World Applied Sciences Journal 5 (6): 680-685, 2008
ISSN 1818-4952

Sensitivity Analysis of Artificial Compressibility and Artificial Dissipation Parameters for Solution of Inviscid Flow on Unstructured Triangular Meshes

Saeed-Reza Sabbagh-Yazdi and Fatemeh Namazi-Saleh

Department of Civil Engineering, K.N. Toosi University of Technology, 1346 Valiasr St. Tehran, Iran

Abstract: In this study, the incompressible form of the mass and momentum conservation equations are solved in a coupled manner using artificial compressibility Method. In this study, two dimensional incompressible forms of the continuity and momentum equations are converted to discrete form by application of Galerkin Finite Volume algorithm on triangular meshes. The biharmonic artificial dissipation formulation suitable to triangular meshes is applied to guarantee the convergence to the steady state. The use of triangular meshes, provide great flexibility for modeling the flow about complex shaped geometries. The shortcoming of the efficiency associated with the use of triangular meshes is overcome by application of a face-base solution algorithmand, techniques for preserving stability and efficiency of the matrix free explicit solution method are described. Simulation of a convective dominated flow which may give rise to high frequency numerical errors in explicit solution is presented by solution of the potential flow around Rankin body. For this case, initially the convergence behavior of the model is assessed by sensitivity analysis on the parameters of the Artificial Compressibility Technique, CFL number for computational step limit and artificial dissipation coefficient. Then, the accuracy of the computed results are evaluated by comparison with the available exact solution..

Key words: Incompressible flow . SGS turbulence eddy viscosity model . Artificial compressibility method . Artificial dissipation formulation . Triangulate mesh . Galerkin finite volume method

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 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 computations results in the incompressible flow solution [1].

Corresponding Author: Dr. Saeed-Reza Sabbagh-Yazdi, Department of Civil Engineering, K.N. Toosi University of Technology, 1346 Valiasr St. Tehran, Iran

In present study, the Galerkin Finite Volume Method is used to derive the discrete formulas of the governing equations on triangular 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 triangular 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 described Galerkin finite volume algorithm, in which, artificial dissipation for triangular meshes are utilized for stabilizing of the explicit solution of incompressible steady state flow cases is describedand, the accuracy of model is assessed for various flow conditions.

The effects of some parameters such as the Artificial compressibility parameters, multistage time stepping limit and artificial dissipation coefficient which would affect the converge behavior of numerical model are investigated by sensitivity analysis of the parameters in the explicit marching solution of a bench mark inviscid flow test case.

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

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

Where

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

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

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

W represents the conserved variables while, F^c, G^c e, 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 velocityand *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 [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]

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

In order to prevent numerical difficulties in the region of very small velocities (ie, in the vicinity of stagnation pints), the parameter β^2_{min} is considered in the range of 0.1 to 0.3and 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 in developing a method of obtaining steady-state solutions.

NUMERICAL METHOD

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

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

Where, W_i represents conserved variables at the center of control volume Ω_i (Fig. 1).

Here, F^c, G^c are the mean values of convective fluxes at the control volume boundary faces. Superscripts n and $n+1$ shows n^{th} and the $n+1^{th}$ 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 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;

$$C(W_i) = \sum_{k=1}^{N_{edges}}[F^c\Delta y - G^c\Delta x] \quad (3)$$

and

$$D(W_i) = \sum_{k=1}^{Ncell}[F^v\Delta y - G^v\Delta x] \quad (4)$$

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

$$\nabla^4 W_i = \varepsilon_4 \sum_{j=1}^{Ne}\lambda_{ij}(\nabla^2 W_j - \nabla^2 W_i) \quad (5)$$

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*).

$$\nabla^2 W_i = \sum_{j=1}^{Ne}(W_j - W_i) \quad (6)$$

In equation 10, 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]:

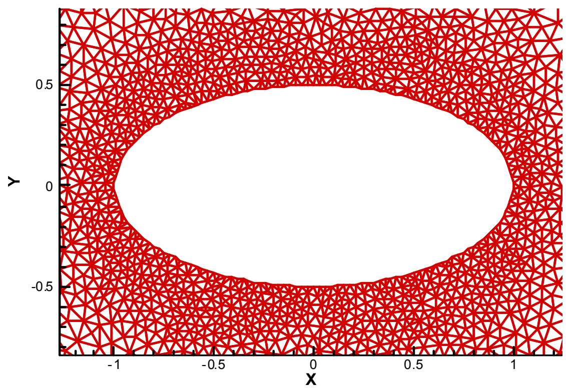
$$\lambda_i = \sum_{k=1}^{Ne}\left|(u.\Delta y - v.\Delta x)_k\right| + \sum_{k=1}^{Ne}\sqrt{(u.\Delta y - v.\Delta x)^2_k + (\Delta x^2 + \Delta y^2)_k} \quad (7)$$

