PRESSURE-VELOCITY COUPLED FINITE VOLUME SOLUTION OF STEADY INCOMPRESSIBLE INVISCID FLOW USING ARTIFICIAL COMPRESSIBILITY TECHNIQUE

S. R. Sabbagh Yazdi and A. Hadian

Department of Civil Engineering, KNT University of Technology
Tehran, Iran, SYazdi@kntu.ac.ir

(Received: August 6, 2003-Accepted in Revised Form: February 26, 2004)

Abstract Application of the computer simulation for solving the incompressible flow problems motivates developing efficient and accurate numerical models. The set of Inviscid Incompressible Euler equations can be applied for wide range of engineering applications. For the steady state problems, the equation of continuity can be simultaneously solved with the equations of motion in a coupled manner using the Artificial Compressibility Technique. This technique helps coupling the pressure and the velocity fields during the explicit computation procedure of the incompressible flow problems and therefore speeds of the convergence of the solution. The discrete form of the two-dimensional flow equations are formulated using the Cell Vertex Finite Volume Method for triangular unstructured meshes. Using triangular unstructured meshes provides great flexibility for modeling the flow in geometrically complex domains. Some numerical techniques adopted for the unstructured meshes are used to stabilize and accelerate the explicit solution procedure without degradation of accuracy. In order to verify the accuracy of the numerical model, computed results are compared with the analytical solutions of potential flow problems.

Key Words Steady Incompressible Inviscid Flow, Artificial Compressibility Technique, Finite Volume Method, Triangular Unstructured Meshes

چکیده رواج شبیه‌سازی رایانه‌ای برای تحلیل جریانهای تراکم‌ناپذیر توسعه مدلهای عددی کارآمد و دقیق را ایجاب مینماید. دسته معادلات تراکم‌ناپذیر غیرلزج اولر برای محدوده وسیعی از مسائل مهندسی قابل کاربرد میباشد. برای جریانهای دائمی با بکارگیری فن تراکم‌پذیری مصنوعی میتوان معادله پیوستگی را بصورت همزمان با معادلات حرکت حل نمود. این فن با درگیر نمودن میدانهای فشار و سرعت در خلال روند محاسبات صریح جریانهای تراکم‌ناپذیر به تسریع همگرائی نتایج حل کمک مینماید. در کار حاضر معادلات دو بعدی جریان با روش رئوس سلول احجام محدود برای شبکه‌های بی‌ساختار بشکل گسسته فرمول‌بندی شده‌اند. استفاده از شبکه‌های بی‌ساختار انعطاف‌پذیری زیادی را در امر مدلسازی هندسی محیط جریان بدست میدهد. برای پایدارسازی بدون کاهش دقت و نیز سرعت بخشیدن به روند حل صریح بر روی شبکه‌های بی‌ساختار فن‌آوریهائی مورد استفاده قرار گرفته است. بمنظور ارزیابی کیفیت نتایج مدل عددی میدانهای فشار و سرعت محاسبه شده برای چند آزمون دارای پاسخ تحلیلی مورد استفاده قرار گرفته است.

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. In this respect, many attempts have been made to develop several efficient and accurate numerical methods suitable for the complex solution domain.

The assumption of incompressibility is valid for common civil and environmental engineering problems. For the most civil engineering flow problems, the boundary layer is confined to thin regions close to the solid surfaces. Since these regions are negligible comparing to the main domain of interest, the effect of viscous stresses can be omitted in the equations of the motion. The resulted set of equations, which is known as the incompressible form of the Euler equations, provides considerable simplicity in the absence of second order spatial derivative terms. This simplification of the governing equations provides the ease of the solution procedures, and consequently, saves the computational efforts.

For the incompressible flow condition, the time derivative of the density vanishes from the continuity equation. If the boundary layer thickness is negligible in the flow domain, the inviscid form of the equations of motion can be used in desired dimensions. These set of equations which consists of time-independent velocity and the time-dependent equations of motion, mathematically represent the behavior of fluid flow. For steady state problems, adding a pseudo time derivative of pressure to the continuity equation removes the troublesome problem of coupling pressure-independent equation of continuity to the pressure-dependent equations motion. This method has been widely applied, mostly with the use of explicit schemes. The computational procedure is to choose the pressure field such that continuity is satisfied at each time-step. This procedure normally requires a relaxation scheme iterating on pressure until the divergence free condition is reasonably realized. The method using Artificial Compressibility was initially proposed by Chorin to achieve an efficient computation. Note that, when the solution converges to the steady state condition, the pseudo time derivative tends to zero and the result of computations results in the incompressible flow solution [1].

