International Journal of Offshore and Polar Engineering (ISSN 1053-5381)

Vol. 24, No. 2, June 2014, pp. 106–113

http://www.isope.org/publications

Parallel Finite Element Method for 3D Lid-driven Cubic Cavity Flows

Jifei Wang and Decheng Wan*
State Key Laboratory of Ocean Engineering, School of Naval Architecture, Ocean and Civil Engineering
Shanghai Jiao Tong University, Shanghai, China

A parallel finite element method coupling the fractional step finite element method with the domain decomposition method is applied for parallel simulation of 3D lid-driven cubic cavity flows, based on the open source codes PETScFEM. The Reynolds numbers (Re) between 1 and 10,000 are considered, covering laminar and partly turbulent field. Primary eddy, secondary eddies, corner eddies, Taylor-Gortler-like (TGL) vortices, and other cavity flow features are investigated. At high Reynolds numbers, the mean and mean-root-square velocities statistics along the horizontal and vertical centerlines in the symmetry plane, respectively, maintain reasonable agreement with experimental data. Parallel performance is also analyzed.

INTRODUCTION

The lid-driven cavity flow mimics many flows in nature and plays a role in naval and offshore engineering. For oil transport in line-pipes, a large cubic cavity installed at the specified sites can reduce or absorb pump pressure and flow pulsation. In the nautical navigation field, the flow in the cavity or nozzle at the submarine surface is one of the noise sources, which may reveal the trace of the submarine. Lid-driven cavity flows are not only technologically important, but also are of great scientific interest. These flows display many kinds of fluid mechanical phenomena, including corner eddies, Taylor-Gortler-like (TGL) vortices, transition, turbulence, etc. (Deshpande and Milton, 1998; Ryu and Baik, 2009). Simple geometrical settings and easily posed boundary conditions have made cavity flows popular test cases (Chiang et al., 1996, 1998; Chiang and Sheu, 1997; Feldman and Gelfgat, 2010).

As a classic benchmark, 2D lid-driven cavity flows have been extensively studied with numerical methods. However, the pioneering experimental work of Koseff and Street (1984) clearly showed that cavity flows were inherently 3D in nature. With the increase in computing capability in recent years, 3D lid-driven cavity problems have matured as a standard Re-dependent benchmark. Jiang et al. (1994), Wong and Baker (2002), and many other researchers have investigated the lid-driven cubic cavity at low and moderate Reynolds numbers. Prasad and Koseff (1989) have studied 3D lid-driven cavity flows at Re = 3200, 5000, 7500, and 10000 with experimental methods, which improves the research for these problems at moderate and high Reynolds numbers with numerical methods. Zang et al. (1993) have applied finite volume method in a lid-driven cavity at Reynolds numbers of 3200, 7500, and 10000, showing agreement with the experimental data. Large eddy simulations (LES) have enjoyed popularity for turbulent flows. Bouffanais et al. (2006), Bouffanais and Devilleb (2007), and Shetty et al. (2010) have analyzed the lid-driven cubic cavity at high Reynolds numbers by LES. Direct numerical simulation (DNS) is also popular and can be used for both laminar and turbulent flows (Leriche and Gavrilakis, 2000). Hachem et al. (2010), Verstappen and Veldman (1997), and Leriche (2006) have simulated cubic cavity flows at moderate and high Reynolds numbers by DNS. Bessonov et al. (1995) carried out parallelization of the solution of 3D Navier-Stokes equations for fluid flows in a cavity with moving covers. As reviewed in Shankar and Deshpande (2000), the flow fields of the lid-driven cubic cavity are laminar when Re < 6000; transition to turbulence takes place in the range 6000 < Re < 8000, and sufficient partition of the fields are turbulent by Re = 10000. TGL vortices can be observed in both unsteady laminar and turbulent flows.

It is difficult to obtain the solution of incompressible Navier-Stokes (NS) equations using classical finite element method. The mathematical analysis of the Stokes problem shows that the approximation spaces for velocity and pressure must satisfy a compatibility condition known as the *inf-sup* LBB condition, proposed by Ladyzhenskaya (1969), Babuska (1973), and Brezzi (1974). This has the drawback where only some combination of interpolation spaces for velocity and pressure can be used. However, the fractional step method, based on the Poisson projection, can be used with spatial interpolations that do not satisfy the LBB condition. These methods are applied widely because of the computational efficiency. Guermond and Quartapelle (1998) investigated the stability and convergence of fractional step method with equal order interpolations. It was shown that there is a lower bound for the time step for stability reasons. Codina (2001) got similar results and presented a stabilized fractional step finite element method. These results are used in this work.

Based on the open source codes PETScFEM, a parallel finite element method coupling the fractional step finite element method (FEM) with the domain decomposition technique is applied for parallel simulation of 3D lid-driven cubic cavity flows. The numerical method is briefly introduced as follows: In the preprocessing, the computational domain is discretized by the regular mesh with the brick elements. The fractional step method is applied to decouple the incompressible Navier-Stokes system in three substeps. All three subequations are discretized by finite element method with the equal order interpolation of the velocity and pressure in space. For the parallel computation, the whole mesh is decomposed to several non-overlapping subdomains (Tai et al., 2005; Paz et al., 2006). All of the subdomains are computed at the same time. The information of the interface among the subdomains is passed among the processors by MPI (Message Passing Interface) (Sonzogni et al., 2002). All of the linear systems are solved by GMRES (Generalized Minimal RESidual) method with Jacobi preconditioner, which is carried out in PETSc (Portable, Extensible Toolkit for Scientific

*ISOPE Member.

Received July 29, 2011; updated and further revised manuscript received by the editors December 14, 2013. The original version (prior to the final revised manuscript) was presented at the Twenty-first International Offshore and Polar Engineering Conference (ISOPE-2011), Maui, Hawaii, June 19–24, 2011.

KEY WORDS: PETScFEM, 3D cavity flows, domain decomposition, parallel computation, TGL vortices, high Reynolds number.