Alexandria Engineering Journal (2016) xxx, xxx–xxx

HOSTED BY

ELSEVIER

Alexandria University

Alexandria Engineering Journal

www.elsevier.com/locate/aej
www.sciencedirect.com

ORIGINAL ARTICLE

Revisiting the lid-driven cavity flow problem: Review and new steady state benchmarking results using GPU accelerated code

Tamer A. AbdelMigid [a], Khalid M. Saqr [b], Mohamed A. Kotb [a,*], Ahmed A. Aboelfarag [c]

[a] Department of Marine Engineering, College of Engineering and Technology, Arab Academy for Science, Technology and Maritime Transport, Alexandria, Egypt
[b] Department of Mechanical Engineering, College of Engineering and Technology, Arab Academy for Science, Technology and Maritime Transport, Alexandria, Egypt
[c] Department of Computer Engineering, College of Engineering and Technology, Arab Academy for Science, Technology and Maritime Transport, Alexandria, Egypt

Received 21 May 2016; revised 10 September 2016; accepted 20 September 2016

KEYWORDS

Lid-driven cavity flow;
GPU accelerated;
Numerical method;
CFD;
Finite volume method

Abstract This paper presents a broad account of the lid-driven cavity flow problem which is an important benchmark problem for the validation of CFD codes. A comprehensive review of the literature on the problem is presented and discussed, and available benchmarking results are compared in tabulated format to provide a comprehensive source of validation data. In addition, the problem was solved using a Graphical Processing Unit (GPU) accelerated in-house code developed by the authors (https://github.com/TamerAbdelmigid/DrivenCavity_FVM.git), which solves the steady Navier-Stokes equations, using the Finite Volume Method (FVM) in primitive variable formulation. Case studies of steady incompressible flow in a 2D lid-driven square cavity are investigated for $100 < Re < 5000$. Detailed second order spatially accurate results are verified and presented in a tabulated form for the sake of serving as benchmark dataset for future works on the same problem. In the present work, collocated grid arrangement along with a uniform structured Cartesian grid up to 1301×1301 was used.

1. Introduction

The driven cavity flow for over the past half a century served as a benchmarking case for anyone to validate their techniques and methods against, and over this period it has been studied by hundreds of authors with nearly every numerical method that exists, and yet only a handful of accurate and complete

* Corresponding author.
E-mail address: kotb2000@yahoo.com (M.A. Kotb).

Peer review under responsibility of Faculty of Engineering, Alexandria University.

http://dx.doi.org/10.1016/j.aej.2016.09.013
1110-0168 © 2016 Faculty of Engineering, Alexandria University. Production and hosting by Elsevier B.V.

benchmark results are available in the literature. In the present paper, all available data on the problem have been reviewed and discussed in details. In addition, a GPU accelerated finite volume code has been developed and utilized to produce accurate benchmarking results using grids of resolution up to 1301×1301 cells. The purpose of the existing review was to make some of the most important work done on the steady state square driven cavity flow in the past 50 years available in one source. However, it is worth mentioning that the presented review is by no means intended to be comprehensive.

The discussion proposed in the present paper concentrates on the discretization technique, spatial accuracy, grid, and Reynolds number range considered in the literature.

Though all the methods have been used to study the driven cavity case, the Finite Difference method is by far the most used one, from as old as Burggraf [1] to as new as Kalita and Gupta [2]. Most of the authors formulated the governing equations in stream function-vorticity variables, most famous of which are Ghia et al. [3] who used coupled strongly implicit multigrid (CSI-MG) based on the work of Rubin and Khosla [4] method in the solution of the driven flow in a square cavity, for Reynolds number $R_e \leqslant 10,000$. They used a uniform mesh of 257 * 257. They presented a Second-order accurate tabulated benchmark results that have served as "The" result to compare against ever since. Recently, Erturk et al. [5] using a fine uniform grid mesh of 601×601, computed a steady solution for the driven cavity flow for Reynolds number $R_e \leqslant 21,000$ with maximum absolute residuals of the governing equations that were less than 10^{-10}, although their solution was second order spatially accurate, but they provided a six order accurate solution for some variables using Richardson extrapolation.

On the other hand other authors formulated their equations using only the stream function as a variable. Of those we mention Schreiber and Keller [6] who presented fourth-order spatially accurate results for Reynolds number $R_e \leqslant 10,000$. Their numerical methods combined an efficient linear system solver, an adaptive Newton-like method for nonlinear systems, and a continuation procedure for following a branch of solutions over a range of Reynolds numbers, on a 180×180 uniform grid. Also, Poochinapan [7] obtained a solution up to Reynolds number $Re = 5000$ with second-order spatial accuracy on a 122×122 grid.

