Cubic-Interpolated-Pseudo-Particle Method to Predict Dynamic Behaviour of Fluid in Shear Driven Cavity

S.M.R. Attarzadeh, C.S. Nor Azwadi and F. Haghbin

1, 2Department of Mechanical Engineering Universiti Teknologi Malaysia (UTM), 81310, Skudai, Johor, Malaysia
3Department of Computer Science Universiti Teknologi Malaysia (UTM), 81310, Skudai, Johor, Malaysia

1, 2e-mail: smraniaki@live.utm.my, 3e-mail: haghbin@live.utm.my

Keywords: CFD; Cubic Interpolated Pseudo Particle; Finite difference; Navier Stokes equation, Lid-Driven cavity.

Abstract. In this paper, Cubic Interpolated Pseudo Particle Method is proposed to investigate the dynamic behavior of fluid motion in shear lid-driven cavity. The CIP scheme is individually performed to observe the behavior of the fluid motion at varying Reynolds numbers of 100, 400 and 1000. Comparison of the achieved results with the experimental results approves the capability of CIP to establish the sophistication of fluid structure in the system. Although the achieved trajectory had slightly difference but it was almost following the same pattern published in the literatures. The most advantage of this method is that it aims to accelerate processing time as well as higher exponent of accuracy.

Introduction

Looking widely, we see that rotating fluid has number of industrial application and how to analyze dynamic behavior of this fluid may be a privilege to get about its application. In this paper our main concentration relies on lid-driven cavity flow which most of researchers doing so, since it is believed that results of these simulations not only are more efficient but even the numerical achievements can be directly compared. In contrast, variation of Reynolds number is quite independent of boundary condition changes since it stays unchanged. Moreover, fluids in lid-driven cavity almost exhibit all behaviors of dynamic motion in incompressible flows such as eddies, instabilities and turbulences.

Till today researchers preferred to follow their investigation based on computational approach rather than experimentally to realize the dynamic behavior of fluid in different conditions. Although Numerical analysis is an approximate approach and can never enhance entire accurate result, but investigating alternative methods may raise the chance of obtaining results with higher order of accuracy and precision.

To the best of author’s knowledge, so many had done researches on investigation of fluid dynamic in different conditions using different methods. Gia [1] with the aim of high-resolution multi grid mesh structure derived a solution for incompressible fluids using the Navier stoke equation for high Reynolds. Kosinski and Christian [2] worked on how simulation of driven flow of an incompressible fluid in a cavity. Deshpande and Shankar [3] give a comprehensive study on the numerical and experimental work related to this type of flows.

In this regard so many had done numerical investigation of the same concept but in different cavity cross sections. For instance, some works reported in rectangular cavity rather than square cavity. As an example Cheng and Hung[3] two dimentional vortex structure analysis in a rectangular lid driven using Lattice Bolzman had reported results with different aspect ratios and varying Reynold numbers. In other side, Povitsky[4] took 3D cavity to aim his research and investigating dynamic behaviour of fluid where it moves along diagonal of cavity. Tsai and Sheu [5] 3D present. As mentioned above, many researchers using driven cavity flows to validate their


A glamorous point is the experimentally reports revealed for analysis of cavity flow by Migeon et al. [24] is where they investigate the time-dependent laminar flow in cavities where one wall moves after a sudden start.

Ease of Use

Investigating the behavior of fluid motion may have various industrial applications. For this aim, direct numerical solution is offered to visualize the propagation of fluid within cavities. How it propagates within sophisticate cavity geometry may cause a heavier numerical calculation. Here, an alternative numerical solution is offered to both reduce the mass of computations (less processing core) and raise the exponent of accuracy respectively.

The lid-driven cavity problem being used long for validation of newly code generated and alternative numerical solutions. Meanwhile the geometry is both simple and preferable for numerical 2D simulation. The ideal case is when the fluid is contained in a square cavity with drichlet boundary: 3 stationary sides and one moving side.

Mathematical calculations

**Governing equations.** For a 2D, the governing equation in incompressible and laminar fluid flow can be written in the following form:

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

(1)

Which can best describe the Navier stokes equation.
\[
\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = -\frac{1}{\rho} \frac{\partial P}{\partial x} + \nu \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right)
\]

(2)

\[
\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} = -\frac{1}{\rho} \frac{\partial P}{\partial y} + \nu \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right)
\]

(3)

Whereas Navier stokes relation imply, \( u \) and \( v \) terms are respectively the velocity along \( x \) and \( y \) directions and \( P \) is the fluid density. This relation describes the conservation of momentum of the fluid. To solve this equation it is applied the so-called stream function-vorticity to eliminate number of unknown and define new inter related coordinate between parameters in the main equation. Here it had eliminated the term of pressure by taking both derivatives with respect to \( x \), \( y \) and finally subtracting the two equations.

