Available online at www.sciencedirect.com
SCIENCE DIRECT
ELSEVIER
Applied Mathematics and Computation 176 (2006) 506–515
APPLIED MATHEMATICS AND COMPUTATION
www.elsevier.com/locate/amc

Numerical simulation of cavity flows based on transformed equations

H. Xu [a], C. Zhang [b,*], R. Barron [c]

[a] Department of Mechanical, Automotive and Materials Engineering, University of Windsor, Windsor, Ont., Canada N9B 3P4
[b] Department of Mechanical and Materials Engineering, The University of Western Ontario, 1151 Richmond Street, London, Ont., Canada N6A 5B9
[c] Department of Mathematics and Statistics, University of Windsor, Windsor, Ont., Canada N9B 3P4

Abstract

A new numerical algorithm is applied to simulate two-dimensional lid-driven cavity flows. In this new algorithm, the momentum equations are first transformed using an exponential function to eliminate the convection terms in the equations. Then a central differencing scheme is employed to discretize the transformed equations. The cavity flows studied in this work include those with non-zero velocity component in the *y*-direction on the upper and lower boundaries. The results for the velocity components along the geometric centerline, stream function patterns, and vorticity contours are presented and discussed. The predicted results are in excellent agreement with benchmark solutions.

Keywords: Partial differential equation; Differencing scheme; Fluid flow; Cavity flow; Numerical approach

1. Introduction

The partial differential equations describing fluid flow throughout space and time are, in most cases, nonlinear in nature. Exact solutions only exist for a few specific cases with simple geometries and boundary conditions, or for simplified equations in which some of the more complicated physical phenomena are neglected. As the use of digital computers has become widespread, computational methods are now widely used for complicated nonlinear problems with complex boundary conditions.

In order to solve the partial differential equations numerically, the equations must be discretized. The partial differential equations that govern the physical process of fluid flow usually have four types of terms. These are the transient term, convection term, diffusion term and source term. The transient term can be discretized using a first-order forward or backward differencing scheme. For the source term, no discretization is needed

* Corresponding author.
E-mail address: czhang@eng.uwo.ca (C. Zhang).

0096-3003/$ - see front matter
doi:10.1016/j.amc.2005.09.041

if there are no derivatives in it. If the source term contains either first-order or second-order derivatives, a central differencing scheme can be employed for the interior nodal points. For the nodal points adjacent to the boundary, a backward or forward differencing scheme can be adopted. The diffusion term contains second-order derivatives and the second-order central differencing scheme is the most appropriate scheme to discretize this term. The convection term is nonlinear and involves first-order spatial derivatives. The discretization of the convection term is one of the major difficulties in numerical solutions of the governing equations for fluid flow. The primary difference between various differencing schemes is the method of discretization of the convection terms.

Many research efforts have been made to develop an effective differencing scheme to discretize the convection terms. Typical differencing schemes are the four classical ones, namely, the first-order upwind scheme (FOU), hybrid scheme, power-law scheme, and exponential scheme [1]. Furthermore, more complex higher-order upwind differencing schemes, such as the second-order upwind scheme (SOU) (original idea traced to Price et al. [2]), skew-upwind differencing scheme (SUD) [3], quadratic upstream interpolation for convective kinematics (QUICK) [4] and simple high-accuracy resolution program scheme (SHARP) [5], were developed to discretize the convective terms. In addition to the schemes mentioned above, some researchers proposed quite different schemes adopting different ideas. For examples, Sakai [6] proposed a new finite variable difference method, in which a variable spatial difference instead of the conventional Δx was adopted for the discretization of the convection term. Jasak et al. [7] discussed the issue of boundedness in the discretization of the convection terms in transport equations and developed the total variation diminishing criterion that localizes it. Development of methods to discretize the convective terms has been a major research topic in the academic community.

In the present study, an alternative approach to discretizing the governing equations is used. In this approach, the convection terms in the partial differential equations are first eliminated using a mathematical manipulation, and the resulting equations are discretized using a central differencing scheme. Both the fourth-order central differencing scheme and the second-order central differencing scheme are used to discretize the transformed partial differential equations. This new algorithm has been used to solve the two-dimensional parabolic partial differential equation and elliptic partial differential equation [8,9]. The predicted results using the proposed algorithm agree well with analytic solutions.

The objective of the present work is to apply the new algorithm to lid-driven cavity flows. A lid-driven cavity flow, which is the flow in the cavity-driven by the motion of a boundary surface, is a typical problem that has been extensively studied, both experimentally and numerically. For example, Pan and Acrivos [10] used a photographic technique to determine the flow patterns for finite cavities, as well as for cavities of effectively infinite depth for Reynolds numbers ranging from 20 to 4000. Meanwhile, they obtained a numerical creeping flow solution by solving a biharmonic equation. Chen et al. [11] used the finite analytic method to solve heat transfer in a cavity flow at Reynolds numbers from 100 to 2000 and Prandtl numbers from 0.1 to 10. Ghia et al. [12] utilized the lid-driven cavity flow as a model problem to study the effectiveness of the coupled strongly implicit multigrid method. Kim and Moin [13] employed a method based on a fractional-step, or time-splitting, scheme in conjunction with the approximate factorization technique to simulate the flow inside a lid-driven cavity. de Felice et al. [14] applied a family of single-step time-marching upwind schemes to the lid-driven and thermally-driven square cavity problems.

2. Lid-driven cavity flows

The geometry of the flow in a lid-driven square cavity is shown in Fig. 1. The dimension of the cavity is L. The upper boundary of the cavity moves at a constant velocity U in the x-direction. The lid-driven flows considered in this study also have non-zero velocity component in the y-direction on the upper and lower boundaries of the cavity. The Reynolds number is defined by $Re = UL/\nu$, where ν is the kinematic viscosity of the fluid. Results are presented for $Re = 100$ in this study. The velocity in the x-direction on the upper boundary (U) is 1 and the velocity in the x-direction on the other three boundaries are zero. The velocity in the y-direction on the left and right-hand side boundaries are also set to be zero. The velocity in the y-direction on the upper and lower boundaries are given as follows: