Numerical solution of 2-D steady incompressible lid-driven cavity flow with three different numerical schemes

GHARAHJEH, Siamak,
Middle East Technical University, Ankara, Turkey. 06800

ASHRAF, Ammar,
Middle East Technical University, Ankara, Turkey. 06800

SHIRAZI, Ghorban Mahtabi
University of Zanjan, Zanjan, Iran

Abstract

Numerical solution of 2-D steady incompressible lid-driven laminar cavity flow is presented. Three different numerical schemes are employed to make a comparison on the practicality of the methods. Alternating direction implicit (ADI) scheme for the vorticity stream function formulation, explicit and implicit scheme for primitive variable formulation of governing Navier-Stokes equations of flow were attempted. A fairly fine uniform grid was adopted for all the cases after a technical procedure was applied to come up with the proper mesh size that would make the solution roughly independent of mesh quality. The solutions obtained for different Reynolds numbers are presented and compared. Superiority of numerical approaches are investigated and compared to benchmark solutions available in the literature.

Keywords: Lid driven cavity flow, Numerical schemes, N-S equations.

1. Introduction

The lid driven cavity flow is considered as one of the most studied problems in the field of computational fluid dynamics (CFD) due to its simple geometry and the fluid inside retaining all the flow physics. The flow is confined from all four sides and is a case of a recirculating flow prompted by the moving the top lid with the other three walls at rest. Lid Driven cavity flow is also considered as a benchmark for testing the numerical efficiency and accuracy of different numerical methods (E. Erturk, 2009).

Number of the research articles pertaining to the study of flow in a driven cavity may fall into two categories, i.e. Physical aspect of the flow and Validation of the newly developed numerical schemes (Chandio, 2013).

Most of the studies for flow dynamics inside the cavity concern the steady state, but very few study the mechanisms of transients until the steady state is achieved (Gustafson, 1991).

For low Reynolds number most of the numerical methods show similar results but as the Reynolds number starts increasing, results are different from each other. Also some research predict that the flow remains steady at high Reynolds number while others show that the flow is unsteady.

Yih-Ferng Penget et al., (2003) investigated transition process in square lid driven cavity from laminar to chaotic flow. They found out that the flow remains steady up to Re=7402 and then as Reynolds...
number is increased to 11000 flow is in transition phase after which it is finally routed to chaotic flow for Reynolds number greater than 11000.

Charles-Henri et al, (2006) through numerical simulations investigated the boundaries of steady state solution for high Reynolds numbers in a square cavity. They observed that steady solution is unstable at Re=10000. They identified a large vortex with two secondary vortices in the bottom left and right corners for Re=1000 and a secondary and tertiary vortices in the bottom left corner.

There has so far been no solid agreement in CFD community over the transient Reynolds number at which flow turns into turbulent.

Zdanski et al, (2003) performed numerical simulation to study the comparison between laminar and turbulent flows over shallow cavities and examined the effect of Reynolds number, aspect ratio and inlet turbulence level on turbulent flow. They observed that with increasing Reynolds number the center of laminar structure of vortex is decreased. On the contrary, the center of both the large and small structure for turbulence case was observed unchanged.

Erturk (2009) studied lid-driven cavity flow using physical, mathematical and numerical methods in detail, and suggested that for very high Reynolds numbers finer grids are important for the resolution of the flow. At Re=7500, quaternary vortex at the bottom left corner was observed which indicates improvement over previous results.

Barragy & Carey (1997) showed calculations for the 2D driven cavity incompressible flow problem. A p-type Finite element scheme for the fully coupled stream function-Vorticity formulation of the Navier-Stokes equations is used. To show the vortex flow features in detail and minimize the impact of corner singularities, graded meshes are used. They observed that up to Re= 12500, steady state solutions can be maintained.

Ghia et al. (1982) used the vorticity-stream function formulation of the 2D incompressible Navier-Stokes equations to study the effectiveness of coupled strongly implicit multi grid (CSI-MG) method in the determination of high-Re fine-mesh flow solutions in square cavity flows. Results are obtained for configurations with Reynolds number as high as 10,000 and meshes size of 257 x 257.

