INTERNATIONAL JOURNAL FOR NUMERICAL METHODS IN FLUIDS
Int. J. Numer. Meth. Fluids 2005; **47**:1107–1113
Published online 12 January 2005 in Wiley InterScience (www.interscience.wiley.com). DOI: 10.1002/fld.877

Unstructured moving grids for implicit calculation of unsteady compressible viscous flows

A. Jahangirian*,† and M. Hadidoolabi

Aerospace Engineering Department, Amirkabir University of Technology, Tehran, Iran

SUMMARY

An efficient dual-time implicit approach combined with unstructured moving grids is presented to solve unsteady turbulent flows. Unstructured grids suitable for both inviscid and turbulent viscous flow regions are generated using a successive refinement method and the grid is moved adaptively based on the boundary movements. Special care is taken to maintain the quality of the grid near the surface. The unsteady two-dimensional compressible Navier–Stokes equations are discretized by an implicit approach in a real time basis. The resulting set of implicit non-linear equations is then solved iteratively in pseudo-time using a Runge–Kutta scheme. The k–ε turbulence model equations are solved together with the main flow equations in a fully coupled manner. Results are presented for an unsteady transonic test case (AGARD CT5) about a NACA0012 oscillating airfoil and comparisons with experimental data showed good agreements. Copyright © 2005 John Wiley & Sons, Ltd.

KEY WORDS: implicit method; unsteady compressible viscous flow; unstructured moving grids

1. INTRODUCTION

A great variety of flows with significance to aerodynamic applications are concerned with high Reynold number unsteady flows around complex geometries. Efforts in CFD community have, therefore, been towards the obtaining of efficient procedures for moving grid generation and unsteady flow solution algorithms in order to achieve the required accuracy with a reduced computational effort. A number of calculations have been carried out using explicit methods [1]. However, the numerical stability restriction imposed on the maximum allowable time step would increase the computational effort, particularly in viscous calculations, where the ratio of the maximum to the minimum size of the cells can span several orders of magnitude. Implicit methods, on the other hand, allow the use of much larger time steps leading to a significant efficiency for viscous flows. Several implicit methods have been

*Correspondence to: A. Jahangirian, Aerospace Engineering Department, Amirkabir University of Technology, Tehran, Iran.
†E-mail: ajahan@cic.aut.ac.ir

Received 12 May 2004
Revised 4 October 2004
Accepted 5 November 2004

developed; however, the implicit dual-time method described by Jameson [2] allows an implicit discretization to be used in real time, while at each real time step marches the solution in a pseudo-time through an explicit time marching scheme. This method was successfully implemented for rigid [2] and moving boundaries [3] on structured grids to solve Euler equations. However, the geometric complexity involved in aerodynamic applications often require more flexible, unstructured grids to be used. In addition, despite inviscid models yielding a cost effective approximation to the solution of unsteady problems, when strong shocks and separated flows are involved, it is necessary to incorporate viscosity and turbulent effects into the model. Alonso *et al.* [4] used a zero-equation Baldwin–Lomax turbulence model while Badcock *et al.* [5] have chosen a k–ε two equation turbulence closure. Chassaing *et al.* [6] utilized the more sophisticated Reynolds stress model. Despite promising results, all these works have been carried out on structured grids.

Using unstructured grids for accurate and efficient calculation of turbulent viscous flows were just recently reported [7] and mostly for steady-state flow calculations. Thus, the objective of the present work is to develop an implicit unstructured moving grid approach which can be applied to unsteady turbulent Navier–Stokes calculations.

2. UNSTRUCTURED MOVING GRID GENERATION

Unstructured grids suitable for both inviscid and viscous flow regions are generated based on a combination of grid enrichment procedures, whereby new grid points and point connectivities are created simultaneously [7]. The method does not require an initial distribution of grid points within the flow domain. Also, unlike the majority of the existing methods, which start with a well-refined distribution of the geometry, the present approach adopts a very crude initial discretization of the geometry and the outer boundary. Surface and field grids are then generated simultaneously as the cell subdivision process continues. More details about this method can be found in References [7, 8] and they will not be mentioned here. The grid is then moved adaptively based on the boundary movements using a modified spring analogy

