Available online at www.sciencedirect.com

SCIENCE DIRECT®

Comput. Methods Appl. Mech. Engrg. 194 (2005) 3949–3983

Computer methods in applied mechanics and engineering

www.elsevier.com/locate/cma

Parallel-multigrid computation of unsteady incompressible viscous flows using a matrix-free implicit method and high-resolution characteristics-based scheme

C.H. Tai [a,b], Y. Zhao [b], K.M. Liew [a,b,*]

[a] *Nanyang Centre for Supercomputing and Visualisation, Nanyang Technological University, 50 Nanyang Avenue, Singapore 639798, Singapore*

[b] *School of Mechanical and Production Engineering, Nanyang Technological University, 50 Nanyang Avenue, Singapore 639798, Singapore*

Received 17 February 2004; received in revised form 19 July 2004; accepted 16 September 2004

Abstract

A three-dimensional parallel unstructured non-nested multigrid solver for solutions of unsteady incompressible viscous flow is developed and validated. The finite-volume Navier–Stokes solver is based on the artificial compressibility approach with a high-resolution method of characteristics-based scheme for handling convection terms. The unsteady flow is calculated with a matrix-free implicit dual time stepping scheme. The parallelization of the multigrid solver is achieved by multigrid domain decomposition approach (MG-DD), using single program multiple data (SPMD) and multiple instruction multiple data (MIMD) programming paradigm. There are two parallelization strategies proposed in this work, first strategy is a one-level parallelization strategy using geometric domain decomposition technique alone, second strategy is a two-level parallelization strategy that consists of a hybrid of both geometric domain decomposition and data decomposition techniques. Message-passing interface (MPI) and OpenMP standard are used to communicate data between processors and decompose loop iterations arrays, respectively. The parallel-multigrid code is used to simulate both steady and unsteady incompressible viscous flows over a circular cylinder and a lid-driven cavity flow. A maximum speedup of 22.5 could be achieved on 32 processors, for instance, the lid-driven cavity flow of $Re = 1000$. The results obtained agree well with numerical solutions obtained by other researchers as well as experimental measurements. A detailed study of the time step size and number of pseudo-sub-iterations per time step required for simulating unsteady flow are presented in this paper.

* Corresponding author. Address: School of Mechanical and Production Engineering, Nanyang Technological University, 50 Nanyang Avenue, Singapore 639798, Singapore. Tel.: +65 6790 4076; fax: +65 6793 6763.
E-mail address: mkmliew@ntu.edu.sg (K.M. Liew).

0045-7825/$ - see front matter
doi:10.1016/j.cma.2004.09.010

Keywords: Parallel-multigrid; Matrix-free implicit method; Unstructured tetrahedral grid; High-order scheme; Unsteady incompressible viscous flow

1. Introduction

With the advent of supercomputers and fast numerical methods in these recent years, researchers in the field of computational fluid dynamics (CFD) handle more complicated problems than before. The computing speed and memory capacity of computers have increased exponentially during this period, thus this has led to the work on parallel computing. Algorithms and strategies that successfully map *structured grid* codes onto parallel machines have been developed over the previous decade and become quite established. Extension of the capabilities of these structured grid codes to include unstructured grid codes requires new algorithms and strategies to be developed.

In this work, the artificial compressibility method (ACM) is adopted for the solution of the incompressible Navier–Stokes equations. The advantage of using this method is that it directly couples the pressure and velocity fields at the same time level, thus producing a hyperbolic system of equations. The artificial waves act as a mechanism to propagate the information throughout the domain, and driving the velocity toward the divergence-free condition. The main disadvantage of introducing artificial compressibility into the system of incompressible Navier–Stokes equations is that it is not suitable for time-dependent solutions and as a result, only steady state solutions can be obtained. However, this problem has been solved by introducing the dual-time stepping scheme [1–6]. The time-dependent solution is viewed as the solution of a steady-state problem at each time step and the pseudo-time step is usually set to a large value, which leads to fast convergence. In this paper, a matrix-free implicit dual time stepping scheme [2] is employed to obtain the numerical solutions for the parallel-multigrid method. This method is found to be stable and efficient because the time-step size is not limited by the stability criteria and no matrix manipulation is required. A detailed study of the time step size and number of pseudo-sub-iterations per time step required for simulating unsteady flow are presented in this paper.

Farmer et al. [34] have developed a fast multigrid method for solving the Euler equations in conjuction with the ACM and applied it to free surface flows. They reported that 400 W-multigrid cycles were required to achieve convergence for the inviscid flow around a ship hull with free surface effects. Sheng et al. [35] have developed a multigrid algorithm for 3D incompressible turbulent flows in conjunction with the ACM and Newton relaxation methods. They proposed two different methods for constructing the coarse grid operator and investigated the influence of implicit correction smoothing on increasing the stability of the scheme. They reported fast convergence rates for the case of external flows, but deteriorate in the case of complex internal flows. Drikakis et al. [36] have implemented a non-linear multigrid method in conjunction with ACM and with a third-order upwind characteristics-based scheme, and the fourth-order Runge–Kutta scheme. The multigrid method is based on the full multigrid (FMG)–full approximation storage (FAS) method. Based on the above-mentioned implementation, they further developed and investigated an adaptive-smoothing (AS) procedure to exploit the non-uniform convergence behavior of the numerical solution during the iterations to reduce the size of the computational domain [37]. Tai and Zhao [41] have presented a two-dimensional unstructured non-nested MG method for efficient simulation of unsteady incompressible Navier–Stokes flows. The Navier–Stokes solver is based on the ACM and a higher-order characteristics-based finite-volume scheme on unstructured grids. Unsteady flow is calculated with a matrix-free implicit dual time stepping scheme. Iwamura et al. [38] have presented a robust and efficient algebraic multigrid (AMG) preconditioned conjugate gradient solver for systems of linear equations arising from the finite element discretization of a scalar elliptic partial differential equation of second order on unstructured tetrahedral meshes. Mavriplis [7] have developed a parallel unstructured agglomeration multi-