Journal of Computational Physics 352 (2018) 190–212

Contents lists available at ScienceDirect

Journal of Computational Physics

www.elsevier.com/locate/jcp

Dynamic mode extrapolation to improve the efficiency of dual time stepping method

Yi Liu, Gang Wang *, Zhengyin Ye

Dept. of Fluid Mechanics, School of Aeronautics, Northwestern Polytechnical University, Xi'an, 710072, China

ARTICLE INFO

Article history:
Received 7 March 2017
Received in revised form 3 September 2017
Accepted 24 September 2017
Available online 28 September 2017

Keywords:
Dual time stepping
Dynamic mode decomposition
Convergence acceleration
Initial condition
Unsteady flow

ABSTRACT

This work proposes a methodology to improve the computational efficiency of unsteady flow simulations with dual time stepping scheme. The methodology is developed on the combination of dynamic mode extrapolation and dual time stepping scheme. It accelerates the convergence speed of the inner iterations by using dynamic mode extrapolation to provide an initial solution for each physical time step. The validation and verification are demonstrated by three cases, including unsteady flow past a stationary circular cylinder at Re = 200, transonic flow over periodic and non-periodic pitching NACA 0012 airfoil and buffeting flow around NASA(SC)-0714 airfoil. For comparison, Lagrange extrapolation initial condition and natural initial condition are also applied. The results confirm that the proposed methodology is very successful in reducing computational time for both incompressible and transonic unsteady flow.

1. Introduction

Unsteady flow exists in many engineering applications including aerodynamic flow with separation, helicopter aerodynamics, acoustics, and turbormachinery problems. Owing to the progress in numerical simulation techniques and evolution of computational performance, computational fluid dynamics (CFD) analysis plays important role in this area. However, the computational cost of unsteady problems, such as the rotor environment or full aircraft simulation, is still very large. For a practical CFD solver the efficiency of the numerical method is critical, especially when simulating unsteady flows involving large number of time steps.

Unsteady flows governed by the time dependent Reynolds-averaged Navier–Stokes (RANS) equations are being reported in the literature with increasing frequency. Both explicit and implicit methods are adopted for unsteady flow simulations. Explicit methods are simple to implement, however, for many of problems the permissible time step for numerical stability is much smaller than that required to achieve reasonable accuracy along with the consequence that an excessively large number of time steps may be required. In comparison, implicit schemes allow arbitrarily large time steps to advance the solutions. A major drawback with low order implicit methods is that numerical dispersion and dissipation are high and the time accuracy is undeniably inferior when compared with classical Runge–Kutta time integration. In 1991, Jameson proposed a dual time stepping technique [1,2], which uses an implicit real-time discretization. At each physical time step, the nonlinear system of discretized equations is solved through a sub-iterative pseudo-time scheme. This approach assisted in developing acceleration methods of steady flow calculation such as local time stepping, diagonalization and multigrid meth-

* Corresponding author.
E-mail addresses: nwpuliuyi@163.com (Y. Liu), wanggang@nwpu.edu.cn (G. Wang), yezy@nwpu.edu.cn (Z. Ye).

https://doi.org/10.1016/j.jcp.2017.09.043
0021-9991/

ods. Earlier application of dual time stepping technique to the unsteady Euler and turbulent Navier–Stokes computations were performed by Rumsey [3] and Venkatakrishnan [4]. The accuracy and efficiency were investigated and demonstrated by many test cases [5–8], including flow around circular cylinder and shock buffeting over bicircular airfoil. Motivated by the success of Jameson and others, the researchers adopted dual time stepping scheme as an effective approach to the computation of unsteady flows. Shen and Zha performed large eddy simulation of flow past circular cylinder using dual time method and a new set of sixth order schemes [9] (in 2010). Luo et al. [10] (in 2012) applied the dual time stepping combined with high-order implicit discontinuous Galerkin method to study a variety of unsteady subsonic viscous flow problems. Badcock [11] adopted the dual time stepping scheme to evaluate the dynamic stability derivatives. The encouraging results obtained by these researchers proved the efficiency and capability of dual time stepping method. In fact, the unsteady flows often display highly complex behavior governed by a wide variety of time-scale ranging from high frequency to low-frequency processes. This demands large number of physical time steps to capture the entire frequency band processes and eventually cause the simulation to become computationally expensive. This scenario is particularly true for three-dimensional simulations. In addition, the amount of subiterations used in pseudo time level also has great influence on the computational cost. The recent study on the stability of dual time stepping presented by Chiew [12] and Gerolymos [13] showed that insufficient inner convergence increases both convective and dissipative error. To satisfy the accuracy requirement, about 2 to 5 orders of inner convergence are required. This demands sufficient subiterations when using dual time stepping scheme resulting in higher computational cost. Two key factors affect the amount of subiterations. The first one is the convergence rate of the algorithm applied. This could be improved by implicit time integration, local time stepping, multigrid and other acceleration techniques. The second is the initial solution. If the initial solution falls within the convergence basin of the solution, the pseudo time iteration is expected to be more effective and the amount of subiterations required are reduced.