In present work, the Cell Vertex Finite Volume Method is used to derive the discrete formulas of the governing equations on triangular unstructured meshes. The problem of growing up numerical errors, which usually disturbs the explicit solution of the formulations are overcome by adding artificial dissipation terms suitable for the unstructured meshes. These extra terms are used to damp out the unwanted errors and stabilize the numerical solution procedure while preserving the accuracy of the solution. In order to increase the computational efficiency, some numerical technique such as Runge-Kutta multi-stage time stepping, residual smoothing and the edge-base algorithm are applied.

In this paper, the accuracy of the described algorithm for the solution of the inviscid incompressible flow equations is assessed by simulation of some simple test cases for which the analytical solution of the velocity and pressure field can be obtained by application of the potential function. The results are demonstrated using comparison of velocity and pressure fields. The agreements of the computed and exact solutions encourages for further developments of the model.

2. ARTIFICIAL COMPRESSIBILITY TECHNIQUE

For the flow with high Reynolds number, the boundary layer is thin and limited to a thin layer close to solid walls. In such cases, the effects of viscosity are ignorable in the major part of the flow field. The assumption of inviscid behavior of the fluid flow is acceptable for the regions outside the boundary layer.

In the subsonic flow problems (Mach<0.3), since the density is constant, the fluid flow is considered incompressible. Considering the isothermal condition for the flow problem, the equations of continuity and motions represent the mathematical model equations of the incompressible flow, which is known as the incompressible Euler equations.

Due to negligible variation in the density, there is no time derivative term in the continuity equation. This matter presents some numerical difficulties for the coupled solution of the continuity equation (zero velocity divergence) together with some time dependent equations (the equations with pressure and velocity components as the dependent variables). For the steady state incompressible problems, the Artificial Compressibility technique helps to overcome this numerical solution problem [1]. In this technique, a time derivative of the pressure, which is derived from an equation similarity to the equation of state of the compressible gases, is added to the continuity equation. This transient term, which relates the pressure field to the velocity field, vanishes when the solution procedure converges to the steady state condition.

The conservative vector form of the governing equations in Cartesian coordinates can be written as:

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

where:

$$W=\begin{pmatrix} p/(\rho_o\beta^2) \\ u \\ v \end{pmatrix}$$

$$F=\begin{pmatrix} u \\ u^2+p/\rho_o \\ uv \end{pmatrix}$$

$$G=\begin{pmatrix} v \\ uv \\ v^2+p/\rho_o \end{pmatrix}$$

W represents the conserved variables and *F* and *G* are vectors of convective fluxes of *W* in *x* and *y* directions, respectively. The components *u* and *v* of velocity and pressure *p* are three dependent variables by considering ρ_o as the constant density. The parameter β is introduced using the analogy to the speed of sound in equation of state of compressible flow, by application of the Artificial Compressibility technique [1].

In the above equations, the first row represents the incompressible continuity equation modified according to the Artificial Compressibility technique. The second and third rows correspond to the equations of motion in *x* and *y* directions, respectively.

The system of equations governing the motion of an incompressible flow is of the elliptic type. In elliptic formulation, pressure waves propagate with infinite speed. However, the system of modified equations given by the modified continuity equation and equations of motion is of the hyperbolic type. Thus, in the present formulation, waves of finite speed are introduced. The magnitude of the wave speed depends on the parameter β. Therefore, the success of the present method depends on the value of β that must be used for fast convergence to steady-state solutions, and whether the incompressibility is really achieved within desired accuracy by the use of above equations. These important points are analyzed in the literature [2].

The use of the Artificial Compressibility technique results in a system of hyperbolic-type equations of motion. Waves of finite speed are introduced into the incompressible flow field as a medium to distribute the pressure. For a truly incompressible flow, the wave speed is infinite, whereas the speed of propagation of these pseudo waves depends on the magnitude of the pseudo compressibility. 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 and, if the Artificial 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 Artificial 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].

