A MULTIGRID-ACCELERATED CODE ON GRADED CARTESIAN MESHES FOR 2D TIME-DEPENDENT INCOMPRESSIBLE VISCOS FLOWS

D. Santhosh Kumar*, Anoop K Dass* and Anupam Dewan**

* Department of Mechanical Engineering, Indian Institute of Technology Guwahati, Guwahati - 781039, India
** Department of Applied Mechanics, Indian Institute of Technology Delhi, Hauz Khas, New Delhi - 110016, India
E-Mail: adewan@am.iitd.ac.in (Corresponding Author)

ABSTRACT: The present work deals with the development of a multigrid-assisted solver for the 2D time-dependent incompressible Navier-Stokes equations on graded Cartesian meshes. As finite difference method is used to discretize the governing equations on nonuniform staggered grids, a transformation of the governing equations from the physical space to the computational space is performed. To obtain second-order time accuracy a fractional-step method is employed. The convective terms are discretized using a third-order accurate upwind scheme and the viscous terms are centrally differenced to fourth-order accuracy. To improve the time-wise efficiency of the code a multigrid technique is employed to solve the pressure-Poisson equation that is required to be solved at every time-step. To establish the accuracy and performance of the code the standard 2D lid-driven cavity flow is computed for unsteady, periodic and asymptotically obtained steady solution for a wide range of Reynolds numbers (Re). The code is then used to compute the transient and asymptotically approached steady flows in a hitherto unexamined problem of two-sided lid-driven square cavity, which involves gradual development of a free shear layer and accompanying off-corner vortices. The computations also show that for this configuration, at Re=2000, there exists a steady solution, about which there was some doubt earlier. The reliability of all known and unknown results in the paper is carefully established and efficiency of the method in respect of grid economy is demonstrated.

Keywords: incompressible flows, fractional-step method, multigrid, nonuniform grid, grid transformation

1. INTRODUCTION

Unsteady incompressible flows occur frequently in problems of academic interest or with engineering applications and one approach adopted for the solution of the unsteady, incompressible Navier-Stokes equations is the fractional-step method (Chorin, 1967; Kim and Moin, 1985). In this approach, an incomplete form of the momentum equations is integrated at each time step to yield an approximate velocity field, which will in general not be divergence-free. Subsequently a correction is applied to the approximate velocity field to produce a divergence-free velocity field. Integration then proceeds to the next time-step. The correction takes the form of the gradient of a scalar, with the scalar field obtained by solving a Poisson equation with the divergence of the approximate velocity as the right hand side. The scalar itself is either the pressure or a pressure correction, depending on the exact form of the fractional-step scheme being used. In the present work the scalar is the pressure instead of pressure correction. Such schemes are called projection methods as the correction to the velocity is accomplished via an orthogonal projection onto the divergence-free field.

Traditional fractional-step or projection methods for incompressible Navier-Stokes equation were introduced by Chorin (1967, 1968). The original Chorin method was modified by Kim and Moin (1985). They used the explicit Adam-Bashforth scheme for the convective terms and the second order Crank Nicholson for the viscous terms. The Poisson equation was solved by a direct method based on trigonometric expansion. In an improvement of the fractional-step method for incompressible Navier-Stokes equations, Le and Moin (1991) solved 3D unsteady flows with fractional-step method combined with three-step Runge-Kutta-type scheme and achieved a gain in time over that obtained by Kim and Moin (1985).

Time dependent turbulent flows can also be solved by the fractional-step method (Rai and Moin, 1991; Choi and Moin, 1994). Fractional-step methods can also be extended to curvilinear coordinate system (Wu, Squires and Wang, 1995).
and these have been modified to solve heat transfer problems (Zhu and Yang, 2003; Armfield and Street, 2004; Cheikh, Beya and Lili, 2007; Younis, Pallares and Grau, 2007).

Finite difference method is frequently used in computational fluid dynamics. The method essentially involves setting up a suitable grid in the problem domain, discretizing the governing equations with respect to the grid and solving them numerically. The common practice is to use a uniform grid, though it may not be the most appropriate one for an efficient computation. An accurate spatial resolution of the solution requires that grid points are clustered in the regions of large gradients. Hence nonuniform grids are used for many flow configurations. Finite difference formulations cannot be easily applied to nonuniform grids. The usual approach adopted is to map the physical space with a nonuniform grid on to a computational space with uniform grid where a transformed set of equations is first solved before mapping this solution back on the physical space for interpretation. The disadvantage of this approach is that, there is an increase in the number of terms to be discretized in the transformed governing equations giving rise to added computations (Kalita, Dalal and Dass, 2004). Many times, however, advantages of using a nonuniform grid outweigh the disadvantages mentioned above. By use of nonuniform grids it is possible to cluster grid lines in the regions of sharp gradients like shear layers and use a relatively coarser grid in the regions involving small gradients, thus obtaining a better spatial scale resolution with a smaller overall grid size. This is the reason why graded Cartesian meshes are used in the present work to have good resolution of the wall-bounded and free shear layers. It has already been mentioned that the transformed governing equations contain a larger number of terms than the original equations, which may result in some increase in the computational cost. Particularly, the pressure-Poisson equation that requires very accurate solution at every time step and consumes more than eighty percent of the total computational time needs special attention and an efficient algorithm for its numerical solution is highly desirable. When the Poisson equation is solved by iterative methods such as the point Gauss-Seidel or line Gauss-Seidel, high-frequency components of the error are effectively reduced, but the low-frequency errors are relatively difficult to remove resulting in a substantial increase in the computational time. Successive over-relaxation (SOR) is known to improve the convergence behaviour of the Gauss-Seidel method. But multigrid, which is arguably the best general convergence acceleration technique (Tannehill, Anderson and Pletcher, 1984), has been found to work better than SOR, especially when the grid size is large. This is the reason why multigrid technique is used in the present paper to compute the pressure-Poisson equation. Fedorenko (1962, 1964) is the originator of the idea of multigrid, which initially did not draw much attention. Interest in the technique was reawakened later by Brandt (1977), which gave it a sound mathematical foundation. Since then the multigrid technique has become one of the most extensively used convergence-acceleration devices in computational fluid dynamics (CFD). Further details on the multigrid method can be found in (Brandt, 1977; Tannehill, Anderson and Pletcher, 1984; Briggs, 1987). In the present paper how multigrid method performs in the numerical solution of the pressure-Poisson equation on both uniform and graded Cartesian meshes is examined in some details and also the relative performances of the algorithm for these two types of meshes is studied. Expectedly multigrid accelerates the convergence of the Gauss-Seidel iterations significantly, which in turn brings about a substantial reduction of cost in the overall transient flow computation.

