Simulation of Wind Pressure on Circular Cylinder at Super-Critical Reynolds Number

SAEED-REZA SABBAGH-YAZDI*, FARZAD MEYSAMI AZAD **
Civil Engineering Department,
KN Toosi University of Technology,
No.1346 Valiasr Street, 19697- Tehran
IRAN

and

NIKOS E. MASTORAKIS ***
Military Insitutes of University Education (ASEI)
Hellenic Naval Academy
Terma Chatzikyriakou 18539,
Piraues, GREECE ***

Abstract: Application of the computer simulation for solving the incompressible flow problems motivates developing efficient and accurate numerical models. In this paper, the accuracy of two-dimensional incompressible flow solver of the Numerical Analyzer for Scientific and Industrial Requirements (NASIR) for the solution flow around circular cylinder at supercritical Reynolds number is assessed by comparison of computed results with experimental coefficient of pressure measurements. 2D Navier-stokes equations for an incompressible fluid combined with a SGS eddy viscosity model to simulate turbulent viscosity flow and satisfactory results are obtained by the use of proper boundary conditions.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.

Key words: SGS Turbulent Viscosity, Triangular Unstructured Finite Volumes, NASIR Wind Flow-Solver.

1 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 usage of evolutionary computing in partial differential equations is also a promising idea and has been introduced by Mastorakis [12], [13].

In this paper, the ability of the NASIR (Numerical Analyzer for Scientific and Industrial Requirements) finite volume solver is applies to simulate wind flow at supercritical Reynolds number ($\mathrm{Re} = 4.5 \times 10^5$) on the pressure distribution on circular cylinder is presented and discussed. In this software 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. Some research has been done by Salvett on large eddy simulations of the flow around circular cylinders [10]. Also Murakami has done useful researches on numerical modeling of flow past 2D cylinders and CFD analysis of wind [11].

2 Model Formulation

2.1 Governing Equations

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 large eddy simulation (LES) of the flow around circular cylinder. The non-dimensional form of the governing equations in Cartesian coordinates can be written as:

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

Where,

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

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

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

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 $\rho_o U_o^2$.

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 [1]. 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 [2].

Some algorithms have used constant value of pseudo compressibility parameter and some workers have developed sophisticated algorithms for solving mixed incompressible and compressible problems [3]. However, the value of the parameter may be considered as a function of local velocity using following formula proposed [4]

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

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 [5].

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 [6]. In present paper, the primary interest is to develop a method of obtaining steady-state solutions.

2.2 Numerical Formulations

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:

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

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

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 nth and the n+1th 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 Smagorinsky model is used for the Sub-Grid Scale (SGS) turbulence viscosity. Eddy viscosity $\nu_t=\nu_{SGS}$ is computed as follow [7]:

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

$$(\bar{s}_{ij}\bar{s}_{ij})^{1/2}=\left|\bar{s}_{ij}\right|=\sqrt{2(\frac{\partial u}{\partial x})^2+2(\frac{\partial v}{\partial y})^2+(\frac{\partial u}{\partial y}+\frac{\partial v}{\partial x})^2} \qquad (4)$$

Where, $i, j=1,2$ 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:

$$\Delta x_k=x_{n_2}-x_{n_1},\ \Delta y_k=y_{n_2}-y_{n_1} \qquad (5)$$

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;

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

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

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

$$\nabla^4 W_i=\varepsilon_4\sum_{j=1}^{Ne}\lambda_{ij}(\nabla^2 W_j-\nabla^2 W_i) \qquad (6)$$

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).

$$\nabla^2 W_i=\sum_{j=1}^{Ne}(W_j-W_i) \qquad (7)$$

In equation 6, 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 [6]:

$$\lambda_i=\sum_{k=1}^{Ne}\left|(u.\Delta y-v.\Delta x)_k\right|$$
$$+\sum_{k=1}^{Ne}\sqrt{(u.\Delta y-v.\Delta x)^2{}_k+(\Delta x^2+\Delta y^2)_k} \qquad (6)$$

3 Solution Results

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).

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[9].

The results on the cylinder wall at supercritical Reynolds number ($\mathrm{Re}=4.5\times10^5$) 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 [9] 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.

4 Conclusion

The NASIR (Numerical Analyzer for Scientific and Industrial Requirements) flow-solver is successfully used for investigation of SGS turbulent eddy viscosity model on computation of wind pressure at supercritical Reynolds number ($Re = 4.5 \times 10^5$). 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 two-dimensional model show that, there are differences in computed pressure fields on the wall surface of the circular cylinder due to application of SGS turbulent eddy viscosity model. Such an efficient algorithm for computation of both velocity and pressure fields on certain Cartesian unstructured grid facilitates future modeling of the incompressible flow problems.

References:

[1] Chorin A. ,“A Numerical Method for Solving Incompressible Viscous Flow Problems”, Journal of Computational Physics, Vol. 2, (1967), 12-26.

[2] 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 (1984) Reno.

[3] Turkel E., Preconditioning Methods for Solving the Incompressible and Low Speed Compressible Equations", ICASE Report 86-14 , (1986)

[4] Dreyer. J., “Finite Volume Solution to the Steady Incompressible Euler Equation on Unstructured Triangular Meshes”, M.Sc. Thesis, MAE Dept., Princeton University, (1990).

[5] Rizzi A. and Eriksson L., "Computation of Inviscid Incompressible Flow with Rotation", Journal of Fluid mechanic, Vol. 153, (1985), 275-312.

[6] 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 (1995) Reno.

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

[8] John M.Lieb, “API 650 External pressure design appendix”, Tank Industry Consultants, November,(2003).

[9] Zhifa Gu, “On interference between two circular cylinders at supercritical Reynolds number”, Journal of Wind Engineering and Industrial Aerodynamics 62 , pp. 175-190 (1996).

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

[11]S. Murakami and A. Mochida, "On turbulent vortex shedding flow past 2D square cylinder predicted by CFD, J. Wind Eng. Ind. Aerodyn. 54/55, pp.191-211 (1995).

[12] Nikos E. Mastorakis, “Genetic Algorithms and Nelder-Mead Method for the Solution of Boundary Value Problems with the Collocation Method”, WSEAS Transactions on Information Science and Applications, Issue 11, 2005

[13] Nikos E. Mastorakis, “Numerical Solution of Non-Linear Ordinary Differential Equations via Collocation Method (Finite Elements) and Genetic Algorithms”, WSEAS Transactions on Information Science and Applications, Issue 5, 2005

Figures:

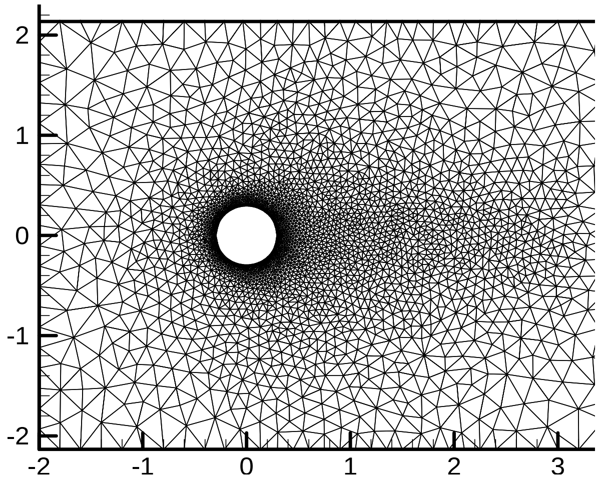

Fig. 1, Computational domain of the problem

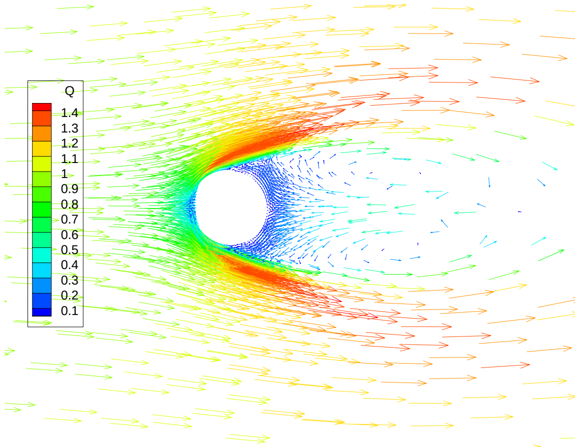

Fig.2, Computed velocity vectors at $Re = 4.5 \times 10^5$

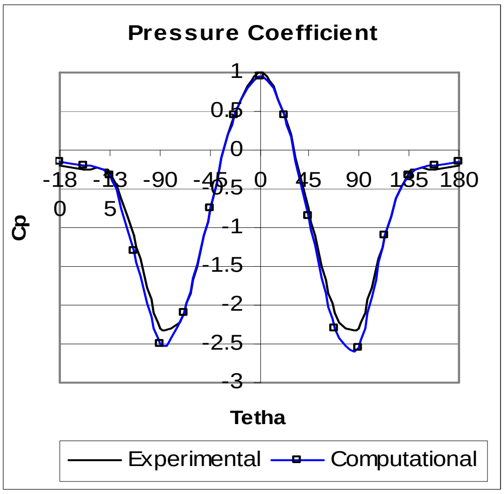

Fig.3, Coefficient of pressure on cylinder walls, (Numerical results without turbulent viscosity)

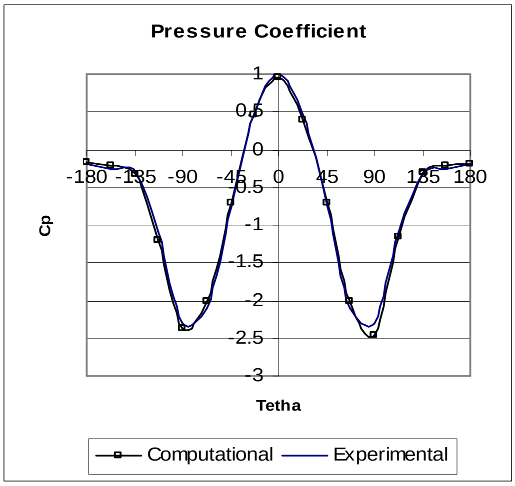

Fig. 4, Coefficient of pressure on cylinder walls, (Numerical results with SGS turbulent viscosity)

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%

.