end{pmatrix},\quad G^{c}=\begin{pmatrix}v\\ uv\\ v^2+p/\rho_0\end{pmatrix},$$

$$F^{v}=\begin{pmatrix}0\\ \nu_T\dfrac{\partial u}{\partial x}\\ \nu_T\dfrac{\partial v}{\partial x}\end{pmatrix},\quad G^{v}=\begin{pmatrix}0\\ \nu_T\dfrac{\partial u}{\partial y}\\ \nu_T\dfrac{\partial v}{\partial y}\end{pmatrix}$$

W represents the conserved variables while, F^c, G^c are the components of convective flux vector and F^v, G^v are the components of viscous flux vector of W in non-dimensional coordinates x and y, respectively. Components of velocity *u*, *v* and pressure *p*, are three dependent variables. ν_T is the summation of kinematic viscosity ν and eddy viscosity ν_t.

The variables of above equations are converted to non-dimensional form by dividing *x* and *y* by L, a reference length u and v 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 [3]. 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 [4].

Some algorithms have used constant value of pseudo compressibility parameter and some workers have developed sophisticated algorithms for solving mixed incompressible and compressible problems [5]. However, the value of the parameter may be considered as a function of local velocity using following formula proposed [6]

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

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 [7].

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 [8]. 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 Galerkin Finite Volume Method. This method ends up with the following 2D formulation after multi-plying the vector form of the governing equations by linear shape function of triangular elements and integration by part [9, 10]:

$$\frac{\Delta W}{\Delta t}=\frac{(W_j^{n+1}-W_j^{n})}{\Delta t}=-\frac{P}{\Omega}\left[\sum_{k=1}^{Nedge}(F^{c}\Delta y-G^{c}\Delta x)\right]$$
$$-\frac{P}{A}\left[\frac{3}{2}\sum_{i=1}^{Ncell}(F^{v}\Delta y-G^{v}\Delta x)\right] \qquad (2)$$

Where, W_i represents conserved variables at the center of control volume Ω_i.

Here, F^c, G^c are the mean values of convective fluxes at the control volume boundary faces and F^v, G^v are the mean values of viscous fluxes which are computed at each triangle. Superscripts *n* and *n+1* show n^{th} and the $n+1^{\text{th}}$ computational steps. Δt is the computational 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.

In this study, the Sub-Grid Scale (SGS) model is used for computation of the turbulence viscosity, as follow [11]:

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

$$\approx \frac{1}{\Delta}\sum_{1}^{3}[\bar{v}\Delta y - \bar{u}\Delta x] \quad \bar{s}_{ij} = \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \tag{4}$$

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 equation 4, $\bar{u}, \bar{v}$ are mean values of velocity in each edge of the triangular element. Δx, Δy for edge k of control volume Ω are computed as follow:

$$\Delta y_k = y_{n_2} - y_{n_1}, \quad \Delta x_k = x_{n_2} - x_{n_1} \tag{5}$$

In order to damp unwanted numerical oscillations associated with the explicit solution of the above algebraic equation a fourth order (Bi-Harmonic) numerical dissipation term is added to the convective, $C(W_i)$ and viscous, $D(W_i)$ terms. Where;

$$C(W_i) = \sum_{k=1}^{N_{edges}}[F^c\Delta y - G^c\Delta x] \tag{6}$$

and

$$D(W_i) = \sum_{k=1}^{Ncell}[F^v\Delta y - G^v\Delta x] \tag{7}$$

The numerical dissipation term, is formed by using the Laplacian operator as follow;

$$\nabla^4 W_i = \varepsilon_4 \sum_{j=1}^{Ne} \lambda_{ij}(\nabla^2 W_j - \nabla^2 W_i) \tag{8}$$

The Laplacian operator at every node i, is computed using the variables W at two end nodes of all N_{edge} edges (meeting node i)

$$\nabla^2 W_i = \sum_{j=1}^{Ne}(W_j - W_i) \tag{9}$$

In equation 8, λ_i, the scaling factors of the edges associated with the end nodes i of the edge k. 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 [12]:

$$\lambda_i = \sum_{k=1}^{Ne}\left|(u.\Delta y - v.\Delta x)_k\right| + \sum_{k=1}^{Ne}\sqrt{(u.\Delta y - v.\Delta x)^2_k + (\Delta x^2 + \Delta y^2)_k} \tag{10}$$

Using the above described (Galerkin Finite Volume) formulations, similar to the Cell-Vertex Finite Volume Methods, the flow variables are explicitly computed at the nodal points. Therefore, there is no need to use the reconstruction method, and hence, it is computationally superior to the Cell Centre Finite Volume Methods [13]. Furthermore, unlike the Galerkin Finite Element Methods, the explicit nature of the formulations paves the way for matrix free computations procedure [14].

