An Accurate Numerical Method to Predict Fluid Flow in a Shear Driven Cavity

Mohd Rody M. Zin¹, Nor Azwadi C. Sidik²

Abstract – In this paper, the vorticity transport equation is solved to predict the fluid flow in a two-dimensional, shear driven cavity for a wide range of Reynolds numbers and aspect ratio. The advection term in the governing equation is discretised by the constraint interpolated profile method. First, the code is validated for a one-dimensional wave equation and then the results of the flow structure are presented. Several features of the flow, such as the dynamics of the central vortex, the formation of the corner vortices are predicted and compared with the previous findings from other researchers. We found that the vortices structures are significantly dependent on the value of the aspect ratio of the cavity and the dimensionless Reynolds numbers. The predicted results are also shown to be consistent with the experimental study. Copyright © 2009 Praise Worthy Prize S.r.l. - All rights reserved.

Keywords: Advection Equation, CIP Method, Vorticity Transport Equation, Shear Driven Cavity

I. Introduction

Solution to fluid flow by the computational method has received rapid attention in the past few decades and a lot of research attention has been given to this alternative approach due to its flexibility and capability in solving the phenomena at critical conditions, such as high speed flow, flow at microgravity condition, blast furnace, microchannel flow, etc [1]-[5].

The fundamental law of any fluid flow problems is the Navier-Stokes equations, which define any single-phase fluid flow. These equations can be simplified by removing terms describing viscosity to yield the Euler equations. Further simplification, by removing terms describing vorticity yields the full potential equations. Finally, these equations can be linearized to yield the linearized potential equations.

Historically, finite difference method (FDM) [6] was the first computational method used by researchers to solve fluid flow and heat transfer problem by solving Navier-Stokes equation. However, due to the frustration on FDM, which cannot be effectively used on complex geometry, finite element method (FEM) has been introduced in 1950s [7]. In 1980s, finite volume method (FVM) was developed at Imperial College, mainly to solve fluid dynamic problems [8]. Since then, the finite volume method is extensively used to solve transport phenomena problems.

Newly proposed numerical solutions for Navier-Stokes equations are often tested for code validation, on a very well known benchmark problem; the shear-driven cavity flow. Due to its simple cavity geometry, applying a numerical method on this flow problem in terms of computer coding is quite easy and straightforward. However, it is usually very difficult to capture the flow phenomena near the singular points at the corners of the cavity.

There are many numerical solutions have been proposed to solve the Navier-Stokes equations for the case of square shear-driven cavity flow, among others [9-15]. One of the
first studies was achieved by Davis and Mallinson [9]. They have used the method of upwind differencing to approximate the convection term in the three-dimensional cavity. The effect of artificial diffusion has been evaluated for the simulation at high values of Reynolds number. Wright and Gaskell [10] have applied the multigrid strategy for use with the block implicit multigrid method (BIMM). They have presented shear-driven cavity flow results obtained on a $32^2$ grid size for Reynolds number of 100 and 1000.

Ghia et al [11] have used the vorticity-stream function formulation of the two-dimensional incompressible Navier-Stokes equation and investigated the effectiveness of multigrid method for the simulation at high Reynolds number flow situation. Hou et al [12] have applied the particle distribution function approach and reproduced the phenomenon of lid-driven cavity flow problem. They have used 256$^2$ grid points and demonstrate the flow characteristics up to Reynolds number of 7500. Instead vast number of research for flow in a square cavity, only few researchers studied on the shallow cavity and fewer works were devoted to study the flow characteristics at various aspect ratios of the cavity and Reynolds numbers.

In this work, we present the vorticity-stream function simulation of flow inside a 2D shear-driven cavity with different depth-to-width aspect ratios. The advection term in the governing equation is solved using an efficient, high accuracy of the constraint interpolated profile method. The details of the propose method will be discussed in the next section for one- and two-dimensional point of view. Extension to three-dimensions can be easily done and will be discussed in our next report.

II. Mathematical Formulation

For two-dimensional, incompressible and laminar fluid flow, the governing equations can be written in the following continuity equation

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0$$  \hspace{1cm} (1)

which describes the conservation of mass, and the Navier-Stokes equation

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

$$\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} = -\frac{1}{\rho} \frac{\partial p}{\partial y} + \nu \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right)$$  \hspace{1cm} (3)

which describe the conservation of momentum of the fluid. Before considering any numerical solution to the above set of equations, it is convenient to rewrite the equations in terms of dimensionless variables. The following dimensionless variables will be used here

$$X = \frac{x}{L}, \quad Y = \frac{y}{L}$$

$$U = \frac{u}{u_\infty}, \quad V = \frac{v}{u_\infty}$$

$$P = \frac{p}{\rho u_\infty^2 L}$$

In terms of these variables, (2) and (3) become

$$\frac{\partial U}{\partial T} + U \frac{\partial U}{\partial X} + V \frac{\partial U}{\partial Y} = -\frac{1}{Re} \frac{\partial^2 U}{\partial X^2} + \frac{\partial^2 U}{\partial Y^2}$$  \hspace{1cm} (5)

$$\frac{\partial V}{\partial T} + U \frac{\partial V}{\partial X} + V \frac{\partial V}{\partial Y} = -\frac{1}{Re} \frac{\partial^2 V}{\partial X^2} + \frac{\partial^2 V}{\partial Y^2}$$  \hspace{1cm} (6)

The normalized $X$ and $Y$ momentum equations are then combined together to eliminate the pressure terms to yield the normalized vorticity transport equation as

$$\frac{\partial \omega}{\partial T} + U \frac{\partial \omega}{\partial X} + V \frac{\partial \omega}{\partial Y} = \frac{1}{Re} \left( \frac{\partial^2 \omega}{\partial X^2} + \frac{\partial^2 \omega}{\partial Y^2} \right)$$  \hspace{1cm} (7)

where

$$\omega = \frac{\partial V}{\partial X} - \frac{\partial U}{\partial Y}$$  \hspace{1cm} (8)

In terms of stream function, the vorticity equation becomes

Copyright © 2007 Praise Worthy Prize S.r.l. - All rights reserved
\[
\frac{\partial^2 \psi}{\partial x^2} + \frac{\partial^2 \psi}{\partial Y^2} = -\omega
\]  

(9)

From the above derivations, we can see that the conventional evolution equations for flow velocity \(u, v\) and pressure \(p\) were transformed into dimensionless stream function and vorticity transport equations. There are many numerical solutions have been proposed to solve the above set of equations [13]-[16]. However, the most common characteristics of the available method is: high order of numerical accuracy requires high mesh resolution. In the current study, we deviate from this common sense and propose third order numerical accuracy from the interpolation between two node points. The details of the proposed method will be discussed in the next section for one- and two-dimensional point of view.

II.1. Constraint Interpolated Profile (CIP) Method

In this section, we begin by recalling Eqn. (7) and express in one-dimensional form as follow.

\[
\frac{\partial \omega}{\partial t} + U \frac{\partial \omega}{\partial x} = 0
\]  

(10)

Note that we dropped the right hand side of the original equation and Eqn. (10) reduces to a pure wave equation without source term. The theoretical solution of (10) is obtained by shifting a profile

\[
\omega(X, T + \Delta T) = \omega(X - U\Delta T, T)
\]  

(11)

which represents a simple translation of profile with constant velocity \(U\). If we differentiate (11) with spatial variable \(X\), then we get

\[
\frac{\partial \omega}{\partial t} + U \frac{\partial \omega}{\partial x} = 0
\]  

(12)

where \(\omega_x = \partial \omega / \partial x\). Equation (12) coincides with (10) and represents the translation of \(\omega_x\) with velocity \(U\). The novel idea behind this approach is, we trace the time evolution of both \(\omega\) and \(\omega_x\) using (12) and (10) and the profile at each points after one step is specified according to (12). With this restriction, we can greatly reduce the numerical diffusion when we construct the profile.

In this method, the spatial quantities in the point interval are approximated with constrained polynomial using \(\omega\) and \(\omega_x\) at neighbouring points as follow

\[
F_i(X) = a_i X^3 + b_i X^2 + \omega_{x,i} X + \omega_i
\]  

(13)

where \(X = X_i\). The coefficients \(a\) and \(b\) are then determined so that the interpolation function and its first derivatives are continuous at both ends. As a result, we have

\[
a_i = \frac{(\omega_{x,i} + \omega_{x,i+1}) - 2(\omega_{x,i} - \omega_{x,i+1})}{\Delta X^2}
\]  

(14)

\[
b_i = \frac{2(\omega_{x,i} + \omega_{x,i+1}) - 3(\omega_{x,i} - \omega_{x,i+1})}{\Delta X^2}
\]  

(15)

where \(\Delta X = X_i - X_{i+1}\). Once \(F_i(X)\) is determined for all point intervals, the spatial derivatives are calculated as

\[
F_{x,i}(X) = (3a_i X_i + 2b_i) + \omega_{x,i}
\]  

(16)

After all, the advected profile is given by

\[
\omega^{n+1}_i = F_i \left( X_i + \xi \right) = 3a_i \xi^2 + 2b_i \xi + \omega_{x,i}
\]  

(17)

where \(\xi = -U\Delta T\) and the superscript \(n\) indicates the time evolution.