In the present work a finite-difference discretization of the governing equations is carried out on a staggered grid (Harlow and Welch, 1965; Patankar and Spalding, 1972; Patankar, 1980). It is our experience that implementation of the pressure boundary condition plays a vital role in the computational accuracy of transient flows and that though a colocated grid is possible to use, better results can be obtained using a staggered grid. The convective terms in the Navier-Stokes equations are discretized here using a third-order accurate upwind scheme (Kawamura and Kuwahara, 1984) and the viscous terms are discretized centrally to fourth-order accuracy. As the time accuracy of the fractional-step method used in this work is also of second order, the transient results produced here have the potential of being very accurate. This aspect of the code is examined by first applying it to compute the transient flow in a single-sided lid-driven cavity and then the time-marching steady flow in the same configuration. However, flow does not remain steady for all Reynolds numbers ($Re$) and as it increases it becomes unsteady and sometimes periodic on the way to turbulence. Computations for two such Reynolds numbers $Re=8200$ and 10000 at which the flow becomes
periodic are also carried out. Close agreement with validated results proves the accuracy of the present computation. The code is then applied to a hitherto unexamined situation of computing the transient flow in a two-sided lid-driven cavity that involves gradual development of a free shear layer with associated vortices. Although there are no existing results with which the present ones could be compared the computations are acceptable because of the fact that the accuracy of the code has already been substantiated and that it has a good spatio-temporal accuracy. Use of high-precision arithmetic to offset accumulation of round-off errors in the transient computations lends added credibility to the results. Also the steady results, obtained by marching sufficiently in time, are compared with an existing result (Perumal and Dass, 2008) and a very close agreement is obtained. Thus it can be summarised that the purpose of the present paper is to develop an accurate and efficient computational tool for computing incompressible transient-viscous flows using multigrid method and to gain experience about the relative performance of the algorithm on uniform and nonuniform meshes in the two problems studied that include one hitherto unexamined situation.

The paper is organized in seven sections. Section 2 describes the governing equations in the physical (x,y) and the computational (ξ,η) planes with some details of how the transformation is carried out. Section 3 gives a brief description of the fractional-step method employed and section 4 provides some basics of the multigrid algorithm. Section 5 deals with the numerical method and section 6 with the two test cases. Finally in section 7 we list our observations and conclusions.

2. GOVERNING EQUATIONS

The unsteady, 2D, incompressible N-S equations in the traditional primitive variable formulation in the non-dimensional form can be written as

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0$$

$$\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = - \frac{\partial p}{\partial x} + \frac{\partial^2 u}{\partial y^2}$$

$$\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} = - \frac{\partial p}{\partial y} + \frac{\partial^2 v}{\partial y^2}$$

where $u$ and $v$ denote the velocities along $x$ and $y$ directions, $t$ the time and $p$ the pressure. In view of Eq. (1) solenoidality of the velocity field must be ensured at every instant in a transient computations. It may also be noted that time derivative of pressure does not explicitly appear in the governing equations and hence time-evolution of pressure cannot be computed directly from these equations. Another point worth noting is that the velocities are not dependent on the magnitude of the pressure field but only on its gradient. Values of the pressure to be used in the momentum Eqs. (2–3) are generally determined up to a constant by solving a pressure-Poisson equation that is obtained by taking the divergence of the vector momentum equations (Sunderesan, 1996). This equation containing the Laplacian of pressure on the left hand side and a source term containing various velocity gradients on the right, in two dimensions is

$$\nabla^2 p = \nabla . L(\hat{\mathbf{q}})$$

where $L(\hat{\mathbf{q}}) = \frac{1}{Re} \nabla . \hat{\mathbf{q}} - \frac{\partial}{\partial t} \hat{\mathbf{q}} - (\nabla . \hat{\mathbf{q}}) \hat{\mathbf{q}}$ and $\hat{\mathbf{q}}$ is the velocity vector.

It is clear that in the computation of transient flow using the pressure-velocity formulation this Poisson equation has to be solved very accurately at every time step. As opposed to the solution of the transient momentum Eqs. (2–3), which is a relatively simple matter with an explicit technique, the solution of the pressure-Poisson equation (4) is a nontrivial task that consumes a lot of computational time. For an efficient computation of transient flows using the pressure-velocity formulation it is therefore imperative that this equation is solved with an algorithm involving low computational cost. That is why a multigrid technique is used to solve this pressure-Poisson equation in the present work. We will see later that the form of the source term in the context of the fractional-step will be somewhat different.

2.1 Transformation of governing equations

For incompressible flow the governing equations (1–3) are in the Cartesian coordinates and their general form can be written as

$$a \phi_{,x} + b \phi_{,y} + c \phi_{,x} + d \phi_{,y} + e \phi + f \phi = g(x,y)$$

where $g(x,y)$ is the source term. In order to use nonuniform grids, the governing equations (1–3) have to be transformed to a computational plane in which the computational mesh is uniform and the mathematical characteristics of the equations do not change with the transformation. The transformed version of the governing equations is then solved in the computational plane by a higher order approximation and mapping of the results back to the physical plane is done. Here
we present some details of the transformation. The transformation is of the form
\[ x = x(\xi, \eta), \quad y = y(\xi, \eta) \]
from the physical \( x-y \) plane to the computational \((\xi, \eta)\) plane.

In the discretized N-S equations the derivatives are in the physical domain. These operators are related to those in the computational domain and the key is the simple chain rule of differentiation.

\[
\frac{\partial \phi}{\partial x} = \left( \frac{\partial \xi}{\partial x} \right) \frac{\partial \phi}{\partial \xi} + \left( \frac{\partial \eta}{\partial x} \right) \frac{\partial \phi}{\partial \eta}
\]

A similar relation for the second derivative can be written as
\[
\frac{\partial^2 \phi}{\partial x^2} = \left( \frac{\partial \xi}{\partial x} \right)^2 \frac{\partial^2 \phi}{\partial \xi^2} + \left( \frac{\partial \eta}{\partial x} \right)^2 \frac{\partial^2 \phi}{\partial \eta^2} + \left( \frac{\partial \xi}{\partial x} \right) \left( \frac{\partial \eta}{\partial x} \right) \frac{\partial^2 \phi}{\partial \xi \partial \eta}
\]