CONVERGENCE BEHAVIOR OF THE MODEL

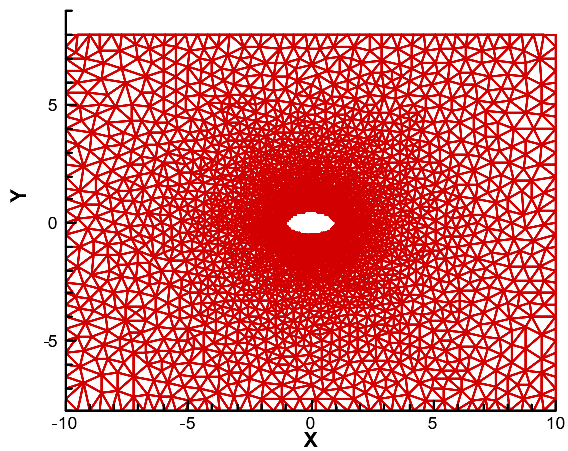
In order to study the parameters affecting the converge behavior of numerical model which is developed based on the Artificial compressibility technique, multistage time stepping and adding artificial dissipation term to stabilize the explicit solution toward the steady state condition a bench mark convection dominated flow test case is considered in this section. The test case is the incompressible inviscid flow facing a Rankin body. The flow field around this object is modeled in a discrete form using 4033 grid points and 7750 unstructured triangular elements (Fig. 1) in which no physical damping (i.e. diffusion due to the fluid viscosity or flow turbulence) excite to damp out the numerical oscillations [9].

Coefficient of artificial compressibility: The effect of coefficient Artificial compressibility (β^2_{min}, C) on converge behavior of the solution procedure are examined. Considering $\varepsilon_4 = 0.0125$ and *CFL*=1.0 for local computational steps, various values of Artificial compressibility coefficient C and the minimum limit of the computed local Artificial compressibility parameter β^2_{min} are examined (Fig. 2 and 3). As can be seen, the values of C = 1 and β^2_{min} = 1 provides better convergence behavior than the other values.

Allowable computational time step limit: The effects of allowable computational time step limit (*CFL* number)



a. Close view



a. General view

Fig. 1: Unstructured triangular mesh for inviscid flow facing a Rankin body

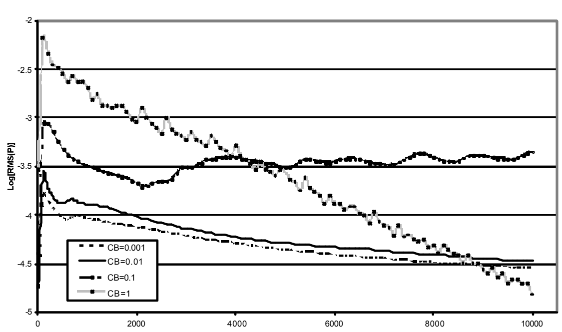


Fig. 2: Effects of C on convergence behavior

which modifies the value of the time step on converge behavior of the solution procedure to the steady state condition, various values of *CFL* number for local computational time step limit are examined by considering $\varepsilon_4 = 0.0125$, $C = 1.0$ and $\beta^2_{min} = 1.0$ (Fig. 4). The plots of convergence history for three CFL numbers of 1.0, 2.0 and 3.0 for the utilized three-stage Runge-

