A NUMERICAL ANALYSIS OF TWO DIMENSIONAL LID DRIVEN CAVITY FLOW USING CUBIC INTERPOLATED PSEUDO-PARTICLE METHOD

IDRIS BIN MAT SAHAT

A project report submitted in partial fulfilment of the
requirements for the award of the degree of
Master of Engineering (Mechanical)

Faculty of Mechanical Engineering
Universiti Teknologi Malaysia

APRIL 2010

ABSTRACT

The Finite Different Navier Stokes equation (FDNSE) and Cubic Interpolated Pseudo-Particle Navier Stokes equation (CIPNSE) are applied to simulate the 2-D square lid driven cavity flow of water at Reynolds numbers 100, 400 and 1000. CIPNSE scheme solves hyperbolic type equations which is the vorticity transport equation In the CIPNSE, the gradient and the value of the vorticity at the nodes is determined and the stream function is then determined using the vorticity equation. It is discovered that the numerical simulation of CIPNSE provided a very good agreement with the established ‘benchmark’ results by Ghia et. al. but is slightly less accurate compare to FDNSE. The flow structure of 2-D shallow lid driven cavity flow of water at Reynolds numbers has been studied numerically using CIPNSE and compared with FDNSE and Ghia result which show good agreement. The location of the primary and secondary vortex also settled at the same place. For shallow cavity flow, both streamline pattern using FDNSE and CIPNSE show concluding remarks compare to CIPLBM. On top of that, the accuracy of CIPNSE and FDNSE are compared based on Ghia result within the midsection velocity profile where minor differences of accuracy are demonstrated for CIPNSE. On the other hand, both midsection velocity for CIPNSE and FDNSE show tiny differences compare to FDNSE and Constrained Interpolation Profile Lattice Boltzmann Method (CIPLBM). Yet CIPNSE is practical in simulating two dimensional lid driven cavity.

ABSTRAK

Pembeza terhingga persamaan Navier Stokes (FDNSE) dan sisipan semata-padu Pseudo-Zarah persamaan Navier Stokes (CIPNSE) digunakan untuk mensimulasikan aliran di dalam rongga persegi dua dimensi dengan penutup bergerak pada nombor, Reynolds 100 400 dan 1000. Teknik CIPNSE menyelesaikan persamaan jenis hiperbolik yang menyelesaikan persamaan pengangkutan pusaran. Dalam CIPNSE, kecerunan dan nilai pusaran pada setiap nod dapat dicerap dan fungsi arus kemudian ditentukan dengan menggunakan persamaan pusaran. Hasilnya, simulasi berangka CIPNSE memberikan jawapan yang hampir sama simulasi Ghia et. al. Walaubagaimanapun, ianya sedikit kurang tepat berbanding dengan FDNSE. Struktur aliran tutup cetek 2-D aliran rongga didorong air pada nombor Reynolds secara berangka menggunakan CIPNSE dan dibandingkan dengan keputusan FDNSE dan Ghia yang menunjukkan kesesuaian yang baik. Lokasi pusaran primer dan sekunder juga didapati di tempat yang sama. Untuk aliran rongga cetek juga, pola garis arus yang terhasil oleh penggunaan FDNSE dan CIPNSE menunjukkan hasil yang lebih baik jika dibandingkan dengan CIPLBM (kaedah isipadu terkekang Lattice Boltzmann). Selain itu, ketepatan bandingan CIPNSE dan FDNSE berdasarkan keputusan Ghia dalam profil kelajuan di kawasan tengah menunjukkan sedikit perbezaan ketepatan jika menggunakan CIPNSE. Dari aspek lain, baik kelajuan kawasan tengah untuk CIPNSE dan FDNSE menunjukkan perbezaan kecil berbanding dengan FDNSE dan CIPLBM. Sesungguhnya, CIPNSE bersifat praktikal dalam simulasi kedua dimensi rongga tudung didorong.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF ABBREVIATIONS

CIP	-	Cubic Interpolated Pseudo-particle
CFD	-	Computational Fluid Dynamics
PDE	-	Partial Differential Equation
NSE	-	Navier-Stokes Equation
LBM	-	Lattice Boltzmann Method
FDM	-	Finite Difference Method
FEM	-	Finite Element Method
FVM	-	Finite Volume Method
FDNSE	-	Finite Difference Navier-Stokes Equation Method
CIPNSE	-	Cubic Interpolated Pseudo-particle Navier-Stokes Equation Method
CIPLBM	-	Constrained Interpolated Profile Lattice Boltzmann Method

LIST OF SYMBOLS

$\boldsymbol{AR}$	-	Aspect Ratio
H	-	Height of cavity
p	-	Pressure
ρ	-	Density
Re	-	Reynolds Number
t	-	Time
T	-	Dimensionless time
u	-	Velocity in x direction
u_∞	-	Lid velocity
U	-	Dimensionless velocity in x direction
v	-	Velocity in y direction
V	-	Dimensionless velocity in y direction
W	-	Width of cavity
x	-	Axial distance
X	-	Dimensionless axial distance
y	-	Vertical distance
Y	-	Dimensionless vertical distance
μ	-	Dynamic viscosity
ν	-	Kinematic viscosity
ω	-	Vorticity
Ω	-	Dimensionless vorticity
ψ	-	Stream function
Ψ	-	Dimensionless stream function

Superscript

n	-	Current value
$n+1$	-	Next step value
$*$	-	Non advection phase value

Subscript

i	-	x direction node
j	-	y direction node
$max\ i$	-	x direction maximum node
$max\ j$	-	y direction maximum node
∞, e	-	Free stream condition

LIST OF APPENDICES

CHAPTER 1

INTRODUCTION

1.1 Background

Mankind and would fall into great disaster and catastrophe to humanity if the human best friend, computer, is suddenly disappear. Since the first computer was built on in the early 20th century, human life changes exponentially. In the earlier era of computer invention, computer was as big as house and the performance were very slow. But thanks to IBM and APPLE, the companies that have developed the computer up until today as the size, the weight and the performance are terrifically boosted. With the latest version of computer, people finish their job within few minutes rather than spending days or months doing tedious work and recursive task long before the existence of computer. Researchers and accountant would be the happiest person to be using the computer as their working time can be reduced greatly and they have more time to do more calculations and researches.

Centuries before the existence of the computer, researcher only relies on experimental data to understand the behaviour of fluid flow and come out with many correlations, and such worthy correlations was the famous and widely-used Reynolds Number, Re which was discovered after hundreds of successful experiment. The experiments were conducted by Osborn Reynolds in the 1880s producing the establishment of the dimensionless Reynolds number, Re**,** as the key parameter for the determination of the flow regime in pipes, whether turbulent or not [1]. The next most successful experiment ever achieves back in few decades was the airplane which was invented in 1903. Oliver and Wilbur Wrights was the team that successfully lead the world into a new dimension and those victories were achieved after thousands of worthy experience and experiment. Obviously, the most difficult part of an experiment is to get a successful data which require a large number of experiments. The result from experiment is very promising because it is the actual thing that is really happening. Somehow, it is difficult when conducting an experiment since the preparation of the instrumentation and devices is tedious if it does not follow the instruction in a proper manner.

As the world developed, computers were also improved. CFD or Computational Fluid Dynamic is one of the applications which introduced by the computer. CFD tremendously facilitate the cracking of fluid flow problem by presenting the real problem in an abundance of knowledge that would be really useful to replicate the real fluid flow problem. Many simulations were done using the CFD and have been a great help for engineers and scientists. CFD is easier to be implemented rather than conducting an experiment that is very expensive and time consuming. Hence, the use of CFD is undeniable nowadays and it always yield good results if the formulation, especially for the numerical simulation was correctly selected and evaluated. More research were done using computer and vast type of numerical method were implemented using computer.

Overall, the fluid dynamic solution can be divided into three major divisions which are purely from experiment where many correlations were developed. The second division is completely theoretical which most of the fluid problem has its own assumption and mathematical equation that will lead to analytical solutions. This division is not practical for complex Partial Differential Equation (PDE) especially the Navier-Stokes equation which will be covered later in this chapter. The last and the most recent method in solving the fluid dynamic is the CFD. These divisions can be illustrated in Figure 1.1

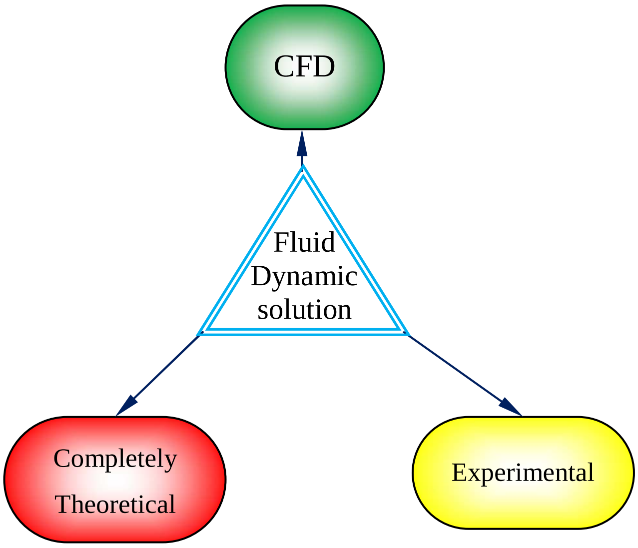

Figure 1.1 Division of fluid dynamics solution

1.2 Computational Fluid Dynamic (CFD)

There are many application of CFD which are tools for research, design, education, Automotive, Sports and many other fields. In this thesis, the focus is based on usefulness of CFD to solve the non linear partial differential equation (PDE) which usually the analytical solution does not exist. Despite that, there are flow with analytical solution were applied with numerical method for validation purposes. The heart of CFD is the famous and unsolvable non-linear incompressible full Navier-Stokes equation.

There are two type of CFD simulation which is the numerical and the other one is ready to use software. The latter one, or in other word, the software for example FLUENT© is very easy to use and infinite type of flow problem with many variables can be easily solved but there are disadvantages such as the user probably doesn’t know to the depth about the formulations that has been applied, the assumptions and a lot more. This software normally used for the practical application which the complicated geometry and conditions. Despite that, this software is based on the numerical method but it is not being revealed. It purposes is solely to reduce the tough part and to make it user friendly.

However, the former type of simulation is very remarkable as the individual who create the codes understand very well the formulation, the assumption, boundary conditions and others. This style of simulation usually applicable for knowledge sharing as many publications spawn everyday with new type of method for example the Lattice Boltzmann method, Bifurcation method and more, claiming the method is among the best through various comparison and validation with the earlier or the classical method. The simulation requires the creator to be well-verse in programming software i.e. FORTRAN, C++, Matlab.

On the other hand, the limitation is the simulation sometimes cannot be applied to complicated geometries, difficult and not very useful for industrial or life application. Nevertheless, both type of simulation if correctly combined, the result would be very reliable as the simulation results almost yields the experimental result for example the simulation of flow over cylinder [2**]** and the experimental [3**].**

1.3 Governing Equation in CFD

In a fluid flow, there are many variables that exist and control the characteristic of the flow. These variables normally depend on the physics of the flow, the nature of the fluid or the surrounding system. The variables that normally occur in a fluid flow are:

- velocity, u
- pressure, p
- temperature, T
- fluid density, ρ
- fluid viscosity, dynamic (μ) and kinematic (ν)

These familiar variables are very important for CFD simulation because it is useful and usually incorporated in a three major general governing equation. These governing equations are the key to CFD and also for heat transfer simulation. It also can be modified depending on the physical flow of the fluid or base on the assumption that can be made. The equations which for incompressible fluid are:

- The continuity equation (conservation of mass)

$$\nabla \cdot \boldsymbol{u} = 0 \tag{1.1}$$

- The Navier-Stokes Equation (conservation of momentum)

$$\boldsymbol{u}_{\mathrm{t}} + \boldsymbol{u} \cdot \nabla \boldsymbol{u} = -\frac{1}{\rho} \nabla \cdot p + \nu \nabla^2 \boldsymbol{u} + \boldsymbol{f} \tag{1.2}$$