In this paper the attempt is to make a robust comparison over the extent to which three different but major numerical algorithms of solving cavity problem can be applied. In addition, algorithms of solving the governing equations are explained in a more down to detail fashion as of an instructive vessel. With all this, the computer codes and discretization procedure can be supplied for the avid reader upon request. It is noteworthy that the solution with explicit second order accurate numerical scheme has only been superficially reported in the literature to the best of authors’ knowledge.

2. Numerical methods

2.1. Alternating direction implicit method (ADI)

It is usually convenient to adopt the vorticity ($\xi$) transport and stream function ($\psi$) formulation of Navier-Stokes equations for two dimensional plane flows. The governing equations are as follows:

$$\frac{\partial^2 \psi}{\partial x^2} + \frac{\partial^2 \psi}{\partial y^2} = -\xi$$  \hspace{1cm} (1)

$$\frac{\partial \xi}{\partial t} + \frac{\partial (u \xi)}{\partial x} + \frac{\partial (v \xi)}{\partial y} = \nu \left( \frac{\partial^2 \xi}{\partial x^2} + \frac{\partial^2 \xi}{\partial y^2} \right)$$  \hspace{1cm} (2)

In order to solve for the $\xi$ and $\psi$, a numerical approach is required.
Many algorithms exist in the literature, the one method suitable for the condition is alternating direction implicit method (ADI) for vorticity transport equation and point successive over relaxation (PSOR) method for stream function equation. The type of discretization for the vorticity transport equation is forward time central space differences (FTCS) as shown in expression (3). Also, the convective terms have to undergo first order upwinding.

\[
\frac{\zeta_{i,j}^{n+1/2} - \zeta_{i,j}^n}{\Delta t} = \frac{n_{i,j}^n}{\Delta x} \left( \frac{\zeta_{i+1,j}^n - \zeta_{i-1,j}^n}{2\Delta x} \right) + \frac{\zeta_{i,j+1}^n - \zeta_{i,j-1}^n}{2\Delta y}
\]

In ADI method, \( \zeta_{i,j}^{n+1/2} \) is the unknown that is to be explicitly determined from a previous step of \( \zeta_{i,j}^n \) in either of primary directions (initial value of \( \zeta_{i,j}^n \) can be assumed). Then, the entire step is marched towards \( \zeta_{i,j}^{n+1} \) (final solution) using the solution available from a previous time step in the other direction normal to the initial direction, that is, \( \zeta_{i,j}^{n+1/2} \).

The stream function is a Poisson type differential equation which is solved via the SOR algorithm. The finite difference discretization -with 2nd order accuracy -of the equation is written below:

\[
\frac{\Psi_{i+1,j} - 2\Psi_{i,j} + \Psi_{i-1,j}}{\Delta x} - \frac{\Psi_{i,j+1} - 2\Psi_{i,j} + \Psi_{i,j-1}}{\Delta y} = -\zeta_{i,j}^n
\]

The solution is followed by the calculation of residuals and substitution of it in the point successive over relaxation formula to compute the point wise stream function all over the flow domain. For \( \Delta x=\Delta y=\Delta \)

\[
R_{i,j} = \frac{1}{4} \left[ \Psi_{i+1,j} + \Psi_{i-1,j} + \Psi_{i,j+1} + \Psi_{i,j-1} - \Delta^2 \zeta_{i,j} - 4\Psi_{i,j}^n \right]
\]

\[
\Psi_{i,j}^{n+1} = \Psi_{i,j}^n + \omega R_{i,j}
\]

In which \( \omega \) is an over relaxation parameter that accelerates the convergence of the iterations. The value of \( \omega \) is dependent on the mesh size, geometry of the flow domain and the boundary conditions. The optimum value of the \( \omega \) was determined as 1.93 through a different approach.

The method used for determination of \( \omega \) value has been shown in the Figure 1. In this Figure, N is the number of grid nodes in each direction over the flow domain and N=130 was decided after careful observations of numerical independency on grid size for N=130 (more explanation In part 3). It is noteworthy that the curves overlap on top of one another after N exceeds 101, thus the Curve for N=130 has not been shown to avoid confusion while the value of \( \omega \) is absolutely the same as that of N=101.
In Figure 1, error specified on the vertical axis is an overall space average error over the computational domain and is the summation of differences of the exact and numerical solution of the Poisson equation for a variety of the mesh configurations.