IV. Model Verification

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 a mesh of unstructured triangles (Fig.1).

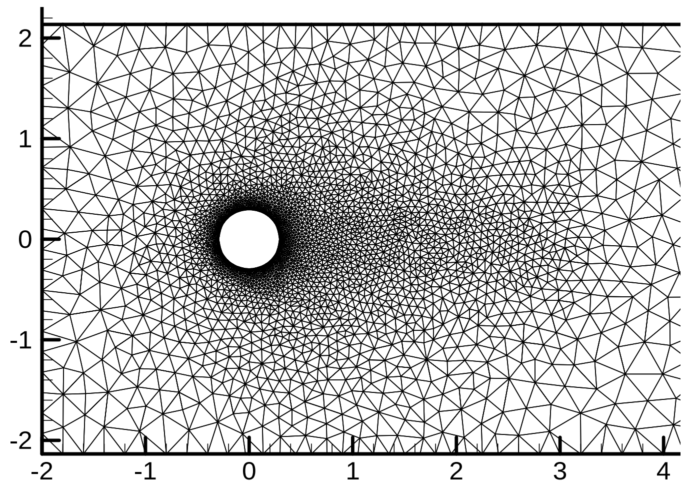

Fig. 1, Computational domain of the problem

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.

Accuracy of the developed turbulent flow solver is examined by solving case with experimental solutions which is done in Peking University. The tunnel has an open circular test section of 2.25 *m* in diameter and 3.65 *m* long. Maximum speed was 50 *m/s* [15].

The results on the cylinder wall at supercritical Reynolds number ($Re=4.5\times10^5$) are plotted in terms of velocity vectors in (Fig.2). Distribution of the coefficient of pressure on cylinder wall are compared with the experimental measurements] in (Fig. 3) and (Fig. 4), for the computations without and with SGS turbulent eddy viscosity model, respectively. Table 1 shows the percentage of changes in pressure coefficient due to application of SGS turbulent eddy viscosity model.

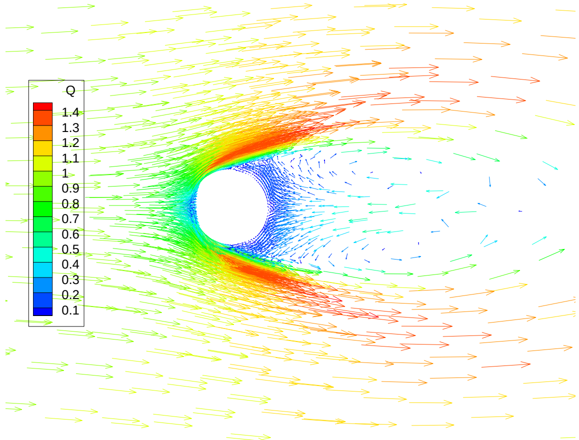

Fig.2, Computed velocity vectors at Re=4.5×10^{5}

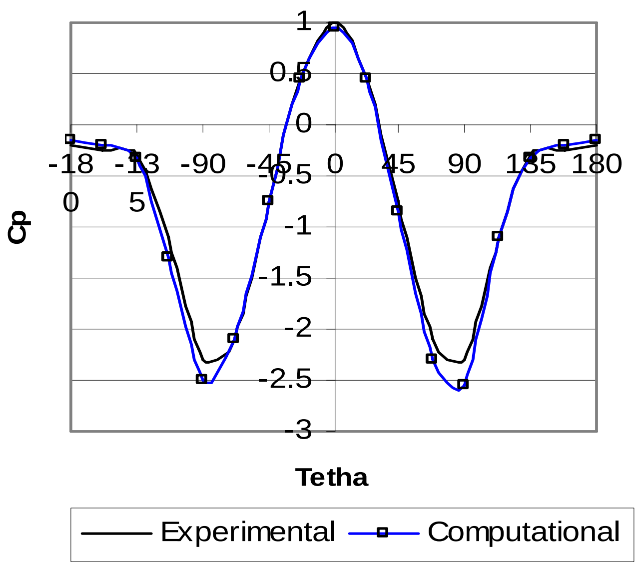

Fig.3, Coefficient of pressure on cylinder walls, (Numerical results without turbulent viscosity)