However, some authors used primitive variables like Vanka [8] who presented a second order accurate solutions for steady flows up to Reynolds Number $R_e = 5000$. He used a uniform grid of 321×321. Bruneau and Saad [9] obtained a steady and periodic solutions for various Reynolds numbers by solving the unsteady Navier-Stokes equations on a 1024×1024 uniform staggered grid. Their numerical simulation lies on a multigrid solver with a cell-by-cell relaxation procedure. Classical Euler or Gear time schemes are coupled to a second-order approximation of the linear terms in space. Convective terms were treated explicitly and approximated by third-order schemes.

Standing apart from them Gupta and Kalita [10], they used stream function-velocity formulation to obtain a second order accurate solution for Reynolds Number $100 \leqslant R_e \leqslant 10,000$. Their computation was done on uniform 161×161 grid. They used a biconjugate gradient method to obtain the numerical solutions of the aforementioned fluid flow problem.

Second in popularity was the Finite volume method in primitive variable formulation as an example, Wright and Gaskell [11] presented a second and fourth order spatially accurate steady solution for Reynolds number $100 \leqslant R_e \leqslant 1000$ using the staggered grid arrangement and control volume formulation. They used the Block Implicit Multigrid Method (BIMM) on a very fine uniform mesh of 1024. Similarly, Magalhães et al. [12] presented a second order spatially and temporally accurate solution for Reynolds Number $R_e = 100,\ 400,\ \textit{and}\ 1000$ on a non-uniform mesh of 51×51.

Finite element method takes the third place where Olson and Tuann [13] recasted the full Navier-Stokes equations in the form of a single, fourth order equation for stream function, with an 18 degrees-of-freedom triangular element, such that the velocities were continuous and the incompressibility was satisfied exactly. They covered Reynolds Number from 10^{-4} to $R_e = 3450$, with a uniform mesh of 8×8, and produced a remarkably accurate results for such coarse mesh. Likewise, Barragy and Carey [14] used finite element for the solution of the lid-driven cavity flow up to Reynolds number $R_e \leqslant 12,500$. They used a graded mesh of elements of degree $p = 8$, and they also incorporated an under resolved solution for $R_e = 16,000$.

After that comes several other methods such as Lattice Boltzmann which have been used by Hou et al. [15] with compressibility effects, for the solution of the driven cavity flow for Reynolds number $R_e \leqslant 7500$ using a 256×256 grid points. Similarly, Lin et al. [16] used the multi relaxation time (MRT) lattice Boltzmann equation (LBE) with D2Q9 model to compute a steady solution at different Reynolds numbers (100–7500), using a 129×129 grid.

Boundary Element has been used by Grigoriev and Dargush [17] for Reynolds number $R_e \leqslant 5000$. They carried out the simulation on a non-uniform mesh with 1680 hexagonal regions. In addition Aydin and Fenner [18] used it to acquire a solution for low-to-moderate-Reynolds number $0 \leqslant R_e \leqslant 1200$. They used four different mesh sizes (maximum being 81 Boundary elements).

Smooth particle hydrodynamics has been used by Szewc et al. [19] along with three different incompressibility treatments namely WCSPH, weakly compressible smoothed particle hydrodynamics; ISPH, incompressible smoothed particle hydrodynamics; with two variants PPS, particle-based Poisson solver; GPS, grid-based Poisson solver, to obtain a solution fir lid driven cavity at $Re = 1000$, with 57,600 particles. And Khorasanizade and Sousa [20] computed a solution for flow at moderate Reynolds numbers $100 \leqslant R_e \leqslant 3200$, employing the mesh-free (SPH), with a new treatment for no-slip boundary conditions. They carried out their study using different spatial resolutions maximum of which is $L/200$.

Chebyshev collocation method has been used by Botella and Peyret [21] to present a highly-accurate spectral solutions with extensive benchmark results for the flow at Reynolds number $Re = 1000$ using with a maximum of grid mesh of $N = 160$ (polynomial degree).

Incremental unknowns were utilized by Goyon [22] to solve the unsteady 2D Navier-Stokes equations on an un-regularized driven cavity. They presented steady solution for Reynolds number $R_e \leqslant 7500$. For Reynolds number $10,000 \leqslant R_e \leqslant 12,500$ they presented a periodic solution, although they admit that this investigation field is less exploited, because of the computational cost.

Please cite this article in press as: T.A. AbdelMigid et al., Revisiting the lid-driven cavity flow problem: Review and new steady state benchmarking results using GPU accelerated code, Alexandria Eng. J. (2016), http://dx.doi.org/10.1016/j.aej.2016.09.013