2.2. Primitive variables \((u, v, P)\) explicit formulation

The following eqns. are continuity and momentum balance in primary directions in integral form (same as Navier Stocks equations).

\[
\frac{\partial}{\partial t} \int_{cv}^{} u d\mathcal{V} + \int_{cs}^{} u \mathbf{V} dA = \int_{cs}^{} \nabla u . dA - \int_{cv}^{} \frac{1}{\rho} \frac{\partial p}{\partial x} d\mathcal{V} \tag{7}
\]

\[
\frac{\partial}{\partial t} \int_{cv}^{} v d\mathcal{V} + \int_{cs}^{} v \mathbf{V} dA = \int_{cs}^{} \nabla v . dA - \int_{cv}^{} \frac{1}{\rho} \frac{\partial p}{\partial y} d\mathcal{V} \tag{8}
\]

\[
\int_{cs}^{} \mathbf{V} . dA = 0 \tag{9}
\]

In equation 7, first term from left hand side is called the local term (or unsteady term) and the 2nd is convective term. On the right hand side of the equation (eqn. 7), first term is named viscous damping term responsible for resistance to flow and right most term is the source term. The equations are valid for two dimensional laminar flow of an incompressible fluid. Finite volume on uniform rectangular grid with 2nd order accuracy has been used to discretize and integrate the governing equations over a control volume. The control volume has been shown in Figure 2 which shows the staggered discretization with dependent variables on it. The time dependent momentum equations in two primary directions are solved explicitly for the velocity field.

The local time dependent terms are handled in a time marching trend iteratively to reach stability which implies that the discrete solution of \((u-v-P)\) coming from an earlier time step is used in place of the next iteration as long as old and new primitive variables change no more with respect to time pass. By looking at the discretized unsteady term of Navier Stocks equations, same understanding can be achieved where \(u_{n+1}\) is replaced by \(u_n\) in a proceeding time step until convergence.

\[
\frac{\partial}{\partial t} \int_{cv}^{} u d\mathcal{V} = \frac{u_{n+1}^x - u_n^x}{\Delta t} \triangle x \triangle y \tag{10}
\]
To prevent solution instabilities, first order upwinding method has been used for convective terms. Upwinding effectively takes into account the propagation of information on the flow field by either back or forward differencing the convective terms based on flow direction.

Continuity equation has undergone major manipulation to be converted into pressure equation. It is noteworthy that discrete pressure is located at the cell center with velocity sitting all around it to build up a staggered uniform grid as shown in Figure 2.

![Figure 2. The finite volume cell with discrete primitive variables shown on](image)

Again for the pressure equation, SOR algorithm has been used with over relaxation value of 1.7.

### 2.3. Primitive variables \((u, v, P)\) implicit formulation

For the implicit solution of the equations, Ansys Fluent the commercial code (licensed by Middle East Technical University (METU)) was used. When a straight calculation of the dependent variables cannot be made in terms of known quantities, the computation is said to be implicit. The equations under investigation (eqns. 7-8-9) are nonlinear and include the calculation of three unknowns at a time. Despite the fact that implicit formulation scheme corresponds to a set of linear equations which can be solved directly, Fluent makes use of iterative algorithms towards a final solution. Iterations are performed to advance a solution through a sequence of steps from a starting state to final, converged state. Of course, the iteration steps usually do not represent a realistic time-dependent behavior. In fact, it is in this aspect of an implicit method that makes it attractive for steady-state computations, because the number of iterations required for a solution is often much smaller than the number of time steps needed for an accurate transient that asymptotically approaches steady state.

### 3. Boundary conditions

Application of boundary conditions for the cavity problem can be fallen into two types, the stationary walls and the moving lid. As long as the stationary wall is concerned, the no-slip boundary condition has to be implemented.