\[
\frac{\partial (\psi_x - \psi_y)}{\partial x} + \frac{\partial (\psi_y + \psi_x)}{\partial y} + \frac{\partial \psi_x}{\partial x} + \frac{\partial \psi_y}{\partial y} u \left[ \frac{\partial \psi_y}{\partial x} - \frac{\partial \psi_x}{\partial y} \right] + \frac{\partial \psi_y}{\partial x} \frac{\partial \psi_x}{\partial y} = 0
\]

(5)

By the help of vorticity function \( \omega = \frac{\partial \psi_y}{\partial x} - \frac{\partial \psi_x}{\partial y} \) combining with continuity equation while above equation can be re-written in this form:

\[
\frac{\partial \omega}{\partial t} + \frac{\partial \omega}{\partial x} + v \frac{\partial \omega}{\partial y} = \frac{\partial \psi}{\partial x} + \frac{\partial \psi}{\partial y}
\]

(5)

In the term of stream function, \( u = \frac{\partial \psi}{\partial y} \) and \( v = -\frac{\partial \psi}{\partial x} \), the equation defining the vorticity becomes

\[
\omega = \left( \frac{\partial \psi}{\partial x} + \frac{\partial \psi}{\partial y} \right)
\]

(6)

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 as;

\[
\Psi = \frac{\psi}{u_x H} \quad \Omega = \frac{\omega H}{u_x} \quad \chi = \frac{x}{H} \quad \gamma = \frac{y}{H} \quad \tau = \frac{t H}{u_x} \quad U = \frac{u}{u_x} \quad V = \frac{v}{u_x} \quad Re = \frac{U H}{\nu}
\]

(7)

We can rewrite equation (5) and (6) in (7) terms of above conventions:

\[
\frac{\partial \Omega}{\partial \tau} + U \frac{\partial \Omega}{\partial \chi} + V \frac{\partial \Omega}{\partial \gamma} = \frac{1}{Re} \left( \frac{\partial^2 \Omega}{\partial \chi^2} + \frac{\partial^2 \Omega}{\partial \gamma^2} \right)
\]

(8)

and

\[
\Omega = \left( \frac{\partial \psi}{\partial x} + \frac{\partial \psi}{\partial y} \right)
\]

(9)

From the above conventional definition we have transformed the parameters of pressure \( u \) and \( v \) to dimensionless variables of \( \Psi \) an \( \Omega \), besides, many numerical solution being proposed to solve above set of equations. In this sense, high order of accuracy needs to have high grid concentration. **Constrained interpolated method.** In this method, the spatial quantities in the point interval are approximated with constrained polynomial using \( \Omega \) and \( \Omega \chi \) at neighboring points as follow

\[
F_i(X) = a_i X^3 + b_i X^2 + \Omega_i X + \Omega_i
\]

(10)

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

\[
a_i = \frac{(\Omega_{i+1} - \Omega_i)}{\Delta \chi^2} - \frac{2(\Omega_{i+1} - \Omega_{i-1})}{\Delta \chi^3}
\]

\[
b_i = \frac{(2\Omega_{i+1} + \Omega_{i-1})}{\Delta \chi} - \frac{3(\Omega_{i+1} - \Omega_i)}{\Delta \chi^2}
\]

(11)
In order to demonstrate the efficiency of the proposed approach (CIP), 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 [20] 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.

\[
\begin{align*}
\Omega^i_{n,j} &= F^i_n \left( X + \xi_n, Y + \xi_n \right) \\
\Omega^a_{n,i,j} &= F^a_{n,i} \left( X + \xi_n, Y + \xi_n \right) \\
\Omega^n_{n,i,j} &= F^n_{n,i} \left( X + \xi_n, Y + \xi_n \right)
\end{align*}
\]

**C. CIP Solution to Two-Dimensional Stream Function-Vorticity Equation.**

In order to solve two-dimensional stream function-vorticity equation, Eqn. (8) and its spatial derivatives are split into advection and non-advection phases as follow Advection phase:
Non-advection phase:

\[ \frac{\partial \Omega}{\partial t} + U \frac{\partial \Omega}{\partial x} + V \frac{\partial \Omega}{\partial y} = 0 \]  
(13)

\[ \frac{\partial \Omega}{\partial t} + U \frac{\partial \Omega}{\partial x} + V \frac{\partial \Omega}{\partial y} = 0 \]  
(14)

\[ \frac{\partial \Omega}{\partial t} + U \frac{\partial \Omega}{\partial x} + V \frac{\partial \Omega}{\partial y} = 0 \]  
(15)