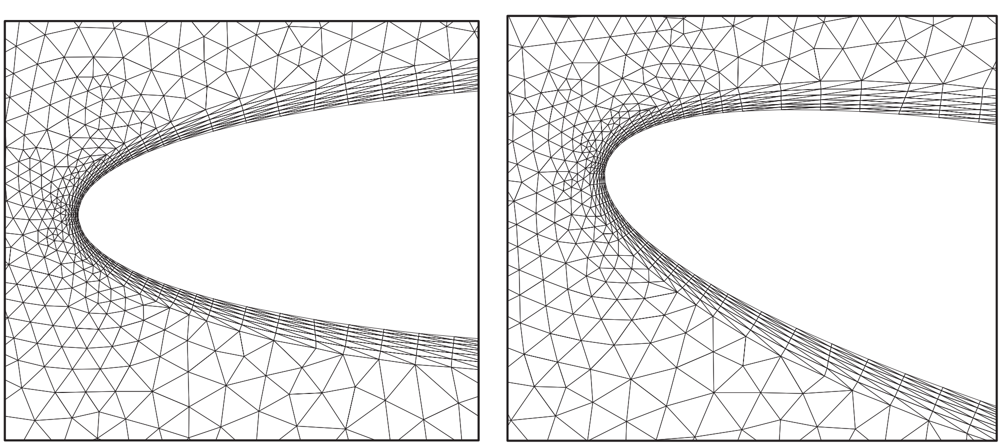

Figure 1. Grid movement strategy for viscous grids (left before and right after movement).

approach. In this approach, the outer boundary nodes are held fixed while the instantaneous location of points on the moving surface are specified. The position of inner nodes are then defined by solving the static equilibrium equations that result from summing the forces in the spring system at each node. Special care is also taken to maintain the quality of the grid near the surface by increasing the stiffness of the near wall springs. This approach also allows geometric deflections required by large CFL numbers of the implicit solution methods. Figure 1 shows the unstructured viscous grid generated around NACA 0012 airfoil before and after movement. Here airfoil reaches pitch angle 15° by 200 interval steps. As illustrated grid quality inside the viscous layer is preserved during the mesh movement process.

3. NUMERICAL FLOW SOLUTION ALGORITHM

Two-dimensional unsteady compressible flow is governed by Reynolds-averaged N.S. equations

$$\frac{\partial Q}{\partial t} + \frac{\partial (F^{\mathrm{i}} - F^{\mathrm{v}})}{\partial x} + \frac{\partial (G^{\mathrm{i}} - G^{\mathrm{v}})}{\partial y} = 0 \tag{1}$$

where $Q = (\rho, \rho u, \rho v, \rho E)$ is the vector of conserved variables, F^{i} and G^{i} represent the convective fluxes and F^{v} and G^{v} describe the effect of viscous diffusion. Here ρ, u, v and E denote the density, Cartesian velocity components and total energy, respectively.

Turbulence effects can be taken into account by using a suitable turbulence model. In this work the standard two-equation k–ε model is used [9]. The turbulent transport equations can be written in a form similar to that used for the mean-flow equations

$$\frac{\partial Q_{\mathrm{t}}}{\partial t} + \frac{\partial (F_{\mathrm{t}}^{\mathrm{i}} - F_{\mathrm{t}}^{\mathrm{v}})}{\partial x} + \frac{\partial (G_{\mathrm{t}}^{\mathrm{i}} - G_{\mathrm{t}}^{\mathrm{v}})}{\partial y} = S_{\mathrm{t}} \tag{2}$$

where $Q_{\mathrm{t}} = (\rho k, \rho \varepsilon)$, $F_{\mathrm{t}}^{\mathrm{i}}$ and $G_{\mathrm{t}}^{\mathrm{i}}$ represent the turbulent convective fluxes and $F_{\mathrm{t}}^{\mathrm{v}}$ and $G_{\mathrm{t}}^{\mathrm{v}}$ describe the effect of turbulent viscous diffusion. The source term S_{t} describes production and dissipation of turbulence quantities. Note that in the present work turbulence Equation (2) is solved together with main flow equations in a fully-coupled manner. The semi-discrete form of Equation (1) can be written as

$$\frac{\mathrm{d}}{\mathrm{d}t}(Q_i A_i) + R_i(Q) - D_i(Q) = 0 \tag{3}$$

where A_i is the area of the cell and $R_i(Q)$ includes the convective and viscous fluxes. The artificial dissipation fluxes $D_i(Q)$ are also added for numerical stability reasons. The properties over each cell edge are evaluated using an averaging method. More details about the development of this approach can be found in Reference [7]. The $\mathrm{d}/\mathrm{d}t$ operator can be approximated by an implicit backward difference formula of kth order of accuracy in the form: $\frac{\mathrm{d}}{\mathrm{d}t} = \frac{1}{\Delta t}\sum_{q=1}^{k}\frac{1}{q}(\Delta^-)^q$, where $\Delta^- Q_i^{n+1} = Q_i^{n+1} - Q_i^n$. Using a second-order accurate time discretization, Equation (3) can be written in a fully implicit form (in real time) as