Application of the chain rule to the second and fourth terms of Eq. (8) and rearrangement gives
\[
\frac{\partial^2 \phi}{\partial x^2} = \frac{\partial \xi}{\partial x} \frac{\partial^2 \phi}{\partial \xi^2} + \frac{\partial \eta}{\partial x} \frac{\partial^2 \phi}{\partial \eta^2} + 2 \frac{\partial \xi}{\partial x} \frac{\partial \eta}{\partial x} \frac{\partial^2 \phi}{\partial \xi \partial \eta}
\]

Similar equations can be written for the \( y \)-derivatives. Under this transformation, Eq. (5) in the physical space assumes the following form in the computational space:
\[
A \phi_{\xi \xi} + B \phi_{\xi \eta} + C \phi_{\eta \eta} = f(\phi, \phi_{\xi}, \phi_{\eta}, \xi, \eta)
\]
where,
\[
A = a \xi^2 + c \xi \eta + c \eta^2,
\]
\[
B = 2a \xi \eta + b(\xi, \eta, + \xi, \eta, + 2c \xi \eta,
\]
\[
C = a \eta^2 + b \eta \eta + c \eta^2.
\]
Therefore, the governing equations in the computational plane corresponding to the equations (1–3) in the physical plane now become
\[
\xi \frac{\partial u}{\partial \xi} + \eta \frac{\partial v}{\partial \eta} = 0
\]

3. FRACTIONAL-STEP PROCEDURE

The four step fractional-step method (Kim and Moin, 1985) is described here in the uniform and nonuniform Cartesian coordinates. In the first step the momentum equation (17) is solved for an auxiliary velocity field \( \hat{u}_i \) using the pressure from the previous time step; the convective terms and viscous terms are solved explicitly.
\[
\frac{\hat{u}_i - u^*_i}{\Delta t} = -\frac{\partial p^*}{\partial x_i} + R(u^*_i)
\]
where \( R \) is an operator representing the discretized convective and diffusive terms. In the second step Eq. (18) is solved to advance the auxiliary velocity field \( \hat{u}_i \) to which is of course not divergence-free.
\[
\frac{\hat{u}_i - \hat{u}^*_i}{\Delta t} = \frac{\partial p^*}{\partial x_i}
\]
In the third step, the incompressibility condition is enforced by solving the pressure Poisson Eq. (19), which is obtained by taking the divergence of Eq. (20) satisfying continuity for the next time step.
\[
\frac{\partial}{\partial x_i} \frac{\partial p^{usi}}{\partial x_i} = \frac{1}{\Delta t} \frac{\partial u^*_i}{\partial x_i}
\]
Finally in the fourth step the velocity which satisfies the incompressibility condition is obtained by using the following correction step:
\[
\frac{(u^{usi} - u^*_i)}{\Delta t} = -\frac{\partial p^{usi}}{\partial x_i}
\]
By using Eqs. (18) and (20) it can be easily shown that the error is of second order in time as the time derivative has been discretized in a centered approximation at the time location \( n+1/2 \). Similar procedure may be followed for
nonuniform grid in the computational plane. The above four-step fractional-step method for the transformed equations on a uniform grid may be written as follows:

\[ u - u^* = \frac{\partial \xi_j}{\partial x} \frac{\partial p^*}{\partial \xi_j} + R(u^*) \]  

(21)

\[ u^* - u = \frac{\partial \xi_j}{\partial x} \frac{\partial p^*}{\partial \xi_j} \]  

(22)

\[ \frac{\partial^2 \xi_j}{\partial x^2} \frac{\partial^2 p}{\partial \xi_j} + \frac{\partial \xi_j}{\partial x} \frac{\partial^2 p}{\partial \xi_j^2} + \frac{\partial \xi_j}{\partial x} \frac{\partial \xi_j}{\partial \xi_j} \frac{\partial^2 p}{\partial \xi_j^2} = \frac{1}{\Delta t} \left( \frac{\partial \xi_j}{\partial x} \frac{\partial u^*}{\partial \xi_j} \right) \]  

(23)

\[ \frac{u^* - u}{\Delta t} = \frac{\partial \xi_j}{\partial x} \frac{\partial p^*}{\partial \xi_j} \]  

(24)

The Poisson equation in the transformed plane is given by Eq. (23), which is solved using the multigrid method.

### 3.1 Poisson solver

The fractional-step method described in the previous section is required to solve the Poisson equation Eq. (19) on a nonuniform grid, which is equivalent to solving Eq. (23) on the uniform transformed grid at each time step to satisfy the divergence-free condition. In this case the pressure-Poisson equation is derived by taking the divergence of the intermediate velocity field Eq. (20) subjected to continuity constraint for the next time step. On the physical plane the Poisson equation is given as

\[ \frac{\partial^2 p}{\partial x^2} + \frac{\partial^2 p}{\partial y^2} = \frac{1}{\Delta t} \left( \frac{\partial u^*}{\partial x} + \frac{\partial v^*}{\partial y} \right) \]  

(25)

On the transformed space this Poisson equation, given below, has more number of terms

\[ \frac{\partial^2 p}{\partial \xi_j^2} + \frac{\partial^2 p}{\partial \eta_j^2} + \eta_{\eta_j} \frac{\partial p}{\partial \eta_j} + \eta_{\eta_j} \frac{\partial^2 p}{\partial \eta_j^2} = \sigma \]  

(26)

where, \( \sigma \) is the source term and that has the form

\[ \frac{1}{\Delta t} \left( \frac{\partial u^*}{\partial \xi_j} \right) + \eta_{\eta_j} \frac{\partial v^*}{\partial \eta_j} \]  

(27)

Since Poisson equation takes almost 90\% of the total CPU time required to update the primitive variables over one time step, a four-level, V-cycle multigrid method is used to accelerate the convergence.

### 4. DESCRIPTION OF MULTIGRID ALGORITHM

#### 4.1 Need for multigrid methods

The pressure-Poisson equations Eq. (25) or Eq. (26) are normally solved by the Gauss-Seidel method (GSM). The iterative solution of the pressure-Poisson equation is the most time consuming part in the overall solution procedure. For high Reynolds number flows and highly stretched grids, it is very time consuming to obtain the converged solution of the pressure Poisson equation up to the machine accuracy. Therefore the use of a convergence enhancing scheme such as the multigrid method is desirable.