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

\[ \frac{\partial \Omega}{\partial t} = \frac{1}{Re} \left( \frac{\partial \Omega}{\partial x} + \frac{\partial \Omega}{\partial y} \right) \]  
(16)

\[ \frac{\partial \Omega}{\partial t} = \frac{1}{Re} \left( \frac{\partial \Omega}{\partial x} + \frac{\partial \Omega}{\partial y} \right) \left( \frac{\partial \Omega}{\partial x} \frac{\partial \Omega}{\partial x} \right) \]  
(17)

\[ \frac{\partial \Omega}{\partial t} = \frac{1}{Re} \left( \frac{\partial \Omega}{\partial x} + \frac{\partial \Omega}{\partial y} \right) \left( \frac{\partial \Omega}{\partial x} \frac{\partial \Omega}{\partial y} \right) \]  
(18)

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

\[ F_{ij}(X,Y) = \left\{ (a1X + a2Y + a3)X + a4Y + \Omega_{x,i} \right\} Y + \left\{ (a5Y + a6X + a7)Y + \Omega_{y,i} \right\} X \]  
(19)

Where;

\[ a1 = \frac{-2d + (\Omega_{x,i+1} + \Omega_{x,i-1})}{\Delta X^2} \]
\[ a2 = \frac{a8 + d, \Delta X}{\Delta X \Delta Y} \]
\[ a3 = 3d + (\Omega_{x,i+1} + 2\Omega_{x,i+1}) \Delta X \]
\[ a4 = \frac{-a8 + d, \Delta X + d, \Delta Y \Delta X}{\Delta X \Delta Y} \]
\[ a7 = \frac{3d}{\Delta Y^2} \]
\[ a8 = \Omega_{x,i} - \Omega_{x,i+1} - \Omega_{x,i+1} + \Omega_{x,i+1} \]  
(20)

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

\[ \Omega_{x,i} = F_{ij}(X + \xi_x, Y + \xi_y) \]  
(21)

\[ \Omega_{y,i} = F_{ij}(X + \xi_x, Y + \xi_y) \]  
(22)

\[ \Omega_{x,i} = F_{ij}(X + \xi_x, Y + \xi_y) \]  
(23)

where \( \xi_x = -U \Delta T \) and \( \xi_y = -V \Delta T \)

The calculated spatial quantities are then be used to solve non-advection phase of Eqns. (21) to (23) and vorticity formulation of Eqn. (9). In present study, we apply the central finite different Discretization method with second order accuracy in time and space. Details of the Discretization techniques will not be shown here and can be found in Ref. [25].

**Numerical Results and discussion**

We carried out prediction of fluid flow driven by shear force in a square cavity at the top boundary. It is usually very difficult to capture the flow in singular points. This type of problems has been used as a benchmark problem due to simple geometry and complicated flow. This type of flow configuration has been used as a benchmark problem for many numerical methods due to its simple geometry and complicated flow behaviors. 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 and 1000 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 [1] were brought in for the sake of results comparison.
Fig. 2 show plots of stream function for the Reynolds numbers considered. It is apparent that the flow structure is in good agreement with the previous work of Ghia et al. [22]. For low Re (Re=100), the center of vortex is located at about one-third of the cavity depth from the top. As Re increases, the primary vortex moves towards the center of cavity and increasing circular. In addition to the primary, a pair of counter rotating eddies develop at the lower corners of the cavity.

Simulation:

After applying three value of Aspect ratios(\(\frac{\Delta x}{\Delta y}\)) and running simulation according to variation of Reynolds number we may observe how close would be the results to the bench mark results published by Gia[1].

Fig. 3. Streamlines plots at Aspect Ratio 2/3
Conclusion

As described, we had offered an alternative numerical solution to solve Navier-Stokes equation to simulate the dynamic behavior of fluid in lid driven square cavity. Transformation of conventional momentum equation into stream function took place to initiate providing a solution using constrained interpolated method. Our predictions based on various aspect ratios demonstrated entire matching with all placed in literature review. This model can easily extend into just about divers schematic of three-dimensional and may aim the field of scientific computing.

References


Fig. 4. Streamlines Plots at Aspect Ratio 1/2

Fig. 5. Streamlines plots at Aspect Ratio 2/3
Mechanical and Aerospace Engineering
10.4028/www.scientific.net/AMM.110-116

Cubic-Interpolated-Pseudo-Particle Method to Predict Dynamic Behaviour of Fluid in Shear Driven Cavity
10.4028/www.scientific.net/AMM.110-116.377