Iterative Imposing Near Wall Logarithmic Velocity Profile in Turbulent Flow Pressure Modeling over Cylindrical for Unstructured Triangular Meshes

FATEMEH NAMAZI-SALEH *, SAEED-REZA SABBAGH-YAZDI **
Civil Engineering Department
K.N Toosi University of Technology
No.1346 Valiasr Street, 19697- Tehran, IRAN

* M.Sc. Student (Namazi.82@Gmail.com)
** Associate Professor (SYazdi@kntu.ac.ir)

Abstract: - In this paper, the effect of imposing logarithmic law for velocity profile on turbulent flow pressure distribution computation over a cylindrical body using unstructured triangular meshes is investigated. For this purpose, the two-dimensional incompressible module of the Numerical Analyzer for Scientific and Industrial Requirements (NASIR) software is utilized. The computational difficulty for Navier-Stokes equations for an incompressible fluid is resolved using artificial compressibility method and the set of equations is combined with SGS eddy viscosity model to simulate turbulent viscosity flow. The quality of solution results for the flow around circular cylinder at supercritical Reynolds number are assessed by comparison of computed results with experimental coefficient of pressure measurements. Satisfactory results are obtained by the imposing logarithmic law for velocity profile normal to the cylinder wall, as proper wall boundary conditions for relatively coarse mesh.


1 Introduction
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 two-dimensional incompressible flow solver module of the NASIR (Numerical Analyzer for Scientific and Industrial Requirements) finite volume solver for solution turbulent flow the over cylindrical is utilized in this work. 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. 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.

The abilities of the two-dimensional incompressible flow solver module of the above mentioned finite volume method software for solution of the inviscid and viscous laminar (boundary layer) as well as turbulent flow (without logarithmic velocity profile implementation) are presented in the literature [9]. In addition, the solution two-dimensional inviscid flow over a cylindrical cylinder [10] and examination of imposing logarithmic velocity profile over a flat plate [11] are reported by the authors.

Review of numerical works shows that imposing logarithmic velocity profile normal to the wall has widely used for proper turbulent boundary layer simulation considering wall friction effect [12]. However, most of these works are suitable for structured meshes (with regular node numbering) and straight boundaries (flat walls). Although extensive works are reported for the numerical simulation of turbulent flow over circular cylinder and associate wakes [13], lack of reports for algorithms suitable for imposing logarithmic velocity profile normal to curved wall for unstructured meshes motivates present work.

Here, the development of the software (by imposing logarithmic velocity profile normal to curved wall for unstructured meshes) for solution of two-dimensional incompressible flow is applied and wind flow over a cylindrical body at supercritical Reynolds number (Re = 4.5 x 10^5) is simulated and the computed pressure distribution on circular cylinder is presented and
discussed by comparison of computed results with available experimental measurements is presented [8]. The computed pressure coefficients with and without imposing velocity law logarithmic are compared with the readily available measurements.

2 Model Formulation

2.1 Governing Equations

2.1.1 Navier-Stokes equations

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:

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

(1)

where,

\[
W = \begin{pmatrix} \frac{p}{\rho_0} \\ \beta^2 \\ u \\ v \end{pmatrix}, \quad F^c = \begin{pmatrix} u \\ u^2 + p/\rho_0 \\ uv \\ uv \end{pmatrix}, \quad G = \begin{pmatrix} v \\ uv \\ v^2 + p/\rho_0 \end{pmatrix}
\]

\[
F^v = \begin{pmatrix} 0 \\ v_T \frac{\partial u}{\partial x} \\ v_T \frac{\partial v}{\partial y} \end{pmatrix}, \quad G^v = \begin{pmatrix} 0 \\ v_T \frac{\partial u}{\partial y} \\ v_T \frac{\partial v}{\partial x} \end{pmatrix}
\]

\[
W \text{ represents the conserved variables while, } F^c, G^c \text{ are the components of convective flux vector and } F^v, G^v \text{ are the components of viscous flux vector of } W \text{ in non-dimensional coordinates } x \text{ and } y, \text{ respectively, } u \text{ and } v \text{ the components of velocity, } p \text{ pressure are three dependent variables. } v_T \text{ is the summation of kinematic viscosity } \nu \text{ and eddy viscosity } \nu_t.
\]

The variables of above equations are converted to non-dimensional form by dividing and by length \( u \) and \( v \) by \( u_o \), upstream wind velocity, and \( \rho \) by \( \rho_0U_o^2 \).