In order to demonstrate the efficiency of the proposed approach, we firstly apply the method to predict the propagation of a square wave. Fig. 1 shows the comparisons of results when the wave moves to a new location from its initial position predicted by the proposed method, Lax-Wendroff [17] and first order upwind method.
From Fig. 1, we can see that the proposed method gives the best accuracy compared to other well-known numerical solutions to wave equation.

II.2. CIP Method to Two-Dimensional Stream Function-Vorticity Equation

In order to solve two-dimensional stream function-vorticity equation, equation (7) and its spatial derivatives are split into advection and non-advection phases as follow

Advection phase:

\[
\frac{\partial \omega}{\partial T} + U \frac{\partial \omega}{\partial X} + V \frac{\partial \omega}{\partial Y} = 0
\] (18)

\[
\frac{\partial \omega}{\partial T} + U \frac{\partial \omega}{\partial X} + V \frac{\partial \omega}{\partial Y} = 0
\] (19)

\[
\frac{\partial \omega}{\partial T} + U \frac{\partial \omega}{\partial X} + V \frac{\partial \omega}{\partial Y} = 0
\] (20)

Non-advection phase:

\[
\frac{\partial \omega}{\partial T} = \frac{1}{Re} \left( \frac{\partial^2 \omega}{\partial X^2} + \frac{\partial^2 \omega}{\partial Y^2} \right)
\] (21)

\[
\frac{\partial \omega}{\partial T} = \frac{1}{Re} \left( \frac{\partial^2 \omega}{\partial X^2} + \frac{\partial^2 \omega}{\partial Y^2} \right) - \frac{\partial \omega}{\partial X} \frac{\partial U}{\partial X} - \frac{\partial \omega}{\partial Y} \frac{\partial V}{\partial Y}
\] (22)

\[
\frac{\partial \omega}{\partial T} = \frac{1}{Re} \left( \frac{\partial^2 \omega}{\partial X^2} + \frac{\partial^2 \omega}{\partial Y^2} \right) - \frac{\partial \omega}{\partial X} \frac{\partial U}{\partial X} - \frac{\partial \omega}{\partial Y} \frac{\partial V}{\partial Y}
\] (23)

Here, we consider a non-constant fluid flow velocity. In computational at two-dimensional space, the spatial quantities in the points interval are approximated as follow

\[
F_{i,j}(X,Y) = \left[ (a_1X + a_2X + a_3)X + a_4X + \omega_{y,j} \right] X
\]

\[
+ \left[ (a_5X + a_6X + a_7)Y + \omega_{x,i} \right] Y
\] (24)

where

\[
a_1 = \frac{-2d_i + (\omega_{y,i,j} + \omega_{x,i,j})}{\Delta X^4}
\]

\[
a_2 = \frac{a_8 - d_i \Delta X}{\Delta X^3 \Delta Y}
\]

\[
a_3 = \frac{3d_i + (\omega_{y,i,j} + 2\omega_{x,i,j}) \Delta X}{\Delta X^4}
\]

\[
a_4 = \frac{-a_8 + d_i \Delta X + d_i \Delta Y \Delta X}{\Delta X \Delta Y}
\]

\[
a_5 = \frac{-d_i + (\omega_{y,i,j} - \omega_{x,i,j}) \Delta Y}{\Delta Y^3}
\]

\[
a_6 = \frac{a_8 - d_i \Delta Y}{\Delta X \Delta Y^2}
\]

\[
a_7 = \frac{3d_i - (\omega_{y,i,j} + 2\omega_{x,i,j}) \Delta Y}{\Delta Y^4}
\]

\[
a_8 = \omega_{x,i,j} - \omega_{y,i,j} - \omega_{x,i,j} - \omega_{x,i,j}
\]

here \( d_i = \omega_{x,i,j} - \omega_{y,i,j} \) and \( d_j = \omega_{x,i,j} - \omega_{y,i,j} \). The spatial derivatives are calculated as

\[
F_{x,i,j}(X,Y) = (3a_1X + 2a_2X + a_3)X + (a_4 + a_6X)Y + \omega_{y,i,j}
\] (26)

\[
F_{y,i,j}(X,Y) = (a_2X + a_3)X + (3a_5 + 2a_6X + 2a_7)Y + \omega_{x,i,j}
\] (27)

In two-dimensional case, the advected profile is approximated as follow

\[
\omega_{x,j}^* = F_{x,j}(X + \xi, Y + \eta)
\] (28)

\[
\omega_{y,j}^* = F_{x,j}(X + \xi, Y + \eta)
\] (29)

\[
\omega_{x,i,j}^* = F_{y,i,j}(X + \xi, Y + \eta)
\] (30)

where \( \xi = -U \Delta T \) and \( \eta = -V \Delta T \).