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

In order to prevent numerical difficulties in the

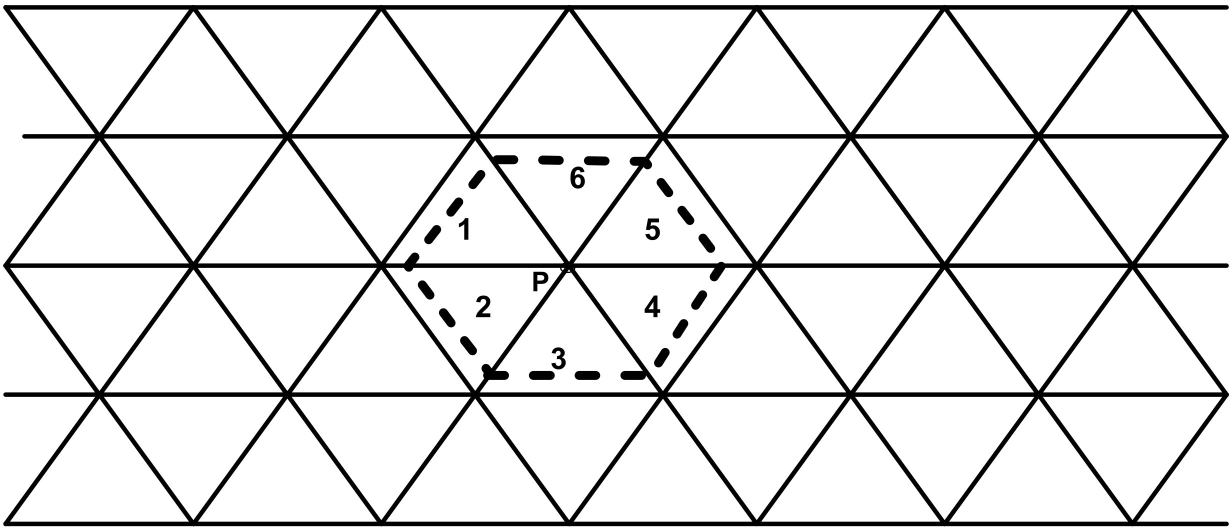


Figure 1. A control volume formed by the triangles sharing a node.

region of very small velocities (i.e., in the vicinity of stagnation pints), 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].

Note that the above equation of continuity differs from the continuity equation for a real compressible flow by the absence of the convective term for the pressure. The absence of this term in the present formulation is an essential feature of this method. Starting from an arbitrary initial condition, the numerical scheme must be chosen such that the solution of above equations of continuity and motion for fixed boundary conditions converges to a steady state as the computation progress. As the steady state is approached, the effect of the pseudo compressibility diminishes, resulting in an incompressible solution.

The method of the artificial compressibility can also be used to solve unsteady problems. For this propose, by considering additional pressure transient term in the modified continuity equation using the previous pressure field. 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 [6-8]. In present paper, the primary interest is in developing a method of obtaining steady-state solutions.

3- NUMERICAL METHOD

The governing equations can be changed to discrete form for the unstructured meshes by the application of Cell Vertex (overlapping) scheme of the Finite Volume Method. This method ends up with the following formulation [9]:

$$W_i^{n+1} = W_i^{n} - \frac{\Delta t}{\Omega_i} . \sum_{k=1}^{N_{sides}} (\overline{F}\Delta y - \overline{G}\Delta x)_k^n \qquad (3)$$

where W_i represents conserved variables at the center of control volume Ω_i (Figure 1). $\overline{F}$ and $\overline{G}$ are the mean values of fluxes on the control volume boundary sides. Here, superscripts n and $n+1$ show nth 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 [10].

The explicit solution of above formulation on the equally spaced grids presents the behavior of the central differencing schemes. These schemes do not provide any dissipation mechanism that would compensate the absence of damping nature of physical viscosity near the high gradient regions. 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 term, $C(W_i) = \sum_{kk=1}^{N_{sides}} (\bar{F}\Delta y - \bar{G}\Delta x)_k^n$ [11].

The numerical dissipation term, $D(W_i) = \varepsilon \sum_{k=1}^{N_{edges}} \lambda_k (\nabla^2 W_j - \nabla^2 W_i)$, is formed by using the Laplacian operator, $\nabla^2 W_i = \sum_{k=1}^{N_{edges}} (W_j - W_i)$. 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).