The parameter \( \beta \) 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 artificial 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].

\[
\beta^2 = \max(\beta(U^2), \beta_{\min})
\]

(2)

In order to prevent numerical difficulties in the region of very small velocities (ie, in the vicinity of stagnation points), the parameter \( \beta_{\min}^2 \) is considered in the range of 0.1 to 1, 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 this study, the Sub-Grid Scale (SGS) model is used for computation of turbulence eddy viscosity. Therefore, Turbulent eddy viscosity \( \nu_T = \nu_{SGS} \) is computed as follow [7]:

\[
\nu_{SGS} = (C_sA)^2 \left[ \frac{1}{2} \bar{\nu}_{ij} \bar{\nu}_{ij} \right]^{1/2}
\]

(3)

\[
\left[ \bar{\nu}_{ij} \right]^{1/2} = \bar{\nu}_{ij} = \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}
\]

(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 $\nu_{SGS}$, where $\Delta$ is the area of a triangular cell and the $C_s = 0.15$ are used. In equation (4), $\vec{u}$, $\vec{v}$ are mean values of velocity in each edge of the triangular element

2.1.2 Near Wall Velocity Profile
Close to the wall boundary sharp velocity gradients form due to molecular viscosity of the fluid, and therefore, boundary layer develops. In this layer the effect of wall surface on generation of viscous and turbulent stresses are pronounce. Because of high velocity gradient at fluid flow closed to the high resolution computational mesh is an essential requirement. However, this requirement may be relaxed by the use of algebraic formulation of velocity variations in the vicinity of the wall. In this concept the surface roughness plays an important role in the algebraic formulas of near wall velocity profile. In this work the effect wall velocity profile law in pressure coefficient is discussed. For high Reynolds number flow over smooth walls, this law is formulated using dimensionless velocity and distance normal to wall surface as:

$$u^* = \frac{1}{\kappa} \ln(Ey^*) \quad \text{or} \quad u^* = \frac{1}{\kappa} \ln(y^*) + 5.5 \quad (7)$$

Where, $\kappa = 0.42$ and $E = 9.793$. The shear stress in the second zone is considered equal to wall shear stress.

2.2 Numerical method
The governing equations can be changed to discrete form for the unstructured meshes by the application of the Galekin Finite Volume Method. This method ends up with the following 2D formulation:

$$\frac{W_i^{n+1} - W_i^n}{\Delta t} = -\frac{P}{\Omega_i} \sum_{k=1}^{N_{cell}} \left[ F^e_{\Omega_i} (\Delta y) - G^e_{\Omega_i} (\Delta x) \right]_{k}^{n}$$

$$+ \frac{P}{\Omega_i} \sum_{k=1}^{N_{cell}} \left[ F^v_{\Omega_i} (\Delta y) - G^v_{\Omega_i} (\Delta x) \right]_{k}^{n} \quad (8)$$

where, $W_i$ represents conserved variables at the center of control volume $\Omega_i$. Here, $F^e$, $G^e$ 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 $(n+1)$th computational steps. $\Delta t$ 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.

$\Delta x, \Delta y$ for the edge of $k$ in control volume are computed as follow:

$$\Delta x_k = x_{n2} - x_{n1}, \Delta y_k = y_{n2} - y_{n1} \quad (9)$$

According to the last equations $\nu_T$ is the summation of kinematic viscosity $\nu$ and eddy viscosity $\nu_T$. To discrete the following formulation can be obtain:

$$\nu_T = (C_\nu \Delta)^2 \frac{1}{\sqrt{2}} A \sum_{k=1}^{N_{cell}} |u \Delta y - v \Delta x| \quad (10)$$

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, \( C(W_j) \) and viscous, \( D(W_j) \) terms. Where:

\[
C(W_j) = \sum_{k=1}^{N_{edge}} [F^c_{ij} \Delta y - G^c_{ij} \Delta x]
\]  
\[
D(W_j) = \sum_{k=1}^{N_{node}} [F^v_{ij} \Delta y - G^v_{ij} \Delta x]
\]

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

\[
\nabla^4 W_i = \varepsilon_a \sum_{j=1}^{N_{edge}} \lambda_{ij} (\nabla^2 W_j - \nabla^2 W_i)
\]  

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

\[
\nabla^2 W_i = \sum_{j=1}^{N_{edge}} (W_j - W_i)
\]

In equation 13, \( \lambda_{ij} \), 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]:

\[
\lambda_k = \sum_{k=1}^{N_{edge}} \left[ U_k (\Delta S_k) + \sqrt{[U_k (\Delta S_k)]^2 + (\Delta S_k)^2} \right]
\]

\[
\lambda_{ij} = \min(\lambda_k, \lambda_{ij})
\]

To use of algebraic formulation of velocity, it should be calculate tangential wall shear stress on triangular unstructured mesh. According to the Green’s theory, to calculate of shear stress at the centre of triangular element adjacent to the wall boundary, the following formulation can be used [14]:

\[
\tau_{xx} = \frac{\partial u}{\partial x}, \tau_{xy} = \tau_{yx} = \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}, \tau_{yy} = \frac{\partial v}{\partial y}
\]