III. Numerical Results

In this section, we begin with the validation
of code written in FORTRAN language for the present method. For this purpose, we carried out prediction of fluid flow in a square cavity driven by shear force at the top boundary. This type of flow configuration has been used as a benchmark problem for many numerical methods due to its simple geometry and complicated flow behaviours. It is usually very difficult to capture the flow phenomena near the singular points at the corners of the cavity.

In the simulations, three values of Reynolds number, 100, 400, 1000 and 3200 were set up defined by the height of the cavity and constant velocity of the top lid of the cavity. Benchmark solutions provided by Ghia et al. [10] were brought in for the sake of results comparison. Fig. 2 and 3 show the plotted velocity profile extracted at the mid-width and the mid-height of the cavity.

![Fig. 2. Comparisons of horizontal velocity profiles at steady state between the present method and Ghia et al.](image)

![Fig. 3. Comparisons of vertical velocity profiles at steady state between the present method and Ghia et al.](image)

![Fig. 4. Plots of streamline function at Re = 500 for aspect ratio of (a) 0.2 (b) 0.4 (c) 0.6 and (d) 0.8.](image)
Fig. 5. Plots of streamline function at $Re = 1000$ for aspect ratio of (a) 0.2 (b) 0.4 (c) 0.6 and (d) 0.8.

Fig. 6. Plots of streamline function at $Re = 3000$ for aspect ratio of (a) 0.2 (b) 0.4 (c) 0.6 and (d) 0.8.
They may be observed that the results computed from the present method are in excellent agreement with the benchmark solution for wide range of Reynolds numbers.

Once we obtained confidence in our proposed method, we then extend numerical simulation for the shear-driven cavity flow at shallow conditions. Three different values of aspect ratio defined by the height over width of the cavity were performed. We then demonstrate the results in terms of streamlines plots in the cavity.

Fig. 4 shows that for the simulation at $Re = 500$ and low aspect ratio, the primary vortex appears at the center height of the cavity and near to the right wall. Two equal sizes of vortices are formed near the bottom left and right corner of the cavity. As we increase the aspect ratio, the left corner vortex grows faster than the right corner vortex as shown in Figs. 4b and 4c. The growth of the right corner vortex is retarded due to the formation of main vortex and compresses this corner vortex. As the main vortex moves to the center of the cavity (Fig. 4d), the size of the two corner vortices appears to be almost the same since equal space available for growth.

For the simulation at higher Reynolds number ($Re = 1000$ and $3200$), Fig. 5 and 6 demonstrate that, when we increase the aspect ratio, the left corner vortex growth faster and faster and larger maximum size before it is then compressed by the central vortex. All of these findings agree well with the results readily published in the literature [18]-[20].

IV. Conclusion

In this paper, the fluid flow characteristics in a shear-driven cavity have been investigated by solving the vorticity transport equations. The advection term in the governing equation has been solved by the constraint interpolated profile method with third order accuracy in space. Results of these simulations show that, almost all the physical detail of this kind of flow at wide range of Reynolds numbers and aspect ratio of the cavity are reproduced. Specifically, the dynamics of the central vortex and the formation corner vortices are well captured by these simulations. Future efforts need to extend the current formulation to three dimensions and investigate various types of fluid flow related to real engineering problems.

Acknowledgements

The authors would like to acknowledge Universiti Teknikal Malaysia Melaka and Universiti Teknologi Malaysia for supporting this work.

References


1Department of Automotive, Faculty of Mechanical Engineering, Universiti Teknikal Malaysia Melaka, Malaysia.
2Department of Thermo Fluid, Faculty of Mechanical Engineering, Universiti Teknologi Malaysia, Malaysia

**Mohd Rody M. Zin** was born in Kelantan on 29th May 1981. He received Master of Engineering degree (2009) from Universiti Teknologi Malaysia, Malaysia. His current interest includes computational fluid dynamics, internal combustion engine and numerical methods. Mr. Mohd Rody is a lecturer at Department of Automotive, Universiti Teknikal Malaysia Melaka, Malaysia.

**Nor Azwadi C. Sidik** was born in Kelantan on 23rd September 1977. He received Ph.D degree (2010) from Keio University, Japan His current interest includes computational fluid dynamics, numerical methods and fluid structure interaction. Dr Nor Azwadi is a senior lecturer at Department of Thermo Fluid, Universiti Teknikal Malaysia, Malaysia.