- The energy equation (conservation of energy)

$$\boldsymbol{E}_t + \nabla(\boldsymbol{E}\boldsymbol{u}) = \dot{q} - p \cdot \nabla \boldsymbol{u} + \boldsymbol{f} \cdot \boldsymbol{u} \tag{1.3}$$

The first two equations play a major role in providing the formulation which is needed to produce the numerical simulation. These two basic equations will morph to a new equation based on the physical model and it is also different from one and another if the applied numerical method is different.

1.4 Simple introduction to Navier-Stokes equation

Navier-Stokes equation is a well known in fluid dynamic fields. The equation is nonlinear and usually the flows that apply this equation are considered incompressible. Many fluids flow were governed by this equation because in describing the conservation of momentum, the equation almost perfect. In the equation lie the unsteady term, the diffusive term, pressure term, convective term and the external force which is a complete package for momentum conservation. In spite, there is no analytical solution to this equation as there are many Partial Difference term in the equation. Till this thesis were wrote down, this equation still not yet be solved but many type of numerical method were tried out by scientist and engineers and produce their own solution of numerical simulation.

However, there is still exception because some fluid flow having the analytical solution and this exception will be discussed later in the next chapter.

1.5 Lattice Boltzmann Method (LBM)

Lattice Boltzmann Method (LBM) is an alternative method for solving fluid flow. LBM was developed from the Boltzmann's original conceptual view in a mesoscopic way. LBM use particles analysis which the particle is confined to the node of the lattice. Variations in momentary due to velocity directions and magnitudes and varying particle mass are reduced to 8 directions, 3 magnitudes and a single particle mass. Overall, LBM method is focussing more on small particles analysis which is completely different from the classical numerical method in the latter section [4].

1.6 Cubic Interpolated Pseudo Particle (CIP) at a glance

CIP method is rather a new method. This method concentrates on solving the advection term which will be described in later in the second chapter. The fundamental of this method is it not only carries the value of a node, but together with the gradients at that node also. Thus, many information can be stored with small value of nodes.

1.7 Problem Statement

Many classical numerical methods have been applied to solve Navier-Stokes equation over the years. Yet, these numerical still lacking where for higher order of accuracy, more grids are needed to satisfy the methods.

REFFERENCES

1. Y. A. Cengel. *Introduction to Fluid Mechanics*. 5th. ed. Singapore: Mc Graw Hill. 2003
2. K. Lee, K. S. Yang. Flow patterns past two circular cylinders in proximity. *Computers & Fluids* , 2009. 38: 778-788.
3. G. Xu, Y. Zhou. Strouhal numbers in the wake of two inline cylinders. *Experiments in Fluids*, 2004, 37: 248-256.
4. M.C. Sukop, and D.T.Jr Thorne, *Lattice Boltzmann Modelling, An introduction for Geoscientists and Engineers*. 2nd. ed. New York: Springer. 2007.
5. J. Tu, G.H. Yeoh, and C. Liu, *Computational Fluid Dynamics: A Practical Approach*. Elsevier. 2008.
6. E.J. Finnemore, and J.B. Franzini, *Fluid Mechanics with Engineering Application*. Mc Graw Hill. 2002.
7. Grebel, W.P. *Advanced Fluid Mechanics*. Academic Press. 2007
8. Panton, R. L. *Incompressible flow*. USA : John Wiley & Son. 1984.
9. Ghia, U. Ghia, K.N. and Shin, C.T. High-Re solutions for incompressible flow using the Navier–Stokes equations and a multigrid method. *J Computational Physics*, 1982. 48: 367-411.
10. E. Barragy and F. C. Carey. Stream Function-Vorticity Driven Cavity Solutiion using p Finite Elements. *Computers & Fluids,* 1997. 26(5): 453-468.
11. S. Albensoeder, H.C. Kuhlmann, and H.J. Rath. Multiplicity of Steady Two-Dimensional Flows in Two-Sided Lid-Driven Cavities. *Theoret. Comput. Fluid Dynamics*, 2001. 14: 223–241.

12. F. Auteri and L. Quartapelle. Galerkin Spectral Method for the Vorticity and Stream Function Equations. *Journal of Computational Physics*, 1999. 149: 306–332.

13. Tafti, D. Comparison of some upwind-biased high-order formulations with a second-order central-difference scheme for time integration of the incompressible Navier–Stokes equations. *Computers & Fluids* , 1996. 25: 647-665.

14. Botella, O. On the solution of the Navier-Stokes equations using Chebyshev projection schemes with third-order accuracy in time. *Computers and Fluids,* 1997. 26: 107-116.

15. Z. Guo, B. Shi and N. Wang. Lattice BGK Model for Incompressible Navier–Stokes Equation. *Journal of Computational Physics,* 2000. 165: 288–306.

16. C. Migeon, A. Texier and G. Pineau. Effects on Lid-driven cavity shape on the flow establishment phase. *Journal of Fluids and Structures*, 2000. 14: 469-488.

17. Anderson, J.D. Computational Fluid Dynamics: The basics with application. Singapore : Mc Graw Hill. 1995.

18. Matyka, Maciej. Solution to two-dimensional Incompressible Navier-Stokes Equations with SIMPLE, SIMPLER and Vorticity-Stream Function Approaches. Sweeden : University of Linkoping, 2003. 1-15.

19. P.H. Oosthuizen, D. Naylor. *Introduction to Convective Heat Transfer Analysis*. Singapore : Mc Graw Hill. 1999.

20. S., Thomas. *Applied Dimensional Analysis and Modeling*. US: Elsevier, 2007.

21. Weinan, E. and J.G. Liu, Vorticity Boundary Condition and Related Issues for Finite Difference Schemes. Journal of Computational Physics, 1996. 124:368–382 .

22. J.C. Tannehil, D.A. Anderson and R.H. Pletcher. *Computational Fluid Mechanics and Heat Transfer*. 2nd. ed. Taylor & Francis. 1984.

23. H. Takewaki, A. Nishigushi and T. Yabe. Cubic Interpolated Pseudo-particle Method (CIP) for solving Hyperbolic type equations. *Journal of Computational Physics*, 1985. 61:261-268

24. T. Yabe, K. Takizawa, M. Chino, M. Imai and C.C. Chu. Challenge of CIP as a universal solver for solid, liquid and gas. *Int. J. Numer. Meth. Fluids*, 2005. 47: 655–676.

25. T. Yabe and T. Aoki. A universal solver for hyperbolic-equations by cubic-polynomial interpolation. I. One dimensional solver. *Computer Physics Communication*, 1991. 66: 219-232.

26. T. Yabe, T. Ishikawa, P.Y. Wang, T. Aoki, Y. Kadota and F. Ikeda. A universal solver for hyperbolic-equations by cubic-polynomial interpolation. II. Two-and three-dimensional solvers. *Computer Physics Communication*, 1991. 66: 233-242.

27. Y. Ogata, T. Yabe and K. Odagaki. An Accurate Numerical Scheme for Maxwell Equation with CIP-Method of Characteristics. *Communication in Computational Physics*, 2006. 1: 311-335.

28. T. Yabe and P.Y. Wang. Unified Numerical Procedure for Compressible and Incompressible Fluid. *Journal Physics of Society (JAPAN)*, 1991. 60: 2105-2108.

29. H. Mizoe , S.Y. Yoon , M. Josho and T. Yabe. Numerical simulation on snow melting phenomena by CIP method. *Computer Physics Communications*, 2001. 135: 154–166.

30. D. Barada, T. Fukuda, M. Itoh and T. Yatagai. Cubic interpolated propagation scheme in numerical analysis of lightwave and optical force. *OPTICS EXPRESS*, 2006.

31. S. Agus. *An accurate Constrained Interpolated Profile for solving BGK Boltzmann Equation*. Master thesis, Universiti Teknologi Malaysia; 2009