Here, λ_k is the minimum of Λ_j, the scaling factors of the edges associated with the end nodes j 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 [9]:

$$\Lambda_j = \sum_{k=1}^{N_e} \{ |u_k \Delta y_k - v_k \Delta x_k| + \sqrt{(u_k \Delta y_k - v_k \Delta x_k)^2 + \beta^2 (\Delta x_k^2 + \Delta y_k^2)} \} \quad (4)$$

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 numerical dissipation term on the specific mesh ($1/256 \le \varepsilon \le 3/256$).

The revised final algebraic formula can be written in the following form [9].

$$W_i^{n+1} = W_i^n - \frac{CFL.\Delta t}{\Omega_i} [C(W_i) - D(W_i)] \quad (5)$$

The quantities W at each node is modified at every time step by adding a residual term of $R(W_i) = \Delta t[C(W_i) - D(W_i)]/\Omega_i$ which is computed using the quantities W at the nodes of boundary sides of the control volume Ω_i (Figure 1). Hence, the edges are referred to all over the computation procedure. Therefore, it would be convenient to use the edge-base data structure for definition of unstructured meshes. It has been shown that using the edge-base computational algorithm reduces the number of addressing to the memory, and therefore, provides considerable saving in computational CPU time [12].

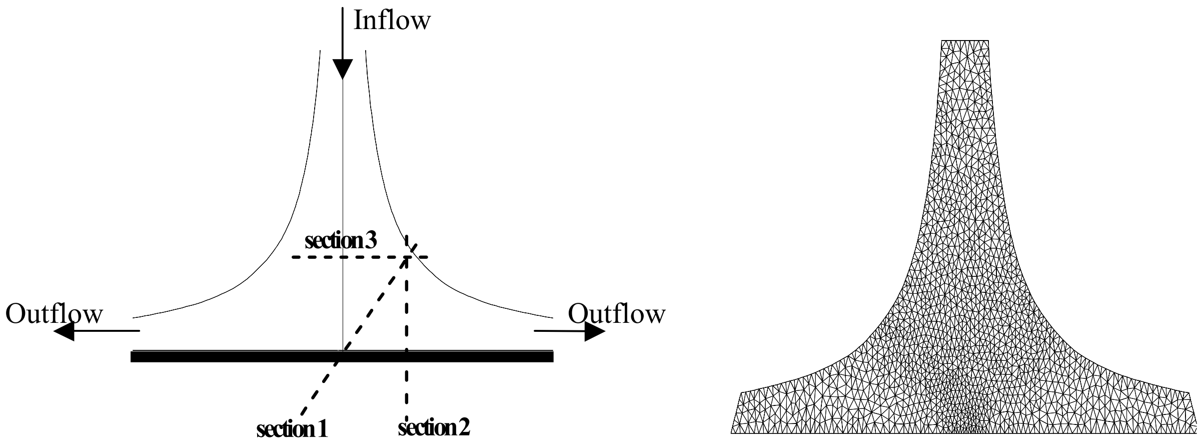
4. BOUNDARY CONDITIONS

The implementation of the flow and solid wall boundary conditions are adopted for the unstructured meshes. For incompressible flow, at the inflow boundaries free stream, values of u and v are imposed and p are extrapolated from inside domain nodes, and at the outflow boundaries free stream, p is imposed and u and v are extrapolated from inside domain nodes. The sign of the dot product of the computed velocity vector and normal vector of the flow boundary curve at computational nodes is used for distinguishing between the in-flow from the out-flow boundary.

Since the flow is considered inviscid, at the solid wall nodes, the component of velocity vector normal to the solid wall boundary edges are set to zero and the tangential slipping velocities are imposed by a technique which suits unstructured meshes [9].

5. DOMAIN DISCRETIZATION

The solution domains are discretized using unstructured triangular mesh. This method of domain discretization facilitates geometric modeling of the flow fields with complex irregularities in boundaries. The irregular triangular mesh was produced using Deluaney



a: Sketch and position of section.

b. Unstructured triangular mesh

Figure 2. Computational domain above the flat plate.

Triangulation technique. Such a mesh generation method allows local refinement of triangular elements by using source points and lines. The use of unstructured mesh for domain discretization provides the ability of mesh refinement near the important region of the flow field, where high gradient of the flow parameters exists. In addition it provides the ability to use coarser mesh spacing at positions where the gradients of the flow parameters have small magnitudes. This increases the speed of numerical computation [13].