In the initial stages of the computation, the solution of the Poisson equation alone takes approximately 95\% of the total CPU time required to update all the primitive variables over one time step. Even though this percentage drops down in the later states of computations, there is a strong need to use some convergence-acceleration technique. Multigrid method has been chosen to accelerate the convergence rate, as it is highly suitable for elliptic equations such as the Poisson equation. Multigrid methods have the ability to provide grid independent convergence rates as the number of grid points is increased to large values in a fixed domain. There are two broad categories of multigrid methods: (a) correction schemes (CS) (Briggs, 1987) and (b) full approximation schemes (FAS) (Hortmann, Peric and Scheuerer, 1990; Lien and Leschziner, 1994). The correction scheme is applicable to linear systems only; they operate with corrections to the solution on coarse grids, which are eventually added to the absolute fine-grid solution. As the Poisson equation is a linear equation, correction scheme is used in the present work.

#### 4.2 Linear two-grid algorithm

A two grid algorithm for linear problems consists of smoothing on the fine grid, approximation of the required correction on the coarse grid, prolongation of the coarse grid correction to the fine grid and again smoothing on the fine grid. Let the equation to be solved is \( \Delta^2 p = S \) where \( S \) denotes the source term. This equation after discretization by the finite difference method can be written as \( L(p)_{ij} = S_{ij} \), where \( L \) denotes the Laplacian operator. A two grid correction scheme is described here.

1. Perform \( n \) iterations on a fine grid, \( L(p)_{ij} = S_{ij} \).
2. Compute residual \( R_{i,j}^f = L(p)_{i,j} - S_{i,j} \) and store at each point. This residual is restricted to next coarser grid so that their lower frequency error components can be smoothed and the restricted residual is denoted by \( I_j^c R_{i,j}^f \).

3. The correction equation, given by
\[
L(\Delta p)_{i,j}^c = -I_j^c R_{i,j}^f
\]  
(28)
is iterated few times on coarser grid using zero as the initial guesses while keeping the residual fixed at each grid point. The solution \( \Delta p_{i,j} \) represents the correction to the grid solution.

4. The corrections obtained on the coarsest grid are prolonged on to the next finer grid and denoted by \( I_j^c (\Delta p)_{i,j}^c \).

5. Correct the fine grid approximations using \( p_{i,j}^{new} = p_{i,j}^{old} + I_j^c (\Delta p)_{i,j}^c \). It is necessary to repeat the above two-level cycle until desired convergence is achieved.

4.3 Multigrid strategies

The two-grid algorithm described above consists of only one coarse grid. If multiple coarse grids are present, steps 2 and 3 of Section 4.2 are repeated until the coarsest grid is reached. It is important to realize that the residual on a coarse grid formed from Eq. (28) is given by
\[
R_{i,j}^2 = L[\Delta p]_{i,j}^c + I_j^c R_{i,j}^f
\]
(29)

Once smoothing is done on the coarsest grid, step 4 is successively repeated until the finest grid is reached again. This procedure is known as the V-cycle. The computational effort can be reported in terms of the equivalent fine grid sweeps.

5. NUMERICAL METHOD

The governing equations are discretized in space, using a finite difference formulation on a staggered, Cartesian grid (Harlow and Welch, 1965), where velocities and pressure are calculated at different locations. The main advantage of the staggered grid arrangement is the strong coupling between pressure and velocities without requiring special interpolation techniques. This helps avoid convergence problems and oscillations in pressure fields. Another advantage of using staggered grid for incompressible flows is that the pressure boundary conditions are not required when the momentum equations are evaluated. The boundary conditions for velocity are straightforward, and are directly imposed on the walls of the cavity whereas the pressure boundary conditions are somewhat tricky. There are no natural boundary conditions for pressure as in the case of velocity. Since pressure is coupled with the velocity, Peyret and Taylor (1983) state that, “The primary difficulty with this approach (primitive variable formulation) is specification of boundary condition for pressure”. We can derive the pressure boundary condition from momentum equation by substituting velocities. The boundary conditions thus obtained are called dynamically consistent pressure boundary conditions. This is naturally true for colocated grid but for staggered grid there is no need of such a pressure boundary condition. For the pressure-Poisson equation the pressure gradient normal to the walls is assumed zero, i.e.,
\[
\frac{\partial p}{\partial x_j} = 0
\]  
(30)
The convective terms of the momentum equations are approximated with Kuwahara’s third-order upwind scheme (Kawamura and Kuwahara, 1984) (Eq. (31)) and viscous terms are discretized with a fourth-order central-difference scheme (Eq. (32)) as given below:

\[
\left( \frac{u}{\partial \xi} \right)_{i,j} = u_{i,j} \left(u_{i-2,j} - 8u_{i-1,j} + 8u_{i+1,j} - u_{i+2,j}\right)/12\Delta \xi
\]
(31)

\[
\left( \frac{\partial^2 u}{\partial \xi^2} \right)_{i,j} = \left( -u_{i-2,j} + 16u_{i-1,j} - 30u_{i,j} + 16u_{i+1,j} - u_{i+2,j}\right)/\left(12(\Delta \xi)^2\right)
\]
(32)
6. COMPUTATIONS, RESULTS AND DISCUSSIONS

The code developed is second-order accurate in time, third-order accurate in the spatial discretization of the convective terms and fourth order accurate in the spatial discretization of the viscous terms. In order to evaluate the performance of the code in regard to time-wise efficiency and accuracy, it is applied to two test cases to compute transient viscous flows which asymptotically reach steady state if the solution is advanced sufficiently long in time. The first test case is the standard 2D incompressible single-sided lid-driven cavity problem where the fluid in a square cavity is set into motion by the movement of one wall. The present code is used here to record the history of flow development and also to capture the final steady-state results. The second test case is the 2D two-sided lid-driven square cavity problem where the motion of the fluid in a square cavity is induced by the parallel motion of two walls. This problem is relatively new for which no transient computations exist. After having gained confidence in the code by applying it first to an already examined problem we believe that the present transient computation for the two-sided lid-driven cavity enjoys a high level of credibility. In both the test-cases, except in the moving walls, velocity everywhere else is assumed zero initially to ensure a divergence-free initial condition.