$$\frac{3}{2\Delta t}(A_i^{n+1}Q_i^{n+1}) - \frac{2}{\Delta t}(A_i^n Q_i^n) + \frac{1}{2\Delta t}(A_i^{n-1}Q_i^{n-1}) + R_i(Q^{n+1}) - D_i(Q^{n+1}) = 0 \tag{4}$$

This equation is nonlinear for Q^{n+1} and therefore cannot be solved analytically. At this stage, it is convenient to redefine a new residual R^*, referred to as unsteady residual, that equals to the left-hand side of Equation (4). The new equation can be seen as the solution of a steady-state problem which can then be solved with a time marching method by introducing a derivative with respect to a fictitious pseudo-time

$$A_i^{n+1}\frac{\partial Q_i^{n+1}}{\partial \tau} + R_i^*(Q^{n+1}) = 0 \tag{5}$$

The pseudo-time problem can then be solved by using any time-marching method designed to solve steady-state problems, utilizing standard acceleration techniques. In the present work an explicit four-stage Runge–Kutta method with local pseudo-time stepping and residual smoothing is used. Geometric conservation laws are also applied to calculate the areas in each time step in order to ensure that large errors are not encountered when solving the physical conservation laws. The no-slip boundary condition is used at the surface boundaries. Non-reflecting boundary conditions based on characteristic analysis are applied in the far field. The wall function conditions are also considered for near wall turbulent calculations.

4. RESULTS

A high Reynolds number unsteady transonic flow over NACA0012 pitching airfoil is considered from the AGARD experimental test cases [10]. For this case, the periodic motion of the airfoil is defined by the angle of attack as a function of time as $\alpha(t) = \alpha_m \alpha_0 \sin(\omega t)$, where α_m is the mean incidence, α_0 is the amplitude of the pitching oscillation and ω is the angular frequency of the motion which is related to the reduced frequency k by $k = \omega c/2U_\infty$ (here, c is the airfoil chord and U_∞ is the free-stream velocity). The airfoil oscillates about its quarter chord. The selected case is AGARD test case CT5, with the conditions: $M_\infty = 0.755$, $\alpha_m = 0.016$, $\alpha_0 = 2.51$, $k = 0.0814$. This is a challenging test case due to a higher value of the Mach number which may make the viscous effects more significant. The flow is characterized by the presence of a strong shock wave, which develops alternatively on the upper and lower surface of the airfoil. A comparison between the viscous and inviscid predicted pressure distributions by the experimental data is shown in Figure 2 for four different incidences during the cycle. An over-prediction of the pressure jump across the shock is observed for inviscid solutions; however, the viscous results show smeared more accurate shock predictions. Three unstructured grids are used for grid study calculations. The number of cells for coarse, medium and fine grids are 8829, 13 487 and 15 304, respectively. The comparison of the normal force and moment coefficients with the experimental data is shown in Figure 3 for these three grids. As illustrated, no significant difference can be seen between medium and fine grids. Therefore, the medium grid is chosen for calculations in this paper with the unsteady CFL number 27 000 for 30 real time steps per period. Another important issue in the dual-time method is to do a proper selection of pseudo-time, steady-state error. Numerical experiments showed that this parameter has an effect on the accuracy of the solutions and the computational time. If large values are used for this parameter, as shown in Figure 4, the accuracy of the solution becomes very poor, whereas the use of small values increase the computational time considerably. The case studied in the present work has shown that the optimum value for pseudo-time, steady-state error has been the same order as 10^{-3}.