6. VERIFICATION OF ACCURACY

The accuracy of the developed incompressible inviscid flow solver is examined by solving some cases with available analytical solutions. The analytical solution is obtained from potential flow theory by using conformal mapping technique [14]. For numerical simulation of the case, unit free stream velocity and pressure is imposed at inflow and outflow boundaries, respectively and at the solid wall nodes slipping velocity are considered.

The test case is the incompressible inviscid flow facing a free slip wall (Figure 2). The computations are performed on an unstructured mesh containing 1483 grid points, 2665 triangular elements and 4147 faces (Figure 2.b).

The comparison of computed velocity components with the exact solution in three sections (are shown in Figure 2.a) present the accuracy of the developed model (Figures 3.a, 3.c and 3.e). The computed pressure fields along the same sections are plotted against the analytical solutions (Figures 3.b, 3.d and 3.f).

The computed results of the verification case proof the accurate performance of the algorithm to compute the flow facing a flat plate without any numerical conflict of velocity components. Despite of irregular distribution of grid spacing in the unstructured triangular mesh, symmetric turning flow is computed without application of numerical symmetric splitter wall and the stagnation point is formed in the flow field at the vicinity of the plate centre.

The computed results demonstrate the accuracy of the algorithm to compute the flow fields experiencing both stagnant and turning conditions. No unwanted circulations circulation