6.1 Transient flow in a single-sided lid-driven square cavity

This problem, over the years, has become the most frequently used benchmark problem for the assessment of numerical methods, particularly the steady-state solution of incompressible fluid flows governed by the Navier-Stokes equations (Ghia, Ghia and Shin, 1982; Bruneau and Jouron, 1990; Barragy and Carey, 1997; Botella and Peyret, 1998; Kallita, Dalal and Dass, 2002; Auteri, Quartapelle and Vigevano, 2002; Bruneau and Saad, 2006; Santhosh, Suresh and Das, 2009). This problem is of significant scientific interest because it displays almost all fluid mechanical phenomena for incompressible viscous flows in the simplest of geometrical settings. The geometry and the boundary conditions are shown in Fig. 1 where the top wall is moving and the remaining three walls are stationary. The no-slip boundary conditions have been employed on all the four walls. The dimensionless boundary conditions are given as

\[ u = 1.0, \quad v = 0.0 \quad (y = 1) \]
\[ u = 0.0, \quad v = 0.0 \quad (y = 0) \]
\[ u = 0.0, \quad v = 0.0 \quad (x = 0, 1) \]

The moving wall generates vorticity which diffuses inside the cavity and this diffusion is the driving mechanism of the flow. At high Reynolds numbers, several secondary and tertiary vortices begin to appear, whose characteristics depend on Re. Because of the presence of large gradients near the walls, the grids are clustered there to capture the smaller spatial scales in these regions.

![Fig. 1 Schematic diagram of single-sided lid-driven square cavity.](image1)

![Fig. 2 A 65x65 grid in (a) Physical plane and (b) Computational plane.](image2)
6.1.1 Grid used

In this case to transform the clustered centro-symmetric grid in the physical plane \((x, y)\) (Fig. 2(a)) to a uniform grid in the computational plane \((\xi, \eta)\) (Fig. 2(b)) the following stretching function (Kalita, Dalal and Dass, 2004) is used:

\[
x = \xi - \frac{\lambda}{2\pi} \sin(2\pi \xi)
\]

(33)

\[
y = \eta - \frac{\lambda}{2\pi} \sin(2\pi \eta)
\]

(34)

where \(\lambda\) denotes the stretching parameter with \(0 < \lambda < 1\). It may be noted that \(\lambda = 0\) results in a uniform grid and \(\lambda = 0.9\) produces dense clustering. In the present case \(\lambda\) is taken as 0.6.

6.1.2 Multigrid solution of the pressure-Poisson equation

In the present study a 4-level V-cycle multigrid shown in Fig. 3 has been used in all the computations to solve the pressure-Poisson equation and the number of sweeps used in various grid-levels is given inside the corresponding circles. For a 2-level scheme on a 129×129 grid at \(Re = 3200\), performance is seen to improve by increasing the number of sweeps in the coarser grid. When the number of sweeps increases to 60, the number of equivalent fine grid sweeps required for the convergence ultimately reduces to 321, which represents a reduction of computational effort to nearly one-third for the 2-level scheme. For \(Re = 3200\) on a 129×129 grid, it is observed that using four or five level of grids is good enough as further increase of levels produce no time-wise gain. Specifically, it was observed that 3736, 321, 164, 36, 29, 29 and 29 work units (WU) (equivalent fine-grid sweeps) were required when using 1-, 2-, 3-, 4-, 5-, 6- and 7-levels respectively (Fig. 4(a)). As mentioned earlier Gauss-Seidel method initially consumes more than 95% of the total computational time required to update all the primitive variables over one time step. By using the multigrid the computational time is therefore cut down to roughly a factor of 1/30, which is significant indeed. To gain an insight into the convergence behaviour after around 100 time steps, at \(t = 0.1\) a plot between the pressure error and number of iterations required to solve the pressure-Poisson equation is shown in Fig. 4(b). Before examining overall quantitative performance data, it is instructive to consider the convergence histories in Fig. 4(b) obtained with the single grid and with sequences of 2-, 3- and 4-levels. Expectedly the convergence curves relating to multigrid are considerably steeper than that relating to the single grid. The ratio of the computational effort in work units for the single grid to that for the multigrid may be termed speed-up. As the convergence limit of error is lowered speed-up is seen to increase. This is
because on a single grid error-reduction rate becomes smaller and smaller as the iteration progresses whereas multigrid maintains more or less the same error-reduction rate throughout. The performance of the multigrid with the optimum number of levels is truly amazing. The number of sweeps is seen to be nearly independent of the number of grid points used. Only 29 sweeps were required for the finest grid compared to 3736 for the conventional Gauss-Seidel method. This is a reduction in effort by a factor of 128. The multigrid with the optimum number of levels, 4 in the present case, requires only 1/11th as much effort as the multigrid with a 2-level scheme. The speed-ups obtained by 2- and 4-level multigrids are plotted against grid nodes in Fig. 5. We note that this convergence rate is achieved on a uniform Cartesian grid. A similar behavior has been experienced on a graded Cartesian grid.

Fig. 5 Speed-up for multigrids with a 2- and 4(optimum)-level.

Fig. 6 Time-step-independence study: (a) Horizontal velocity along vertical centreline and (b) Vertical velocity along horizontal centreline.

Fig. 7 Tertiary vortex at the bottom right corner of the cavity at $Re = 1000$: (a) Uniform grid and (b) Nonuniform grid.
6.1.3 Grid- and time-step-independence study

Fig. 6(a–b) respectively show the horizontal velocity along the vertical centreline and vertical velocity along the horizontal centreline of the cavity at instant \( t=20.0 \) captured with three different time steps \( \Delta t=0.001, 0.0005 \) and \( 0.0001 \) for \( Re=3200 \). These profiles plotted for a \( 129 \times 129 \) grid show no discernable differences, proving the time-accuracy of the present computations together with the fact that the transient results are also physically meaningful. Therefore, a time step of 0.001 has been used for all subsequent computations. Fig. 7(a–b) show the steady state vortices at the bottom right corner of the cavity for uniform and nonuniform grids, respectively at \( Re=1000 \). It is clear that without clustering in the vicinity of the walls a uniform grid cannot predict appropriate tertiary corner vortices at the bottom right corner (Fig. 7(a)), whereas a nonuniform grid is able to resolve the tertiary vortices accurately (Fig. 7(b)). This is the justification for using a nonuniform grid with associated complexities, in that it allows more number of points to be put in the regions of sharp flow gradients thus obtaining better scale resolution. Fig. 8(a–b) present a comparison of the steady-state horizontal velocities on the vertical centreline and the vertical velocities on the horizontal centreline of the square cavity with those obtained by Ghia, Ghia and Shin (1982) for both uniform and nonuniform grids. Both the results agree well with those reported in Ghia, Ghia and Shin (1982) and there is negligible deviation of velocity profiles between the uniform and nonuniform grids. To lend further credibility to the present computations we also compare the present mid-span \( u \) and \( v \) velocity profiles at \( Re=1000 \) and 5000 with the very recent results of Santhosh, Suresh and Das (2009) in Figs. 9(a–b) and 10(a–b). The high accuracy of the present computations is brought out by the close matching of the results. In fact, Fig. 9(a–b) shows no discernible differences between the two sets of results. As the present results are produced on a \( 129 \times 129 \) graded Cartesian mesh and Santhosh et al. (2009) used a uniform \( 512 \times 512 \) grid, the grid economy of the present computations is also impressive.