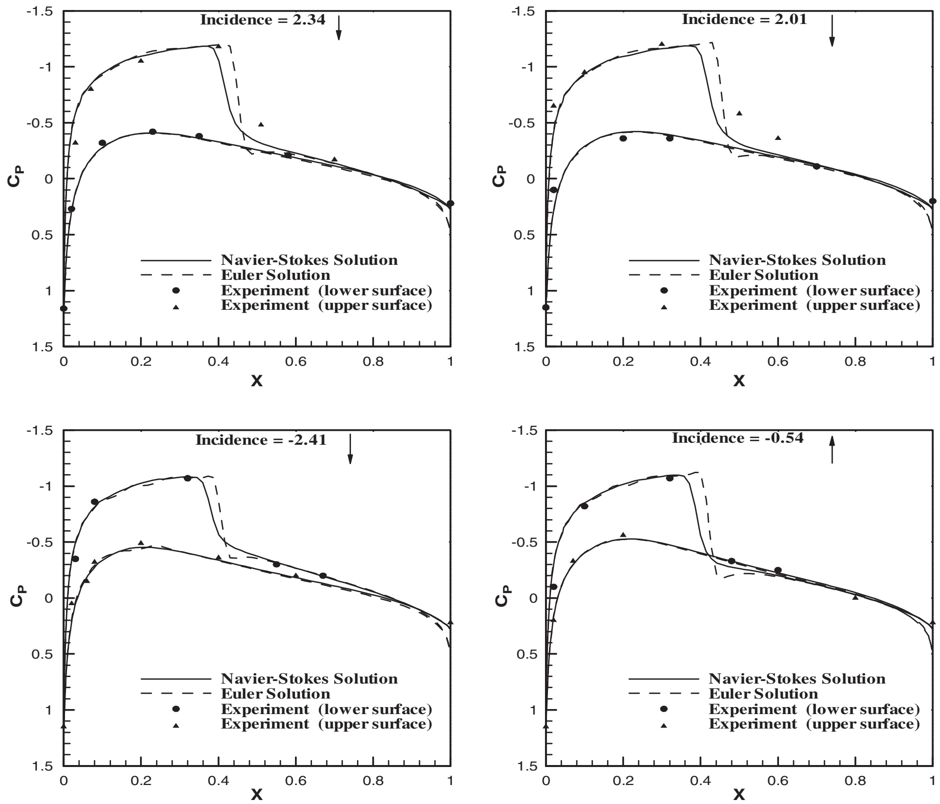

Figure 2. Instantaneous pressure distributions for AGARD test case CT5.

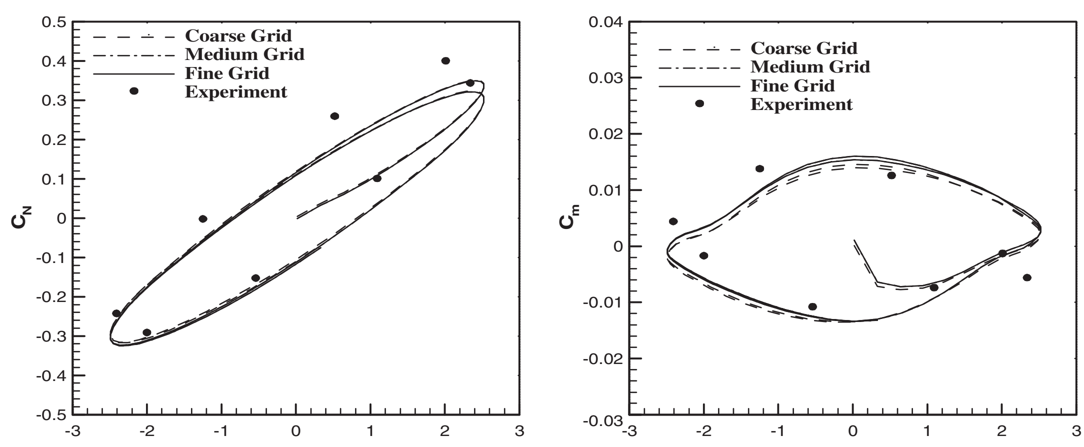

Figure 3. Normal force and moment coefficient loops for AGARD test case CT5.