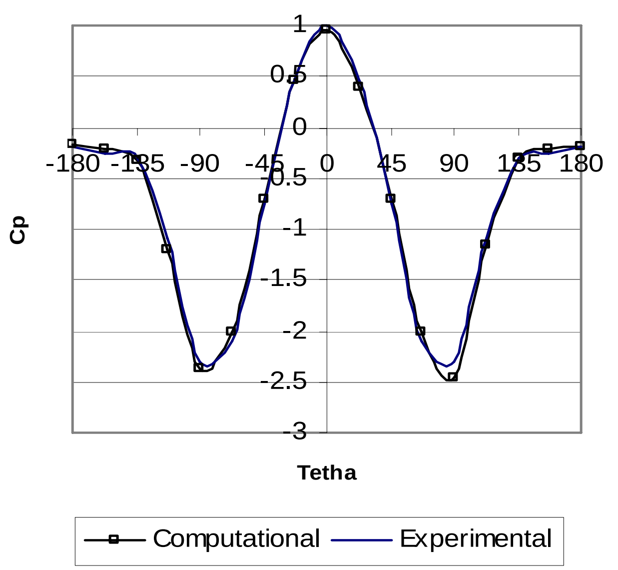

Fig. 4, Coefficient of pressure on cylinder walls, (Numerical results with SGS turbulent viscosity)

Table 1, the maximum error of the $C_{p_i} = \dfrac{2(P_0 - P_i)}{\rho.\left|U^2\right|}$

Error	Average error	Maximum error
Without turbulent viscosity	7%	12%
With SGS turbulent viscosity	4%	10.5%

V. Application of the Model

According to API 650 storage tanks should remain stable during hurricanes with the speed of 100*m/h* (44*m/s*) [16]. For air we have $\rho = 1.23 kg/m^3$, $\mu = 1.795\times10^{-5}$,So Reynolds number for tank A is computed as;

$$\mathrm{Re} = \frac{\rho v D}{\mu} = \frac{1.23\times44\times47.5}{1.795\times10^{-5}} 1.43\times10^{8}$$

The performance of the this flow-solver is examined by solving turbulence flow around two storage tanks in the KAZERUN power station located in south part of IRAN. In figure 5, schematic diagram shows the arrangement of two tanks in KAZERUN site.

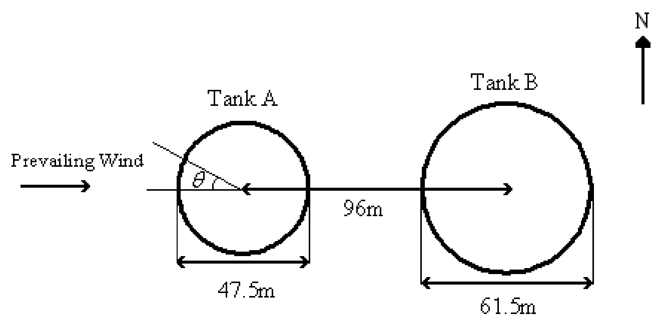

Fig. 5. Schematic diagram of the arrangement of two tanks in KAZERUN power plant

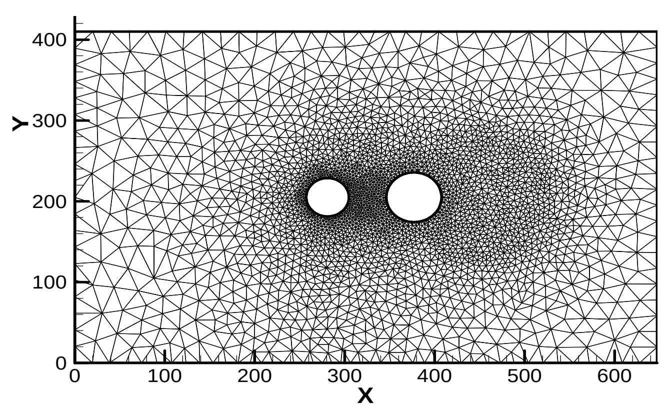

Fig. 6. Mesh for two tanks in tandem arrangement

In order to assess the changes of pressure distribution when two tanks are in tandem arrangement the flow solver is applied to solve the turbulence flow on three different meshes; first tank A, second tank B and finally two tanks with each other (Fig.6).