---

**Fig. 8** Comparison of steady-state velocity profiles: (a) Horizontal velocity along the vertical centreline and (b) Vertical velocity along the horizontal centerline for the single-sided lid-driven cavity.

**Fig. 9** Comparison of present steady state \( u \) and \( v \) profiles on a \( 129 \times 129 \) grid with those by Santhosh, Suresh and Das (2009) on a \( 512 \times 512 \) grid at \( Re=1000 \).
Fig. 10 Comparison of present steady state \( u \) and \( v \) profiles on a 129 \( \times \) 129 grid with those by Santhosh, Suresh and Das (2009) on a 512 \( \times \) 512 uniform grid at \( Re = 5000 \).

Fig. 11 Time evolution of streamlines for the single-sided lid-driven square cavity at \( Re = 1000 \).

6.1.4 Transient results for \( Re = 1000 \)

For transient computations, in order to give a divergence-free initial velocity field, \( x \)-component of velocity \( u \) and \( y \)-component of velocity \( v \) are both taken as zero at all grid nodes except on those on the moving top wall where \( u \) is taken as 1 and \( v \) as zero. Fig. 11 shows the time evolution of the streamline pattern for \( Re = 1000 \) till the steady state is reached. At time \( t = 1.25 \) (Fig. 11(a)) a recirculation zone is seen near the top right corner of the cavity. From Fig. 11(a–f) it is seen that this primary vortex core gradually moves towards the geometric centre of the cavity. At \( t = 5.0 \) (Fig. 11(c)) a small secondary recirculation zone at the bottom right corner is seen and another such zone slightly below the centre of the right wall, which with time moves downwards and merges with the secondary vortex at the bottom right corner. Also seen at this instant is the emergence of an incipient vortex at the bottom left corner, which grows bigger with time. At \( t = 10.0 \) (Fig. 11(d)) it is seen that after the two secondary vortices merge at the bottom right corner the resultant vortex there is much larger and the secondary vortex at the bottom left corner also grows bigger. At time \( t = 40.0 \) (Fig. 11(e)) the bottom left secondary vortex is
seen to be much bigger although there is not much change in the bottom right vortex. Steady state, for a mass residual of $10^{-5}$, is obtained at $t=104$ (Fig. 11(f)) where, compared with $t=40.0$, some minor readjustment is seen to take place especially at the top left corner. By examining the evolution of the streamline pattern two important observations can be made. As mentioned earlier the primary vortex core generated at the top right gradually moves with time towards the geometric centre of the cavity though it does not actually reach there even at the steady state for $Re=1000$ (Fig. 11(f)). Secondly when the top wall moves from the left to right, spots of important near-wall changes gradually move with time from the top left corner in a clockwise sense along the cavity walls.

**6.1.5 Asymptotically obtained steady solutions**

The steady-state streamline patterns obtained by time-marching with the present code for $Re=1000$, 3200, 5000 and 7500 are shown in Fig. 12, which agree quite well with those in Ghia, Ghia and Shin (1982) and Santhosh, Suresh and Das (2009). It may be concluded that though we use a $129 \times 129$ grid, as a result of grid-clustering near the wall and good accuracy of the spatial discretization, the results obtained are very accurate. Apart from the changes of the near-wall vortex pattern with $Re$, another conclusion that can be made is that as the Reynolds number increases the proximity of the primary vortex core and the geometric centre of the cavity increases.

**6.1.6 Periodic solutions**

Until $Re=7500$, we see that the transient computations asymptotically approach a steady solution on both $129 \times 129$ and $257 \times 257$ grids. To lend credibility to our unsteady code, we carried out our computations for still higher Reynolds numbers to capture periodic solutions reported in the literature (Bruneau and Saad, 2006). It has been found from earlier works (Fortin et al., 1997; Auteri, Quartapelle and Vigevano, 2002; Bruneau and Saad, 2006) that the first Hopf bifurcation takes place around $Re=8000$. In order to find the periodic solution, computations are carried out first for $Re=8200$. The numerical solution develops a time periodic pattern on a $257 \times 257$ grid, whether uniform or nonuniform. The streamline pattern for $Re=8200$ is shown in Fig. 13. The pattern plotted at different instants of time looks almost identical as the fluctuations are small in magnitude. Probing further, however,
reveals that small periodic oscillations occur in the solution. The time evolution of $u$, the $x$-component of velocity at the monitoring point (14/16, 2/16) is plotted in Fig. 14(a) and a Fourier analysis performed for this signal is shown in Fig. 14(b). From this figure, the frequency is seen to be 0.45 and has only one dominant harmonic and one subharmonic, confirming that the solution is periodic. The behaviour is exactly identical for other quantities such as $y$-component of velocity, pressure and vorticity. Moreover similar results are obtained at the other points considered (2/16, 2/16), (2/16, 14/16) and (14/16, 14/16) with some variations in amplitude of the signals. At $Re=8200$ these points are chosen so as to analyse the behaviour of the solution in every part of the cavity. The variation in streamline pattern is not illustrated because the variations in velocities are small and there are no easily distinguishable differences in the patterns. It is expected that further increase in $Re$ is likely to increase the amplitude of the oscillations. We therefore, carried out the computations at $Re=10000$. This value was the subject of discussion for quite a long time, as the question was to know whether a stable steady solution is obtained or not for this Reynolds number. Though some of the previous works (Ghia, Ghia and Shin, 1982; Shen, 1991; Barragy and Carey, 1997; Erturk, Corke and Gokcol, 2005) reported stable steady solutions for this value, there are some others (Fortin et al., 1997; Auteri, Quartapelle and Vigevano, 2002; Bruneau and Sadd, 2006) that reported existence of periodic solution at this Reynolds number. Therefore to explore the actual potential of the present code to compute transient flows we decided to use it to re-examine this interesting and somewhat controversial issue. The $x$-component of velocity plotted in Fig. 15(a) with time indicates that the solution reaches a stable periodic state. A power spectrum plotted in Fig. 15(b) using Fourier analysis confirms that the solution is periodic with one dominant harmonic and two subharmonics. After the flow reaches a stable periodic state, a phase diagram (Fig. 15(c)) is drawn between $u$ and $v$ at the monitoring point (2/16, 14/16). These plots indicate a perfect periodic pattern for the solutions obtained by us.
To demonstrate the global periodic nature of the flow, in Fig. 16 the streamline pattern is plotted at discrete time intervals for roughly a period $\Delta t=2.3$ of the oscillations. Clearly the streamline pattern is repeated after a period. Indeed, the primary vortex is still attached to the three walls of the cavity but the secondary vortices are unstable. Also we observe that there are persistent oscillations at the top left and bottom left secondary vortices. The other significant changes during one period are the periodic appearance and disappearance of two tertiary vortices at the bottom left and the top left. These observations strongly indicate the global periodic nature of the solution in the entire domain, which substantiates the observations in (Fortin et al., 1997; Auteri, Quartapelle and Vigevano, 2002; Bruneau and Sadd, 2006). It may be noted the main aim of the present work is to explore and establish the potential of the code and by capturing this periodic flow behaviour the code has established its accuracy and ability. Another aim of the present work is to carry out a comparison exercise to evaluate the performance of multigrid over single grid and also of multigrid for the uniform and nonuniform grids.

### 6.1.7 Multigrid performance

Tables 1 and 2 give the CPU times of multigrid and single grid computations to reach the asymptotic steady state and the time-wise speed-up achieved by multigrid for various Reynolds numbers on both uniform and nonuniform grids of size $129 \times 129$. For the same fall of residual, time-gain by multigrid is impressive. The time-wise speed-up achieved by multigrid at “steady state” is ten or more for the various Reynolds numbers on the uniform grid, which for the nonuniform grid is about ten or slightly less. The fact that on the nonuniform grid multigrid performance is somewhat poorer can be attributed to the appearance of additional terms in the transformed pressure-Poisson equation for the nonuniform grid. It is also observed that work units required to reach the steady state increase as $Re$ increases for both single-grid and multigrid. This can be attributed to the fact that high Reynolds number flows contain multiplicity of scales, which introduce high frequency errors into the computational process. The time-wise speed-up achieved by multigrid is also seen to decrease as $Re$ increases for both uniform and nonuniform grids. Another point worth noting is that the CPU time and speed-up appearing in Tables 1 and 2 give the most conservative picture of the
multigrid performance and the speed-up achieved for initial instants of time is considerably higher. This fact is observed from Table 3 where the speed-ups achieved by multigrid at \( t = 10 \) have been listed for both uniform and nonuniform grids. This is because as the steady state is approached pressure does not change much with time and multigrid does not exert as much influence on the convergence rate as it does on earlier instants of time. All the CPU times given in Tables 1, 2 and 3 correspond to the computations carried out on a Dual Xeon 3.2 GHz based machine.

Table 1 Performance of multigrid for a 129\( \times \)129 grid on a uniform grid at various Reynolds number.

<table>
<thead>
<tr>
<th>Re</th>
<th>CPU time (minutes)</th>
<th>Speed-up</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Single-grid</td>
<td>MG 4-level</td>
</tr>
<tr>
<td>1000</td>
<td>772</td>
<td>63</td>
</tr>
<tr>
<td>3200</td>
<td>1628</td>
<td>138</td>
</tr>
<tr>
<td>5000</td>
<td>3900</td>
<td>368</td>
</tr>
<tr>
<td>7500</td>
<td>6997</td>
<td>686</td>
</tr>
</tbody>
</table>

Table 2 Performance of multigrid for a 129\( \times \)129 grid on a nonuniform grid at various Reynolds number.

<table>
<thead>
<tr>
<th>Re</th>
<th>CPU time (minutes)</th>
<th>Speed-up</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Single-grid</td>
<td>MG 4-level</td>
</tr>
<tr>
<td>1000</td>
<td>1557</td>
<td>161</td>
</tr>
<tr>
<td>3200</td>
<td>3872</td>
<td>412</td>
</tr>
<tr>
<td>5000</td>
<td>5353</td>
<td>645</td>
</tr>
<tr>
<td>7500</td>
<td>7163</td>
<td>885</td>
</tr>
</tbody>
</table>

Table 3 Performance of multigrid on uniform and nonuniform grids at \( Re = 1000 \) and \( t = 10 \).

<table>
<thead>
<tr>
<th>Grid</th>
<th>CPU time (minutes)</th>
<th>Speed-up</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Single-grid</td>
<td>MG 4-level</td>
</tr>
<tr>
<td>uniform</td>
<td>496</td>
<td>17</td>
</tr>
<tr>
<td>nonuniform</td>
<td>612</td>
<td>29</td>
</tr>
</tbody>
</table>

6.2 Transient flow in a two-sided lid-driven square cavity

The validation exercises, results and discussions in the sub-section 6.1 establish the accuracy and the capability of the present code to compute transient viscous flows. After having thus gained confidence in the code it is then applied to compute the transient flow in a configuration for which no transient results exist. The problem consists of a square cavity filled with a stationary fluid that is set into motion by the sudden simultaneous movement of the top and the bottom walls from the left to right (Fig. 17(a)). A similar problem involving rectangular cavities has been studied by Kuhlmann, Wanschura and Rath (1997) to produce multiple steady solutions. Perumal and Dass (2008) used lattice Boltzmann method to compute steady solution for the problem, with which the present steady results obtained asymptotically by marching with time are compared. The reason for selecting this flow configuration for numerical study is that it is fraught with interesting flow features including gradual development of a free shear layer and associated off-corner secondary vortices. Use of a grid (Fig. 17(b)) with appropriate clustering not only to capture the wall bounded shear layers but also the free-shear layer and use of multigrid suggests that the present computations combine accuracy, space-grid economy and time-wise efficiency. Computations are carried out for

![Fig. 17 Two-sided lid-driven square cavity flow: (a) Flow configuration, (b) Grid (129 \( \times \) 129).](image)
Fig. 18 Steady-state streamlines patterns for different Reynolds numbers for the two-sided lid-driven square cavity on a 129×129 clustered grid.