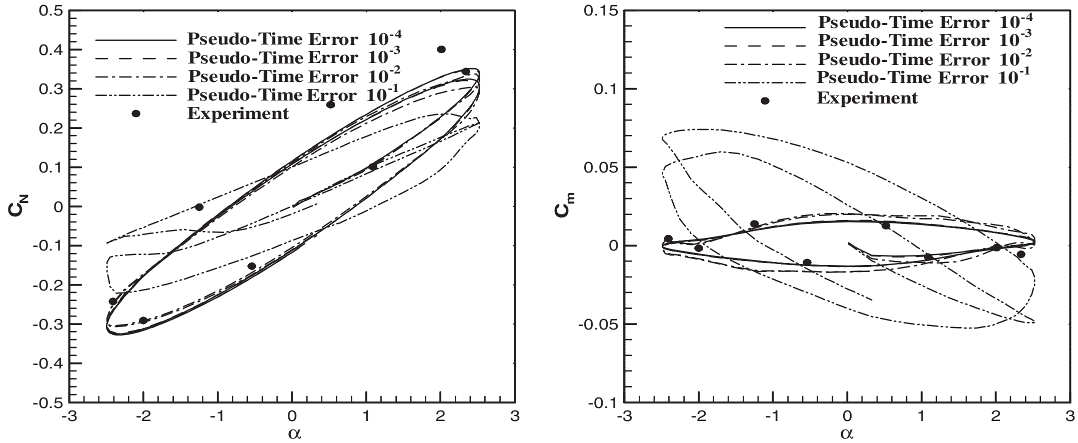

Figure 4. Effect of pseudo-time error on the accuracy of solution.

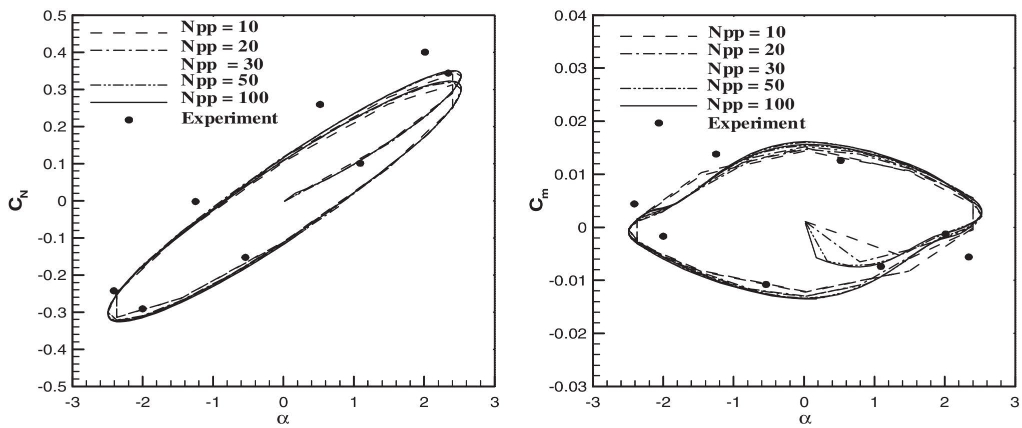

Figure 5. Effect of Npp on the accuracy of solution.

The effect of number of real time steps per period (Npp) on the accuracy of the solution is shown in Figure 5. As illustrated, even reasonably low Npp, i.e. 30, can lead to accurate results. The computational time is reduced by 95 percent in comparison with original fully explicit method.

REFERENCES

1. Batina JT. Unsteady Euler algorithm with unstructured dynamic mesh for complex-aircraft aerodynamic analysis. *AIAA Journal* 1991; **29**(3):327–333.
2. Jameson A. Time dependent calculations using multigrid with applications to unsteady flows past airfoils and wings. *AIAA Paper 91-1596*, 1991.

3. Gaitonde AL. A dual-time method for the solution of the unsteady Euler equations. *Aeronautical Journal* 1994; 283–291.
4. Alonso J, Martinelli L, Jameson A. Multigrid unsteady Navier–Stokes calculation with aeroelastic application. *AIAA Paper 95-004*.
5. Badcock KJ, Cantariti F, Hawkins I, Gribben B, Dubuc L, Richards BE. Simulation of unsteady turbulent flows using the pseudo time method. *Aerospace Engineering Department Report 9721*, Glasgow University, 1997.
6. Chassaing JC, Gerolymos GA, Vallet I. Reynolds-stress model dual-time-stepping computation of unsteady 3-D flows. *AIAA Journal* 2003; **41**(10):1882–1894.
7. Jahangirian A, Johnston LJ. Automatic generation of adaptive unstructured grids for viscous flow applications. *5th International Conference on Numerical Grid Generation in CFD*, Mississippi State University, 1996.
8. Jahangirian A, Johnston LJ. Unstructured grid generation and flow adaptation for external aerodynamic flows. In *Numerical Methods for Fluid Dynamics V*, Morton K, Baines M (eds). Oxford University Press: Oxford, 1995.
9. Launder BE, Spalding DB. The numerical computation of turbulent flow. *Computer Methods in Applied Mechanics and Engineering* 1974; **3**:269–289.
10. AGARD. *Compendium of Unsteady Aerodynamic Measurements*, *R-702*, 1982.