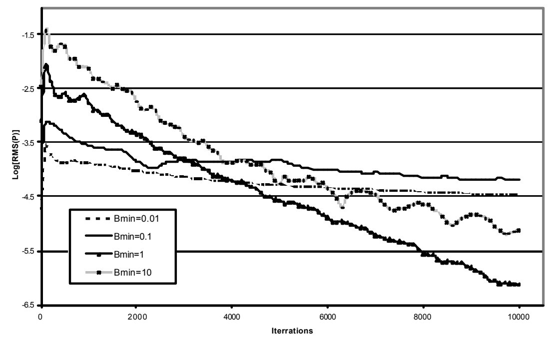


Fig. 3: Effects of β^2_{min} on convergence behavior

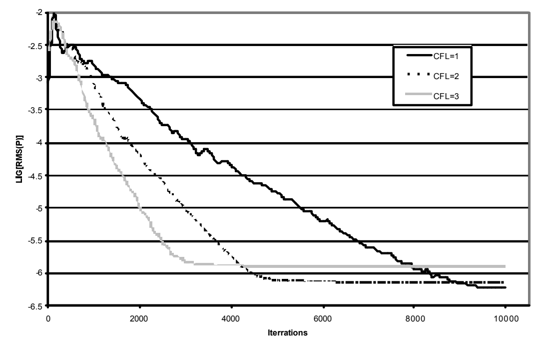


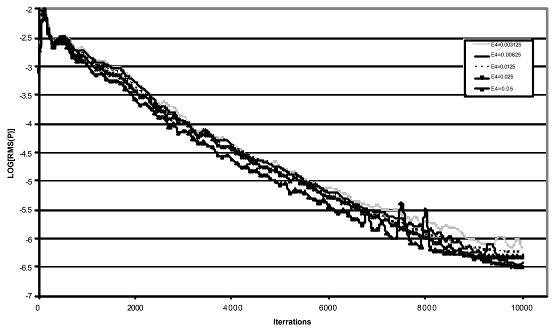
Fig. 4: Effects of *CFL* number on convergence behavior of a three stage Runge-Kutta scheme

Kutta scheme show that can support the CFL number up to 3.0. Although the increase, the allowable time step may provide considerable acceleration in convergence toward steady state, the lower CFL numbers may end up with less convergence error.

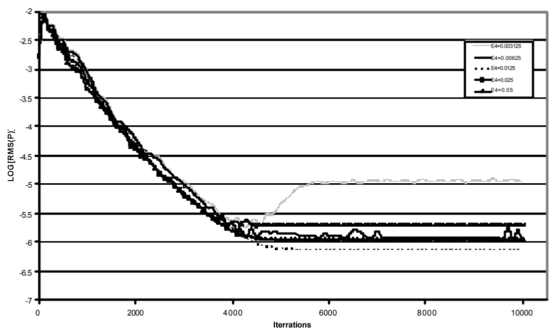
Coefficient of artificial dissipation: The effect of coefficient of artificial dissipation (ε_4) on converge behavior of the solution procedure are examined. Considering various *CFL* number (for local computational steps), $\beta^2_{min} = 1.0$ and $C = 1.0$, various values of artificial dissipation coefficient ε_4 are examined (Fig. 5). For a case of inviscid flow around stream line fitted body such as Rankin body which is modeled by relatively coarse mesh, the large and small values of values of ε_4 may cause some oscillations and limits the reduction of convergence error. However, the values close to $\varepsilon_4 = 0.0125$ provide stable and reasonable convergence behavior.

COMPUTATIONAL RESULTS

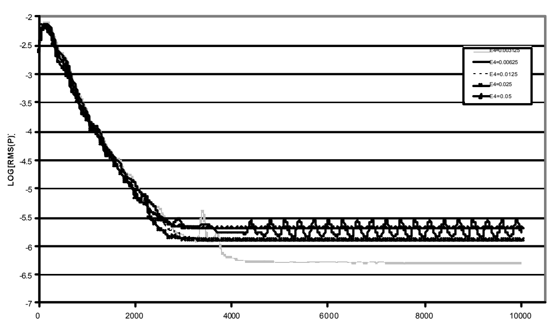
The accuracy of the developed incompressible inviscid flow solver is examined by solving the case



a. CFL = 1



b. CFL = 2



c. CFL = 3

Fig. 5: Effects of ε_4 on convergence behavior

with available analytical solutions which is used in the previous section. The analytical solution is obtained from potential flow theory by using conformal mapping technique [8]. For numerical simulation of the case, unit free stream velocity and pressure is imposed at inflow and outflow boundaries and at the solid wall nodes slipping velocity are considered.

The computations are performed on a triangular mesh containing 12397 grid points, 24316 triangular elements and 36713 faces.

The numerical solution of the case is performed using the Artificial compressibility parameters (β^2_{min}, C), CFL number and artificial dissipation coefficient (ε_4) in optimum range. The computed velocity and pressure components are compared with the exact solution in three sections (Fig. 6). The comparison of these components in section 3 (which is critical section because of dissipation) presents in Fig. 7. This comparison shows the accuracy of the developed model, Percentage of error around Rankin body in comparison with analytical solution in three sections is shown in Table 1.