Re=100, 400, 1000 and 2000. Fig. 18 shows the steady-state streamline patterns asymptotically obtained from the present transient computations. These patterns agree quite well with those reported in Perumal and Dass (2008). Fig. 18(a) shows that at Re=100 two counter-rotating primary vortices appear and there is no trace of any near-wall secondary vortices. At Re=400 (Fig. 18(b)), apart from the primary vortices a counter-rotating pair of secondary vortices symmetrically placed about the horizontal cavity centreline is seen to appear near the centre of the right wall. Also a free shear layer about this centreline is now more prominent. Fig. 18(c–d) shows the streamline patterns at Re=1000 and 2000 respectively. It is seen that from Re=400 to Re=1000 the size of the secondary vortices increases significantly whereas from Re=1000 to 2000 the increase in size is very small. It is also seen that with an increase in Re the distance of the top and bottom primary vortex cores from the top right and bottom right corners increases and at Re=2000 they take position close to the centres of the top half and the bottom half of the cavity. The counter-rotating pairs of primary and secondary vortices maintain their symmetry about the horizontal centreline for all the Reynolds numbers. Fig. 19(a) plots the x-component of velocity along the vertical centreline (x=0.5) of the cavity at Re=100 and Re=1000 and Fig. 19(b) plots the y-component of velocity along the horizontal line y=0.75 at the same Reynolds numbers. Plotted alongside these curves are those reported in Perumal and Dass (2008). Close agreement of the present steady results with the very carefully produced results in Perumal and Dass (2008) shows that the spatial accuracy and resolution obtained by the present code on a relatively coarse clustered grid is quite high. Another conclusion that can be drawn from the present computation concerns the existence of a stable steady solution at Re=2000. Kuhlmann, Wanschura and Rath (1997) points out that for a two-sided lid-driven rectangular cavity with parallel motion of the walls and a certain aspect ratio greater than unity and beyond, multiple steady solutions exist; whereas for a two-sided lid-driven square cavity there are no multiple solutions and steady solutions are unique. However, as steady equations were numerically
solved to produce the results reported in Perumal and Dass (2008) it was not very clear until now whether there exists a unique steady and stable solution for the present flow-configuration at \( Re=2000 \). As the present code with proven ability to produce periodic solutions is seen to produce steady results at \( Re=2000 \), it conclusively proves that for this Reynolds number and configuration, there indeed is a unique steady solution.

For computing transient results, the initial divergence-free velocity field used in this work is obtained by taking both \( u \) and \( v \) as zero at all grid-nodes except those on the top and the bottom moving walls where \( u \) is taken as unity and \( v \) as zero. Fig. 20 gives the picture of time-wise evolution of the streamline pattern at \( Re=1000 \) till the steady state is reached. Fig. 20(a) shows the streamline pattern at \( t=1.0 \) when the primary
vortex cores are seen close to the top right and bottom right corner symmetric to each other about the horizontal centreline. At this instant the secondary vortices have not developed as yet. At time \( t = 3.0 \) (Fig. 20(b)) the two primary vortex cores are seen to move away from the nearest corners maintaining their symmetry about the horizontal centreline. Also seen at this instant is the emergence of a pair of two counter-rotating secondary vortices symmetrically placed about the horizontal centreline near the centre of the right wall. At time \( t = 5.0 \) (Fig. 20(c)) the movement of the primary vortex cores away from their nearest corners is seen to continue whereas the size of the pair of secondary vortices enlarges. At time \( t = 7.0 \) (Fig. 20(d)) the movement of the primary vortex cores away from the respective corners is seen to continue but secondary vortex sizes do not change much compared with those at \( t = 5.0 \). Fig. 20(e) shows the streamline patterns at time \( t = 10.0 \), when the primary vortex cores are seen to move further away from their corners of origin and almost reach the centres of the top and bottom halves. For a fall of mass residual below \( 10^{-5} \) the steady state is reached at time \( t = 82.2 \). At this steady state the two counter-rotating primary vortex cores are stably positioned very close to the geometric centres of the top and bottom halves of the cavity. Also there are no changes in the positions and shapes of the counter-rotating pair of secondary vortices. Observing these figures in the chronological sequence also reveals that the flow gradient in the free shear layer about the horizontal centreline of the cavity becomes sharper with time.

The accuracy of the present steady results for the two-sided lid-driven cavity flow obtained on a relatively coarse clustered grid of \( 129 \times 129 \) is already seen to be high as they compare very well with carefully produced results (Perumal and Dass, 2008) on much finer uniform grids. This has been possible as a result of using discretizations of good spatial accuracy and adequately clustered grids in the regions of sharp flow gradients. The transient computations for this hitherto unexamined flow configuration are also highly reliable. This is because of the second order time accuracy of the scheme and the care taken to maintain time-step-independence of the results throughout. Use of high precision arithmetic to avoid possible accumulation of round-off errors lends further credibility to the results. The time-wise efficiency of the transient computations is also high as multigrid has been used to good effect. Overall these results are produced accurately and efficiently with grid economy.

7. CONCLUSIONS

The main aim of the present work is the development of a 2D transient Navier-Stokes solver and tests its capability and efficiency in known and unknown situations. In the efficiency front, to achieve grid economy graded Cartesian meshes coupled with grid-transformation are used and to enhance the time-wise efficiency, multigrid method is used to solve the cumbersome pressure-Poisson equation that needs to be solved at every time step in a transient solver. To test the capability of the code it is first applied to capture the transient and steady-state flow details in the configuration of a standard 2D single-sided lid-driven cavity. Care has been taken to produce results independent of the space-grid and time step. As already mentioned the discretizations used in the present code achieve a second order temporal and at least a third order spatial accuracy. Consequently the results that are produced on relatively coarser grids (because of graded mesh) with good time-wise efficiency (because of multigrid) agree quite well with highly reliable existing results. The capability of the code is further tested by capturing periodic flows in the cavity for \( Re = 8200 \) and 10000 at which some of the earlier works showed time-stationary results. After having thus gained an insight into various aspects of the code including its accuracy and capability, it is then used to capture the transient flow features that evolved with time in a 2D two-sided lid-driven cavity. This flow configuration is fraught with many interesting issues in that it involves development of a pair of off-corner vortices and a free shear layer. To obtain better flow resolution the grids are clustered adequately at appropriate locations. Given the accuracy and capability of the code there is no doubt that the transient results for this untested flow configuration are highly reliable and accurate. It may be mentioned that steady results for some Reynolds numbers for this geometry exist but as they are produced by solving steady equations it was not clear whether the flow was really steady at, for example, \( Re = 2000 \). This issue was resolved by the present code by showing that the flow at \( Re = 2000 \) for the two-sided lid-driven cavity is indeed steady. Another issue addressed in this work is the comparative performance of the multigrid algorithm on uniform and nonuniform grids of equal sizes. Though acceleration produced by the multigrid on a
uniform grid is seen to be somewhat better than that for a nonuniform grid, this fact is not significant as in many situations it is possible to use a nonuniform grid involving much smaller number of grid nodes to produce the same quality of solution as on a uniform grid. The results and the subsequent discussions suggest that the discretizations and solution algorithm used and grids chosen render the present computations accurate and efficient and the developed code can be considered a good numerical tool.

REFERENCES