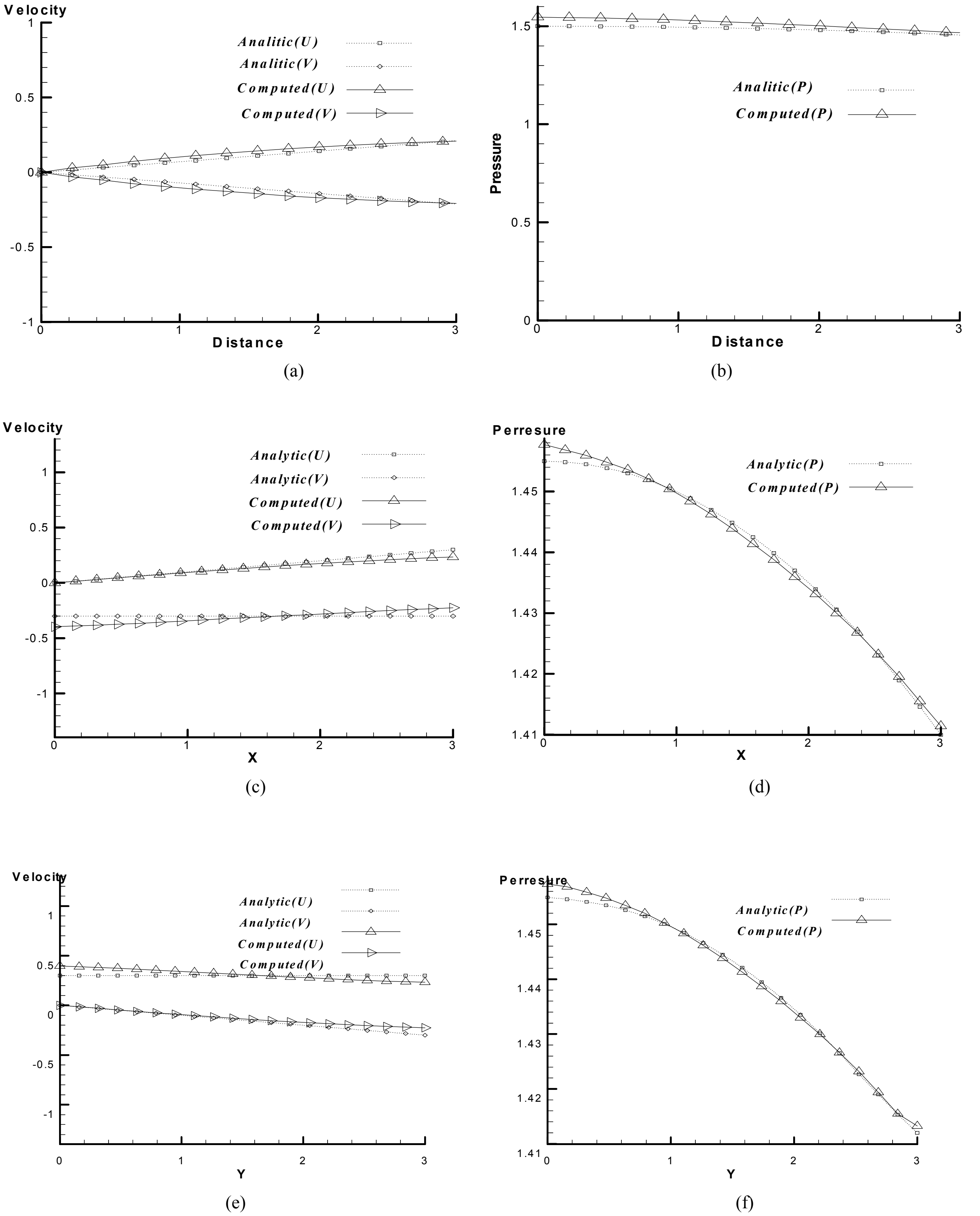


Figure 3. (a) Velocity components at Section 1, (b) nodal pressure at Section 1, velocity components at Section 2, (d) nodal pressure at Section 2, (e) Velocity components at Section 3 and (f) Nodal pressure at Section 3.

disturbs the solution in the vicinity of stagnation and turning points.

7. APPLICATION CASES

The performance of the developed solver is examined by solving incompressible inviscid flow around NACA0012 aerofoil for which both experimental measurements and analytical solutions are available [15,16]. Here, the flow solver is applied to solve two conditions of 0 and 2 degrees angles of free stream velocity. The analytical solution is obtained using conformal mapping technique in potential flow theory [16]. The computations are performed on a fully unstructured mesh (Figure 4).

Unit free stream velocity and pressure is imposed at inflow and outflow boundaries of computational mesh. Slipping velocity is considered at the solid wall nodes by setting zero normal component of computed velocity.

The free stream flow parameters (Outflow pressure and inflow velocity) are set at every computational node as initial conditions. The typical convergence of the flow parameters toward the steady state condition is demonstrated in terms of logarithm of root mean squares of pressure and velocity components. The smooth convergence behavior proofs coupling the velocity and pressure fields by the artificial compressibility technique in conjunction with the applied numerical dissipation terms and local time stepping (Figure 5).

For free stream velocity with 0 degree angle of incidence, the computed pressure field around the aerofoil performs symmetric conditions (Figure 6-a). Despite of irregular distribution of grid spacing in the unstructured triangular mesh, symmetric results are obtained without application of numerical symmetric splitter wall and the stagnation point is formed in the flow field at the aerofoil front. The comparison of the computed results and analytical solutions in terms of coefficient of pressure (C_p) on the aerofoil boundaries presents the accuracy of the developed numerical model (Figure 6-b).

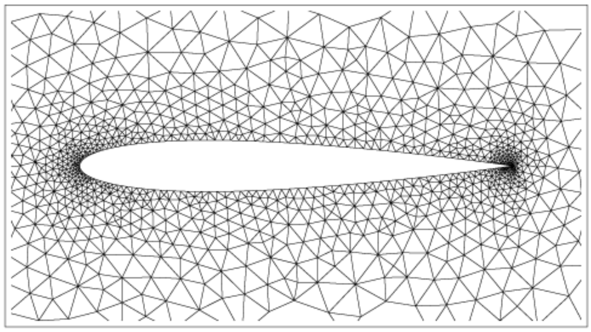


Figure 4. Unstructured triangular mesh.