The dual time stepping advances the unsteady solutions in each physical time step by using the last solution at previous time step to initialize the current solution. This approach is commonly adopted in many researches [3–5,8,12,13] and some named it as natural initial condition (NIC). A limiting aspect of this approach is that the evolution of unsteady flow is not taken into account. The initial solution is always obtained from the previous solution, thus its computational efficiency is relatively low. This is particularly true when a larger physical time steps is used to simulate the complex flowfield involving strong non-linear behavior such as helicopter aerodynamics and shock buffeting phenomenon. To provide a better initial solution, a new dynamic mode extrapolation initial condition (DMEIC) method is presented. Using this method, the initial solution at each time step can be obtained by Dynamic Mode Extrapolation (DME) from the last several solutions at previous time steps. The DME relies on dynamic mode extracted from multiple sets of data, and it works by expressing the function in terms of time and space. The competitive advantage of DME is that it has both superior accuracy and high robustness. Compared with the general polynomial interpolation method, it takes the nonlinear factors of evolution of flows into account ensuring the superior accuracy of extrapolations. Avoiding the Runge's phenomenon that frequently occurs in the high-order polynomial interpolations and thereby enhancing the robustness. At present, the DMEIC method has been implemented into an in-house hybrid unstructured Navier–Stocks solver-HUNS3D [14] developed in Northwestern Polytechnical University for aerodynamic applications in the field of aeronautics and astronautics. The upcoming section briefly describes the basic numerical aspects including URANS simulation and DMEIC algorithm. The later part of this paper shows the validation of this accelerating convergence method on flow past a stationary circular cylinder at low Reynolds number, transonic flow over periodic/non-periodic pitching NACA 0012 airfoil and buffeting flow around NASA (SC)-0714 airfoil, which is followed by several concluding remarks.

2. Numerical method

2.1. Unsteady RANS solver

In this paper an in-house unsteady RANS solver HUNS3D, employed with the functionality of simulating both steady and unsteady viscous flows, is used as the baseline flow solver. HUNS3D solves the time dependent Navier–Stokes equations for the ideal gas expressed by the conservation of mass, momentum and energy for the compressible fluid without incorporating external forces. The integral form of RANS equations expressed on a bounded domain Ω with boundary $\partial\Omega$ and given below.

$$\frac{\partial}{\partial t}\iiint_{\Omega}\mathbf{Q}\,d\Omega+\iint_{\partial\Omega}\boldsymbol{F}(\mathbf{Q})\cdot\vec{\boldsymbol{n}}ds=\iint_{\partial\Omega}\boldsymbol{G}(\mathbf{Q})\cdot\vec{\boldsymbol{n}}ds \tag{1}$$

where,

$$\mathbf{Q}=\begin{Bmatrix}\rho\\ \rho u\\ \rho v\\ \rho w\\ e_0\end{Bmatrix},\qquad \boldsymbol{F}(\mathbf{Q})\cdot\vec{n}=(\vec{V}\cdot\vec{n})\begin{Bmatrix}\rho\\ \rho u\\ \rho v\\ \rho w\\ e_0+P\end{Bmatrix}+P\begin{Bmatrix}0\\ n_x\\ n_y\\ n_z\\ 0\end{Bmatrix},\qquad \boldsymbol{G}(\mathbf{Q})\cdot\vec{n}=\{n_xG_1,n_yG_2,n_zG_3\}$$