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.

The results on the tank A at supercritical Reynolds number (1.43×10^{8}) is plotted in terms of velocity vectors respectively (Fig.7).

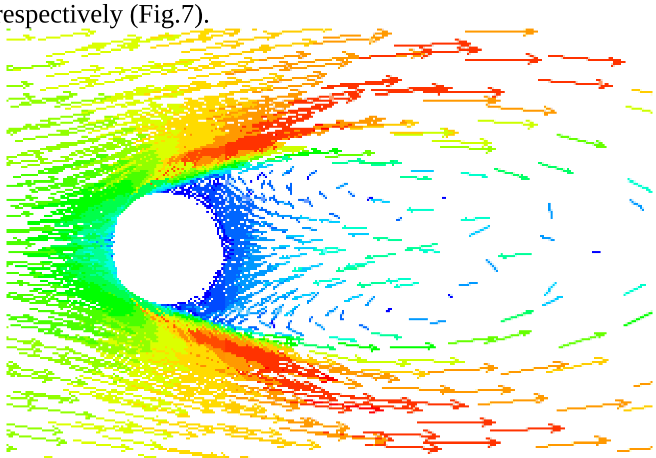

Fig. 7. Velocity vectors around a single tank (A) at supercritical Reynolds number (4.52×10^5)

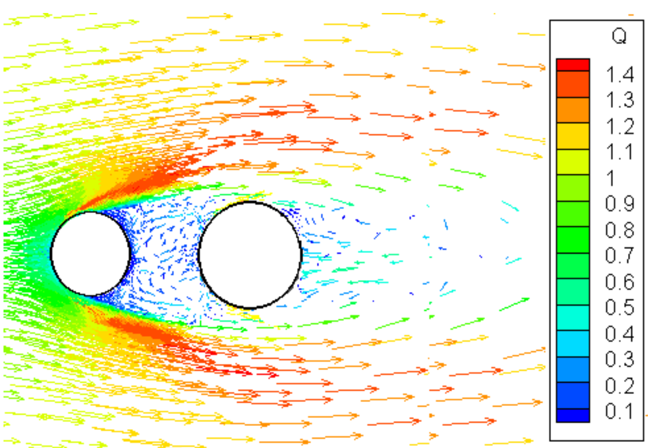

Fig. 8. Velocity vectors around two tandem tanks (A and B) at supercritical Reynolds number (4.52×10^5)

For the case of two storage tanks with tandem arrangement velocity vectors are presented in figure 8. In this case, two tanks behave as a single body because the spacing ratio between two tanks is small. As can be seen, there is almost no fluid flow in the gap of between two tanks.

Pressure distributions around tank A for the conditions of with and without neighboring tank is plotted in figure 9. Minimum pressure reaches the value of (-2.5) and then raises to (+1). Pressure distribution changes around tank A, where two tanks are in tandem. It can be clearly seen that the pressure distribution on tank A is reduced by considering the neighborhood of the downstream thank.

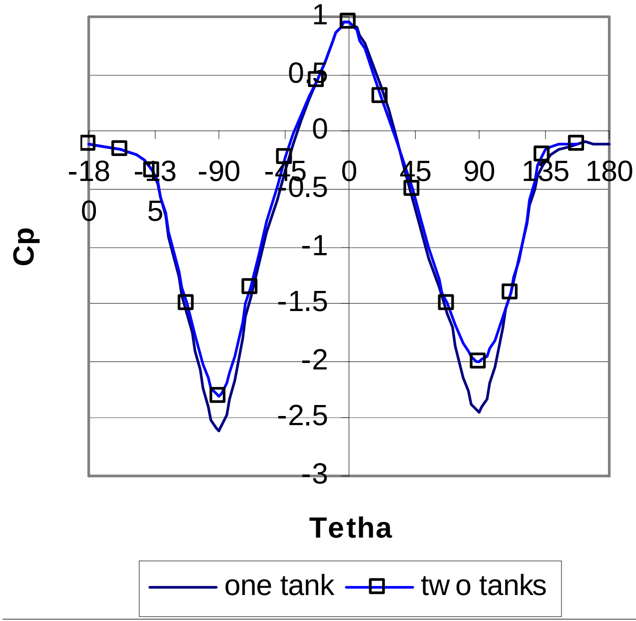

Fig. 9. Pressure distribution around tank A (located up-stream of tank B)