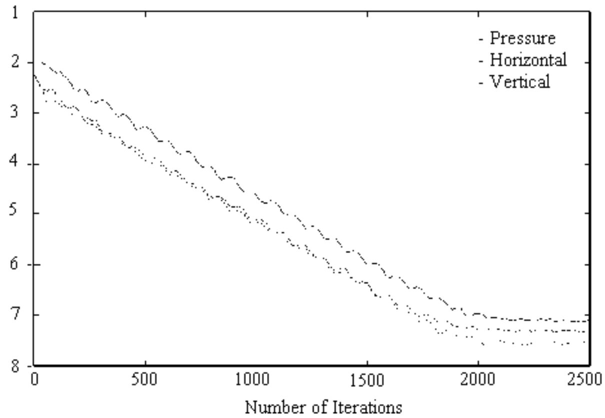


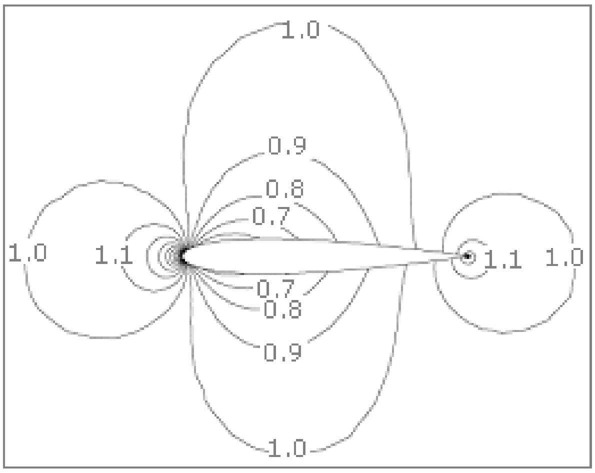
Figure 5. Typical convergence behavior (flow computation around NACA0012 aerofoil).

For the case of free stream velocity with 2 degrees of incidence, the computed pressure fields around the aerofoil section is plotted to obtain pressure contour lines (Figures 7-a). Expected difference between computed pressures on the aerofoil surface can be clearly observed. The computed coefficients of pressure on the aerofoil boundaries nicely match with the analytical solutions (Figures 7.b).

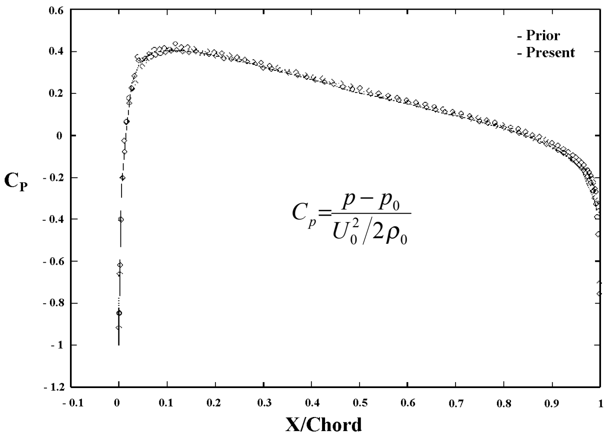
From the computed results, it can be stated that complicated physical conditions around a geometrically complex object can accurately modeled using the presented algorithm.

8. DISCUSSION

The Artificial Compressibility technique

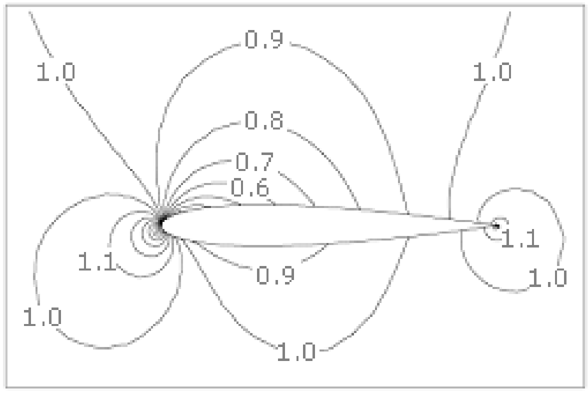


(a)

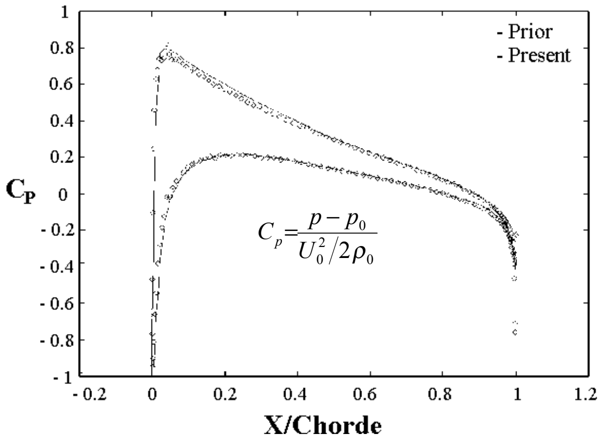


(b)

Figure 6. Plot of the computed pressure contours (a) and comparison of computed and analytical C_p (b) Flow around NACA0012 aerofoil with $\alpha = 0$ angle of free stream velocity.