For the vorticity stream function formulation, boundary condition has to be specified for both stream function and vorticity as such:

#### 3.1. Stationary walls

Stream function takes a constant value on the wall which was adopted as zero in this problem. For vorticity transportation, equation (1) needs to be solved on the wall. By employing proper measures and rearrangements, the boundary condition can be computed as below:

\[
\xi_{\text{wall}}^{n} = \frac{2(\xi_{\text{wall}}^{n} - \xi_{\text{adjacent to wall}}^{n})}{\Delta x \Delta x} + O(\Delta x) 
\]  

(11)
Where \( O(\Delta x) \) is equated to zero since 2nd order accuracy of discretization has been used. The value of vorticity is exact on the wall and no interpolation is required. The boundary condition for the primitive variable formulation is rather different since the staggered grid prevents the coincidence of no-slip boundary condition with the grid itself.

The no-slip boundary condition needs to be imposed as such:

\[
\begin{align*}
\text{u (Next to the wall-Within flow domain)} &= \text{u(Next to the wall-Outside flow domain)} \quad (12) \\
\text{v (Next to the wall-Within flow domain)} &= \text{-v(Next to the wall-Outside flow domain)} \quad (13)
\end{align*}
\]

Equations (12) and (13) are nothing but no-slip conditions at a point (wall) that is located between two grid points and interpolating comes in hand.

### 3.2. Moving wall

For the boundary condition of moving wall, suppose that the cavity lid moves with a constant velocity of \( U_0 \). Again by solving the equation (1) and making use of Taylor series expansion along with certain manipulations following relation can be yielded:

\[
\left( \xi^{n} \right)_{\text{Moving wall}} = \frac{2(\left( \xi^{n} \right)_{\text{wall}} - \left( \xi^{n} \right)_{\text{adjacent to wall}})}{\Delta y \Delta y} - \frac{2U_0}{\Delta y} \quad (14)
\]

For the implementation of the boundary condition on the moving lid in primitive variable formulation, the average of the velocities in the vicinity of the lid must be equated to the velocity of the moving lid.

\[
\begin{align*}
\text{u (Within flow domain)} + \text{u (Outside flow domain)} &= 2U_0 \quad (15) \\
\text{v (At the lid)} &= 0 \quad (16)
\end{align*}
\]

### 4. Grid quality

For determination of grid quality, either previous studies should have been used or a systematic approach can reveal this. Normally, when one is solving a laminar flow there is almost no control over the selection of grid size since rate of strain is linearly related to the shear stress and wall shear stress can be computed linearly with relative ease (\( y+ = u+ \)). The finer the mesh the more details of flow can be captured. The wall functions (used for turbulent flows) which are to include the effect of roughness on the boundary can however be used to identify the best grid quality for a given problem. The trick to bear in mind is that in the turbulent flows the first grid beside the wall must fall in the validity zone of the wall functions (minimum \( y+ = 30 \), fully turbulent region). This is because the wall shear stress and consequent velocity distribution is computed according to that first velocity next to the wall which comes from wall functions.

In this problem, the grid size was chosen as 130 in both directions. From literature, it is known that \( U*/U=0.04 \). If \( U* \) can be known, then the \( y+ = 30 \) can be satisfied. To stay on the safe side the lid velocity was selected as 10 m/s and was replaced into the \( y+ \) relation to yield 133 (or rounded value of 130) number of grids for each direction.
5. Results and discussion

The commercial code Fluent has been used for the solution of the implicit formulation of governing equations. For the solution of vorticity-stream function equations and solution of explicit formulation of primitive variable equations of motion, codes are developed and run in Fortran.

The result of the runs show that no difference exists between the three different numerical solutions up to a limiting Reynolds number of nearly 2500. With increasing the lid velocity above the limiting Reynolds number, explicit numerical solution starts to fall apart of the other two algorithms. This can be seen in Figure 3 where the third vortex has not been identified correctly by the explicit method while the other methods are almost giving identical results. As a matter of fact, the explicit solution has reached its stability edge at this Reynolds number while the third vortex can continuously appear and vanish as the oscillatory simulation goes on. Some technical information can be seen in Table 1 regarding the Re=2500.