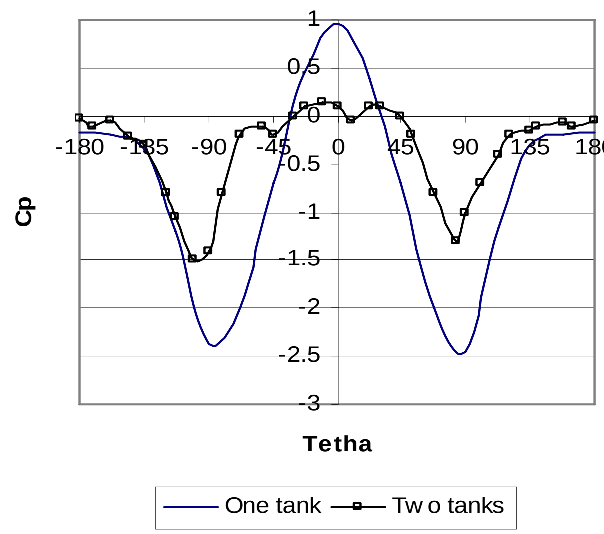

Fig. 10. Pressure distribution around tank B (down-stream of tank A)

Changes in tank B pressure distribution, where two tanks are in tandem, are illustrated in figure 10. It is obvious that the pressure distribution on tank B is highly reduced due to existence of upstream tank.

Percentage of decrease in pressure coefficient on tanks A and B after addition of their neighboring tank are tabulated in table 2. The pressure coefficient decreases up to 90% in $\theta = 0$.

Table 2. Changes on pressure coefficient on tank A and B due to existence of neighboring tank

θ	$\theta = 0$	$\theta = 90$	$\theta = 180$
Decrease in pressure coefficient on tank A	2%	17%	0%
Decrease in pressure coefficient on tank B	90%	53%	50%

VI. Conclusion

A Galerkin finite volume flow-solver, in which there is no need to apply any reconstruction method and cumbersome matrix computations is introduced for explicitly computation of the turbulent air flow variables at the nodal points of unstructured triangular meshes. In the utilized algorithm, there is no need to use the reconstruction method, and hence, it is computationally superior to the Cell Centre Finite Volume Methods and it is more accurate than the Vertex Centre Finite Volume methods particularly at the control volumes located at boundaries. Furthermore, unlike the Galerkin Finite Element Methods, the explicit nature of the formulations makes it free from matrix computations.

This wall function free software is successfully used for investigation of SGS turbulent eddy viscosity model on computation of wind pressure at supercritical Reynolds number (Re=4.5×105). From the computed results, it can be stated that complicated physical conditions around a geometrically complex object can accurately modeled without application of any wall function. However, application of coarse mesh near the no slip wall may associate with some errors in computed turbulent pressure on the solid wall of the cylinder.

For the application case of two tanks in tandem arrangement, the downstream tank is submerged in the wake of the up stream one. When the spacing ratio is small, two tanks behave as a single body. The computed results show that the interference will greatly affect the downstream tank if it locates in the wake region of the up stream tank, and, the downstream tank is wrapped in the shear layers separated from the upstream tank. In the present case study of tandem arrangement, zero computed pressure is detected on a wide area in the front part of the down stream tank.

References

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

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

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

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

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

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

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

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

[11] Yu D., Kareem A. *Two-Dimensional Simulation of Flow around Rectangular Prisms*, Journal of Wind Engineering and Industrial Aerodynamics 62, 1996, pp 131-161.

[12] Sabbagh-Yazdi S.R. and Hadian A. *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

[13] Aftosmis M. Gaitonde D., Sean Tavares T. *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.

[14] Iskandarani M., Levin JC, Choi B.J., Haidvogel D.B., *Comparison of Advection Schemes for High-Order h-p Finite Element and Finite Volume Methods*, Journal of Ocean Modeling 10, 2005, pp 233-252.

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

[16] Lieb J.M. API 650 External pressure design appendix. Tank Industry Consultants, 2003.

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