Table 1: Error of computed parameters for inviscid flow around Rankin body in comparison with analytical solution

Section	Total velocity	Pressure
Section 1	2.80%	2.50%
Section 2	1.10%	1.70%
Section 3	3.20%	2.10%

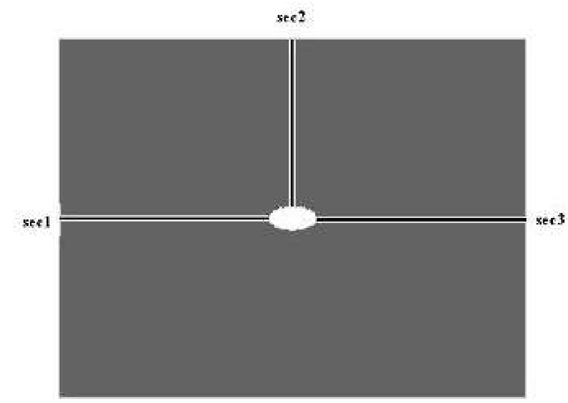
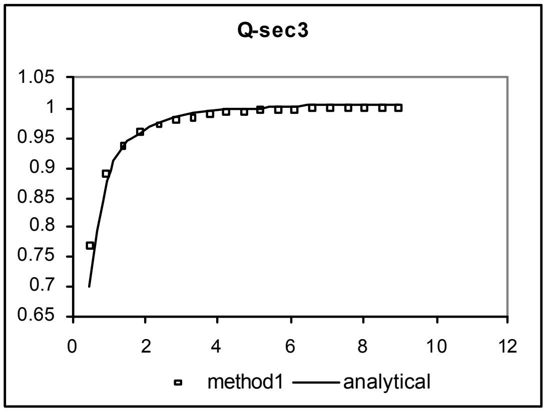


Fig. 6: Sketch and position of sections

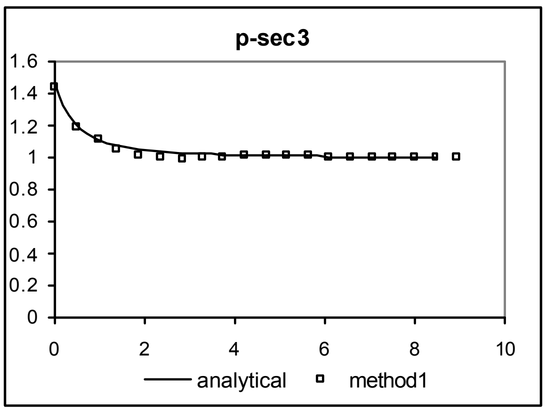
DISCUSSION

In this study, the Galerkin finite volume method is introduced, in which, artificial compressibility technique for incompressible flow solution, multi-stage time stepping for stabilizing explicit solution procedure and increasing allowable CFL numberand artificial dissipation formulation of unstructured triangular meshes for damping out numerical oscillations are utilized.

The sensitivities of the developed Galerkin finite volume method are investigated by solution of a bench mark test of inviscid flow (in which no physical diffusion exists to damp the numerical dissipations). The effects of the artificial compressibility parametersand CFL number of multistage local time stepping limit as well as artificial dissipation coefficient on the converge behavior of numerical model are investigated. By



A: Velocity components at section 3



B: Nodal pressure at section 3

Fig. 7: Comparison of the analytical solution with the computed flow parameters on Rankin body

performing a series of sensitivity analysis of the optimum ranges of the parameters of the model for explicit solution on triangular meshes are obtained.

Finally, the computed results are compared with the analytical solution of the test case. The agreements of the computed with exact solutions encourages for further developments of the model.

REFERENCES

1. Chorin, A., 1967. A Numerical Method for Solving Incompressible Viscous Flow Problems. Journal of Computational Physics, 2: 12-26.
2. Chang, J.L. and D. Kwak, 1984. 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., 1986. Preconditioning Methods for Solving the Incompressible and Low Speed Compressible Equations. ICASE Report, pp: 86-114.
4. Dreyer, J., 1990. Finite Volume Solution to the Steady Incompressible Euler Equation on Unstructured Triangular Meshes. M.Sc. Thesis, MAE Dept., Princeton University.
5. Rizzi, A.and L. Eriksson, 1985. Computation of Inviscid Incompressible Flow with Rotation. Journal of Fluid mechanic, 153: 275-312.
6. Belov, A., L. Martinelli and A. Jameson, 1995. A New Implicit Algorithm with Multi-grid for Unsteady Incompressible Flow Calculations. AIAA 95-0049, 33rd Aerospace Science Meeting and Exhibition 1995 Reno.
7. Yu, D. and A. Kareem, 1996. Two-Dimensional Simulation of Flow around Rectangular Prisms. Journal of Wind Engineering and Industrial Aerodynamics, 62: 131-161.
8. Vallentine, H.R., 1969. Applied Hydrodynamics. Butterworths, S.I. Edition, London.
9. Liggett, J.A., 1994. Fluid Mechanics. McGraw-Hill International Edition, Civil Engineering Series.
10. Fox, R.W. and A.T. Mcdonald, 1994. Introduction to Fluid Mechanics, 4th Edn. John Wiley & Sons, Inc.