<table>
<thead>
<tr>
<th>Numerical Algorithm</th>
<th>Lid force (N)</th>
<th>Iteration number</th>
<th>Convergence criteria</th>
</tr>
</thead>
<tbody>
<tr>
<td>Vorticity-Stream</td>
<td>0.011467</td>
<td>35620</td>
<td>0.000001</td>
</tr>
<tr>
<td>Explicit $u-v-P$</td>
<td>0.011856</td>
<td>59276</td>
<td>0.000001</td>
</tr>
<tr>
<td>Implicit $u-v-P$</td>
<td>0.012401</td>
<td>4048</td>
<td>0.001 &amp; 0.0001</td>
</tr>
</tbody>
</table>

**Table 1.** Details of runs for the different numerical schemes in $Re=2500$

![Figure 3. Comparison of solutions for Re=2500](image-url)
Convergence condition for vorticity stream function solution is to stop iterations when maximum stream function value changes no more than $10^{-6}$ within an iteration.

For explicit primitive variables solution, criteria is to stop iterations when the mean velocity field value changes no more than $10^{-6}$ within an iteration.

Finally, convergence criteria for the fluent was adopted as 0.0001 for x and y momentum equations and 0.001 for continuity equation. These criteria are usually chosen as a rule of thumb for Fluent which are the indication of change in the absolute values of mean variables by no more than 0.0001 and 0.001 within an iteration.

The residuals and convergence state has also been shown in Figure 4 for the Fluent run.

![Residual plot for Re=2500](image)

**Figure 4.** Residual plot for $Re=2500$

After the $Re=2500$, the separation starts to become vivid between the implicit solution and vorticity stream function solution as well. The 3rd vortex next to the back of the lid calculated in ADI (alternating direction implicit) cannot keep in pace with the enlarging speed of that of implicit method. Although the ADI method can handle $Re=5000$ with no signs of instability, the solution is not as accurate as that of the implicit method as well as the benchmark solutions available in the literature. This is due to strength of implicit formulation scheme based on its complexity. Comparison of the mentioned cases are given in Figure 5 and table 2.

![Comparison of solutions for Re =5000](image)

**Figure 5.** Comparison of solutions for Re =5000
Table 2. Details of runs for the different numerical schemes in $Re=5000$

<table>
<thead>
<tr>
<th>Numerical Algorithm</th>
<th>Lid force (N)</th>
<th>Iteration number</th>
<th>Convergence criteria</th>
</tr>
</thead>
<tbody>
<tr>
<td>Vorticity-Stream</td>
<td>0.0573</td>
<td>36695</td>
<td>0.000001</td>
</tr>
<tr>
<td>Explicit $u$-$v$-$P$</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Implicit $u$-$v$-$P$</td>
<td>0.0515</td>
<td>1544</td>
<td>0.001 &amp; 0.0001</td>
</tr>
</tbody>
</table>

It must be noted that the claims of the present paper hold valid as long as grid quality is kept constant as in all three cases discussed ($N=130$). Moreover, the converging criteria is another subject and ought to be kept constant throughout all simulations. On the other hand, it is known according to some literature research that signs of transition to turbulence is observed when Reynolds number reaches 6000 to 8000 and when a grid mesh with less than 257×257 points is used in cavity flow computer simulation, the solution starts to oscillate around Reynolds number range of 7500 to 12500 (Erturk et al., 2005). The same was verified in implicit solution of $Re=7500$ as shown in Figure 6. Although the flow field and number of vortices is similar to the benchmark solutions available in literature, the convergence is suffering a steady state as seen in Figure 6.

Figure 6. Comparison of solutions for $Re=7500$
With all this, Erturk (2006) has increased the Re number to 20000—obtaining a steady state condition—by using a very fine mesh of 1025 in 1025 with a fairly sophisticated numerical method.

Overall, based on observations of the present research it can be claimed that explicit scheme used for primitive variable formulation can be only half the way (as in Re=2500 for explicit to Re=5000 for ADI and implicit schemes) as successful as the other two numerical methods due to its relative simplicity. Then again, the limiting Reynolds number for this method can be raised if finer mesh is used. Comparison of ADI and implicit formulation scheme also reveals that ADI can remain steady with equal Re number of that of implicit formulation, but fails to identify the details of flow field.

6. References