(a)



(b)

Figure 7. Plot of the computed pressure contours (a) and comparison of computed and analytical C_p (b). Flow around NACA0012 aerofoil with $\alpha = 2$ angle of free stream velocity.

is used to overcome the numerical problem associated with the coupled solution of the equations of continuity and motion for the incompressible inviscid flow problems. The results of the Cell Vertex Finite Volume solution of chosen bench mark tests on unstructured meshes present promising agreements with theoretical solutions. Hence, adding a pressure time derivative term to the continuity equation successfully couples the pressure and velocity fields for speeding up the convergence behavior of the explicit solution procedure without any degradation in the accuracy of the results. Such an efficient algorithm for computation of both velocity and pressure fields on certain Cartesian unstructured mesh facilitates three-dimensional numerical modeling of the incompressible inviscid flow problems.

9. NOMENCLATURE

t	Computational time
x,y	Cartesian coordinates

p	Pressure
u,v	Cartesian velocity components
W	Conserved quantity
$F; x$	direction flux
$G; y$	direction flux
ρ_o	Density
β^2	Artificial Compressibility parameter
$C(W)$	Convective operator
$D(W)$	Numerical dissipation operator
$\nabla^2 W$	Laplacian operator
$\nabla^4 W$	Biharmonic operator
Ω	Control volume
ε	Tunable parameter
λ	Maximum eignvalue of the Jacobian
C_p	Coefficient of pressure

10. 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", ***22nd Aerospace Science Meeting and Exhibition, Reno***, (1984), AIAA 84-0252 Paper.
3. Turkel, E., Fiterman, A. and Van Leer, B., "Preconditioning Methods for Solving the Incompressible and Low Speed Compressible Equations", (1986), ICASE Report 86-14.
4. Dreyer, J., "Finite Volume Solution to the Steady Incompressible Euler Equation on Unstructured Triangular Meshes", MSc 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. Choi, D. and Markel, C. L., "Application of Time Integration Schemes to Incompressible Flow", ***AIAA Journal***, Vol. 23, No. 10, (1985), 1518-1523.
7. Rogers, S. E., "A Comparison of Implicit Schemes for the Incompressible Navier-Stokes Equations with Artificial Compressibility", ***33rd Aerospace Science Meeting and Exhibition, Reno***, (1995), AIAA Paper 95-0567.
8. Belov, A., Martinelli, L. and Jameson, A., "A New Implicit Algorithm with Multi-Grid for Unsteady Incompressible Flow Calculations", ***33rd Aerospace Science Meeting and Exhibition, Reno***, (1995), AIAA Paper, 95-0049.
9. Sabbagh Yazdi, R. S., "Simulation of the Incompressible Flow Using the Artificial Compressibility Method", PhD Thesis, University of Wales, Swansea, (1997).
10. Jameson, A., Schmidt, W. and Turkel, E., "Numerical Solution of the Euler Equations by Finite Volume Method using Runge Kutta Time Stepping Schemes", ***AIAA 14th Fluid and Plasma Dynamics Conference, California***, (June 1981), AIAA Paper 81-1259.
11. Sabbagh-Yazdi, R. S., "Using Artificial Viscosity on Unstructured Meshes for Solution of Steady Incompressible Inviscid Equations", ***6th Iranian Aerospace Engineering Association Conference***, Tehran, (2003), 538-548.
12. Sabbagh Yazdi, R. S., "Face-Base Algorithm Solution of Incompressible Inviscid Equations on Unstructured Meshes", ***1st International Conference of Aerospace Engineering, Tehran***, (2000), 147-160.
13. Thompson Joe, F., Soni, Bharat K. and Weatherill, Nigel P., "Hand Book of Grid Generation", CRC Press, New York, (1999).
14. Vallentine, H. R., "Applied Hydrodynamics", Butterworths, S.I. Edition, London, (1969).
15. Eriksson L. E, "Calculation of Two Dimensional Flow Wall Interface for Multi–Component Aerofoil in Closed Low Speed Tunnel", The Aeronautical Research Institute of Sweden FFATN AU-1116, Part 1, Stockholm, (1975).
16. Peraire J., "Potential Flow Past Arbitrary Aerofoils using Conformal Mapping", Private Communication.