Where:

\[
\tau_{xx} = \frac{1}{A} \sum_{k=1}^{N_{edge}} (u \Delta y)_k
\]

\[
\tau_{xy} = \frac{1}{A} \sum_{k=1}^{N_{edge}} (v \Delta y - u \Delta x)_k
\]

\[
\tau_{yy} = \frac{1}{A} \sum_{k=1}^{N_{edge}} (v \Delta x)_k
\]

To calculate tangential wall shear stress on curved surface, Mohr technique can be used [15]:

\[
\tau_{wall} = -\frac{\tau_{xx} - \tau_{yy}}{2} \sin(2\theta) + \tau_{yx} \cos(2\theta)
\]

Where, is the \( \theta \) angle of boundary edge respect to the global \( x \) direction. According to the equations 5 and 6, the value of tangential velocity, \( y' \), and velocity in center of cell, can calculate. For the cases that the centre of the triangular element adjacent to the wall boundary is out of the laminar sub-layer, an iterative process may be need to modify the above mentioned computed shear stress.

\[
\tau_{wall} = \rho \left[ \frac{1}{\kappa} \ln \left( \frac{d_n}{\nu} \sqrt{\frac{\tau_{wall}}{\rho}} + 5.5 \right) \right]^2
\]

Here, \( d_n \) is the normal distance of the node \( n \) (inside domain node of the triangular element adjacent to the wall boundary) and \( v_n \) is the boundary tangent velocity at that node.

Having calculated wall shear stress \( \tau_{wall} \) at each iterative modification, the value of tangential velocity \( v_n \) at near wall node \( n \) can be updated using following relation:

\[
v_n = \sqrt{\frac{\tau_{wall}}{\rho} \left[ \frac{1}{\kappa} \ln \left( \frac{d_n}{\nu} \sqrt{\frac{\tau_{wall}}{\rho}} + 5.5 \right) \right]^2}
\]

The iterative modifications of \( \tau_{wall} \) and \( v_n \) may stop when the difference of sequential values for the tangential velocity \( v_n \) is less than 0.0001 at each node \( n \) in the vicinity of wall boundary. In such a condition, the logarithmic relation between the parameters \( v_n \) and \( \tau_{wall} \) is satisfied close to the wall boundary.

3 Solution Results

In order to assess the changes of pressure on a single circular cylinder, the developed flow solver is applied to solve the turbulence flow with and without logarithmic law on a mesh of unstructured triangles (Fig. 2).

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 of two conditions of with and without imposing logarithmic law are compared in this section.

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, in which the maximum inflow velocity was
considered as 50 m/s for supercritical Reynolds number \((\text{Re} = 4.5 \times 10^5)\) condition [8].
The computed results on the cylinder wall at supercritical Reynolds number \((\text{Re} = 4.5 \times 10^5)\) are
plotted in terms of stream line colored with velocity values and color coded map of pressure (Fig. 3).
Distributions of the mean (time average) coefficient of pressure on cylinder wall are compared with the
experimental measurements for [9] two conditions of with and without imposing logarithmic law (Fig. 4).
In these figures the computed pressure coefficient are computed using following relation:
\[
C_{pl} = \frac{(P_i - P_o)}{0.5 \rho U^2}
\]  
(20)
Table 1 shows the errors in computed pressure coefficient due to application of iterative imposing
logarithmic velocity profile in the vicinity of wall boundary. The errors are computed using following relation:
\[
\text{Error} = \left| \frac{w_{\text{comp}} - w_{\text{exp}}}{w_{\text{exp}}} \right| \times 100
\]  
(21)

Fig. 3, Computed results at supercritical Reynolds number \(\text{Re} = 4.5 \times 10^5\)

Fig. 4, Mean coefficient of pressure on cylinder walls,

Table 1, The average and maximum errors of the computed \(C_p\)
4 Conclusion
The NASIR (Numerical Analyzer for Scientific and Industrial Requirements) flow-solver is successfully developed for investigating the effects of imposing near wall logarithmic velocity profile for numerical simulation of turbulent flow and computation of pressure on cylindrical cylinder for which vortex shedding may form. For this purpose, an iterative process of modification of two parameters of wall tangential shear stress $\tau_w$ and the boundary tangent velocity $v_n$ at that inside domain node $n$ of every unstructured triangular element adjacent to the curved wall boundary of cylinder.

The computations of wind pressure on a circular cylinder are performed at supercritical Reynolds number ($Re = 4.5 \times 10^5$) for two conditions of with and without imposing logarithmic law. 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 logarithmic law. It can be clearly seen that, the percentage of error in pressure coefficient around circular cylinder is less than without use of logarithmic law algorithm.

From the computed results, it can be stated that complicated physical conditions around a geometrically complex object can accurately modeled using the developed flow solver in which the near wall turbulent (logarithmic) velocity profile is imposed iteritively.

References: