NUMERICAL COMPUTATION OF LID DRIVEN CAVITY USING MARKER AND CELL (MAC) METHOD

NURUL IZYAN BINTI MAT DAUD

A thesis submitted in fulfillment of the requirements for the award of the
Master of Science (Engineering Mathematics)

Faculty of Science
Universiti Teknologi Malaysia

JANUARY 2014

To my beloved parents,

Mat Daud Bin Hamat and Wan Azizah Binti Wan Ibrahim,

my supervisor, Dr. Zuhaila Binti Ismail and my friends.

ACKNOWLEDGEMENTS

First of all, thanks to Almighty Allah s.w.t. for graciously bestowing me the perseverance to undertake this research. I sincerely have to thank all those who have supported and helped me writing this report. A special thanks and a deepest appreciation to my supervisor Dr. Zuhaila Ismail, who taught me a great deal throughout the course of this study, enabled me to see things more clearly, encouraged and helped me at any time needed.

I also would like to extend my sincere appreciation to my entire friend, who kindly provided valuable and helpful comments in the preparation of the thesis and to those who have involved directly or indirectly in the preparation of this thesis, whom I have not mentioned above.

ABSTRACT

The numerical analysis of lid driven in a square cavity flow is discussed. The two dimensional, incompressible, time dependent Navier-Stokes equations are investigated. The governing equations which contains continuity and momentum equations are derived from conservation of mass and conservation of momentum. The governing equations are first transformed into nondimensional equations. These equations are solved numerically using Marker and Cell (MAC) method in conjunction with fractional step method. Next, the results plotted using Matlab. Numerical results presented include the velocity profiles, pressure and vorticity contours. The velocity profiles is compared with [1]. The results showed that MAC method is suitable for solving the problem since the error is not quite difference. Different Reynolds number are used causing perturbation on vorticity contours. Then, the problem is extended by adding a velocity at the bottom wall of cavity. The pattern of the flow fluid is changed when different direction of velocities is applied to the problem. Linear stability condition must be obeyed in choosing Reynolds number. The smaller the Reynolds number gives more stability to the problem. The very large Reynolds number make diffusive terms becomes zeroes.

ABSTRAK

Analisis secara berangka iaitu pergerakan penutup dalam satu ruang berbentuk segi empat sama dibincangkan. Persamaan Navier-Stokes yang bergantung pada masa, dua dimensi dan tak boleh dimampat dikaji. Persamaan menakluk yang mengandungi persamaan kesinambungan dan momentum diperoleh daripada keabadian jisim dan keabadian momentum. Pertama, persamaan menakluk ditukarkan kepada persamaan tak berdimensi. Persamaan ini diselesaikan secara berangka menggunakan kaedah Marker and Cell (MAC) dan juga kaedah langkah pecahan. Kemudian, keputusan dilakarkan melalui Matlab. Penyelesaian berangka diperoleh mengandungi profil halaju, tekanan dan kontur pusaran. Profil halaju dibandingkan dengan [1]. Keputusan menunjukkan kaedah MAC sesuai digunakan dalam penyelesaian masalah ini kerana ralat adalah sedikit sahaja. Nombor Reynolds yang berlainan digunakan menyebabkan gangguan pada kontur pusaran. Kemudian, masalah dikembangkan lagi dengan menambah halaju pada bahagian bawah ruang. Pola aliran bendalir bertukar apabila halaju yang berbeza arah digunakan dalam masalah tersebut. Syarat kestabilan linear mestilah diikuti dalam pemilihan nombor Reynolds. Semakin kecil nombor Reynolds, semakin stabil masalah tersebut. Nombor Reynolds yang sangat besar menyebabkan sebutan pembauran menjadi sifar.

TABLE OF CONTENTS

APPENDIX

LIST OF FIGURES

LIST OF SYMBOLS

p	–	Pressure N/m^2
μ	–	Fluid dynamic viscosity (kg/ms)
ρ	–	Fluid density (kg/m^3)
Re	–	Reynolds number
U	–	Lid velocity(m/s)
L	–	Length of the cavity in the flow direction (m)
u	–	Velocity component in x-direction
v	–	Velocity component in y-direction
dS	–	Surface area
ϑ	–	Volume
F_x	–	Force in x-direction
m	–	Mass of fluid
a_x	–	Acceleration in x-direction
P	–	Nondimension of pressure

Chapter 1

Introduction

1.1 Background of Study

The lid driven cavity flow is one of the most studied fluid problems in computational fluid dynamics field. The lid driven cavity flow is the motion of a fluid inside a cavity with a lid moving by a constant velocity. Moreover, extensive computational and experimental of fluid flow behaviours inside lid driven cavities have been studied. Numerous studies have been carried out on flow patterns inside a cavity. The simplicity of the geometry of the cavity flow makes the problem easy to code and apply boundary conditions. The lid driven flow in a cavity has become a widely used for testing two and three-dimensional numerical simulation schemes.

There are different methods of solution in solving lid driven cavity. Numerical solution of the incompressible Navier-Stokes equations has been investigated by researchers. Finite difference, finite element and finite volume methods have been extensively developed. The majority of the early fundamental research work in this field were based on finite difference formulations, whereas the finite volume method dominates more recent activity and the commercial codes used in industry. This problem is commonly encountered in a variety of engineering applications such as cooling of electronic devices, lubrication technologies, drying technologies and dynamics of lakes [3].

In addition, two dimensional viscous incompressible flow equations are usually expressed in one of two different formulations based on the dependent variables used. First, the primitive variable formulation which the equations are expressed in terms of the pressure and velocity. This is the formulation of choice for extension to three dimensional flows. Time splitting method also known as fractional step method, SIMPLE, SIMPLER, SIMPLEC and PISO algorithm will be adopted. The second form of the equations is the vorticity-stream function equations which are derived from the Navier-Stokes equations by eliminating pressure and incorporating the definitions for the vorticity-stream function. The problems will be solved using staggered grid or collocated grid. There are different types of software are used to compute the simulations of the problem. For example, MATLAB, FLUENT and FOTRAN [4]. On the other hand, Bruneau and Saad [5] studied the numerical simulation of driven cavity flow at high range of Reynolds number using high grid numbers had been conducted. They used the conventional Computational Fluid Dynamics (CFD) method which is by solving the two dimensional Navier-Stokes equations.

In recent years, due to rapidly increasing computational power, computational methods have become the essential tools to conduct research in various engineering fields. In parallel to the development of ultra high speed digital computers, CFD has become the new third approach apart from theory and experiment in the philosophical study and development of fluid dynamics. Fudhail [6] used Lattice Boltzmann method (LBM) as an alternative method to conventional CFD. The main advantage of LBM is its flexibility in terms of programming and better accuracy in dealing with complicated boundary of geometries [6]. In addition, the LBM is also better than the classical CFD in the range of small to moderate Reynolds numbers if dealing with flows in complex geometries [7]. Fluid flow behaviours of steady incompressible flow inside lid driven square cavity is studied. Numerical calculations are conducted for different Reynolds numbers by using Lattice Boltzmann scheme.

In the present study, two dimensional unsteady Navier-Stokes equation in primitive variables is considered. Here, Marker and Cell (MAC) method is applied to a well-established benchmark problem namely the flow in a lid driven cavity. The purpose of this model is to investigate the pattern of flow included the velocity profile, vorticity contours, pressure of the flow. The various number of Reynolds number is applied lead to the perturbation on the vorticity contour.

1.2 Objectives of Study

The objectives of these studies are :

(i) to derive and solve the governing equations for the lid driven cavity by using Marker and Cell (MAC) method.

(ii) to investigate the flow pattern in terms of vorticity contour of the lid driven cavity under the influence of varying Reynolds number by using MATLAB software.

1.3 Scope of Study

This research will focus on two dimensional, laminar and time dependent of lid driven cavity. The nonlinear partial differential equations of the governing equations is derive from conservation of momentum and conservation of mass. We use MAC method in order to solve the equation. The disretisation is done under consideration of MAC staggered grid. Here, we adopted fractional step method. Then, we apply the solution into MATLAB software to investigate the simulation of flow in terms of velocity, vorticity and pressure.

1.4 Significance of Study

This research project will lead us to better understand about flow in a lid driven cavity. An imposed of varying moderate Reynolds numbers can create significant changes in the flow patterns in terms of vorticity. Based on this basic understanding, a more challenging exploration can be made such as solving higher Reynolds numbers with different approach formulation such as stream function vorticity. Another algorithm such as SIMPLE, SIMPLER, SIMPLEC or PISO will be applied into problem. Another types of mathematical computation such as FORTRAN and FLUENT can be used into problem.

Bibliography

1. Rusli, N. Numerical Computation of a Two-Dimensional Navier-Stokes Equation Using an Improved Finite Dierence Method. *MATEMATIKA*, 2011. 27(1): 1–9.

2. Anderson, J. D. *Computational Fluid Dynamics, The basics with Applications*. New York: McGraw Hill. 1995.

3. Parvin, S. and Hossain, N. Investigation on the Conjugate Effect of Joule Heating and Magnetic Field on Combined Convection in a Lid Driven Cavity with Undulated Bottom Surface. *Journal of Advanced Science and Engineering Research*, 2011. 1: 210–223.

4. Zogheib, B. and Barron, R. M. Velocity-pressure Coupling in Finite Difference Formulations for Navier Stokes Equations. *International Journal Numerical Methods Fluids*, 2001. 65: 1096–1114.

5. Bruneau, C. and Saad, M. The 2D Lid Driven Cavity Problem Revisited. *Computer and Fluids*, 2004. 35: 326–348.

6. Fudhail, A. M. Numerical Simulations of Flow Behavior in Driven Cavity at High Reynolds Numbers. *IIUM Engineering Journal*, 2011. 12(8): 97–103.

7. Filippova, O. and Hanel, D. Grid Refinement for Lattice-BGK Models. *Journal of Computational Physics*, 1998. 147: 219–228.

8. Harlow, F. H. and Welch, J. E. Numerical Calculation of Time Dependent Viscous Incompressible Flow of Fluid with Free Surface. *The Physics of Fluids*, 1965. 8(12): 2182–2189.

9. Johnstonang, H. and Liu, J. G. Finite Difference Schemes for Incompressible Flow based on Local Pressure Boundary Condition. *Journal of Computational Physics*, 2002. 180: 120–154.

10. Zhang, X. Computation of Viscous Incompressible Flow using Pressure Correction Method on Unstructured Chimera Grid. *International Journal of Computational Fluid Dynamics*, 2006. 20(9): 637–650.

11. Burggraf, O. R. Analytical and Numerical Studies of the Sturcture of Steady Seperated Flows. *Journal of Fluid Mechanics*, 1966. 24: 113–151.

12. Davis, G. D. V. and Mallinson, G. D. An Evaluation of Upwind and Central Difference Approximations by a Study of Recirculating Flow. *Computers and Fluids*, 1974. 4: 29–43.

13. Ghia, U., Ghia, K. N. and Shin, C. T. High-Re Solutions for Incompressible Flow Using Navier-Stokes Equations and a Multigrid Method. *Journal of Computational Physics*, 1982. 48: 387–411.

14. Schreiber, R. and Keller, H. B. Driven Cavity Flows by Efficient Numerical Techniques. *Journal of Computational Physics*, 1983. 49: 310–333.

15. Freita, C. F., Street, R. L., Findikakis, A. N. and Koseff, J. R. Numerical Simulation of Three Dimensional Flow in a Cavity. *Journal of Fluid Mechanics*, 1985. 5: 561–575.

16. Bruneau, C. and Jouron, C. An Efficient Scheme for Solving Steady Incompressible Navier-stokes Equations. *Journal of Computational Physics*, 1990. 89: 389–413.

17. Chorin, A. J. Numerical Solution of the Navier-Stokes Equations. *Mathematics of Computation*, 1968. 22: 745–762.

18. Shen, J. Hopf Bifurcation of the Steady Regularized Driven Cavity Flow. *Journal of Computational Physics*, 1991. 90: 1–20.

19. Poliashenko, M. and Aidun, C. K. A Direct Method for Computation of Simple Bifurcation. *Journal of Computational Physics*, 1995. 121: 246–260.

20. Fortin, A., Jardak, M. and Gervais, J. Localisation of Hopf Bifurcations in Fluid Flow Problems. *International Journal for Numerical Mathods in Fluids*, 1997. 24: 1185–1210.

21. Miller, W. FLow in the Driven Calculated by Lattice Boltzmann Method. *Physical Review*, 1995. 51: 3659–3669.

22. Jackson, A. Multigrid Dolutions to the Navier-Stokes Equations in two and three-dimensions. PhD thesis. *University of Manchester, Manchester, UK*, 1996.

23. Botella, O. and Peyret, R. Benchmark Spectral Results on the Lid Driven Cavity Flow. *Computers and Fluids*, 1998. 27(4): 421–433.

24. Azzam, N. A. Numerical Solution of the Navier-Stokes Equations for the Flow in a Lid Driven Cavity and a Cylinder Cascade. PhD thesis. *University of Manchester, Manchester, UK*, 2003.

25. Albensoeder, S. and Kuhlmann, H. C. Accurate three-dimensional Lid Driven Cavity Flow. *Journal of Computational Physics*, 2005. 206: 536–558.

26. Chiang, T. P., Hwang, R. R. and Sheu, W. H. Finite Volume Analysis of Spiral Motion in a Rectangular Lid Driven Cavity. *Journal of Fluid Mechanics*, 1996. 23: 325–346.

27. Auteri, F., Parolini, N. and Quartapelle, L. Numerical Investigation on the Stability of Singular Driven Cavity Flow. *Journal of Computational Physics*, 2002. 183: 1–25.

28. Tiesinga, G., Wubs, F. and Veldman, A. Bifurcation Analysis of Incompressible Flow in a Driven Cavity by the Newton-Picard Method. *Journal on Applied Mathematics*, 2002. 140: 751–772.

29. Erturk, E., Corke, T. and Gokcol, C. Numerical Solutions of 2-D Steady Incompressible Driven Cavity Flow at High Reynolds Numbers. *International Journal for Numerical Methods in Fluids*, 2005. 48: 747–774.

30. Tee, T. W. and Sobey, I. J. Spectral Method for the Unsteady Incompressible Navier Stokes Equations in Gauge Formulation. *Oxford University Computing Laboratory Numerical Analysis Group, Oxford, England*, 2004: 1–25.

31. Sahin, M. and Owens, R. G. A Novel FullyImplicit Finite Volume Method Applied to the Lid Driven Cavity Problem Part I: High Reynolds Number Flow Calculations. *International Journal for Numerical Methods in Fluids*, 2003. 42: 57–77.

32. Zhu, B. Finite Volume Solution of the Navier Stokes Equations in Velocity Vorticity Formulation. *International Journal for Numerical Methods in Fluids*, 2005. 48: 607–629.

33. Yeckel, A., Smith, J. W. and Derby, J. J. Parallel Finite Element Calculation of Flow in three-dimensional Lid Driven Cavity using the CM-5 and T3D. *Journal of Fluid Mechanics*, 1997. 24: 1449–1461.

34. Young, D. L., Lui, Y. H. and Eldho, T. I. A Combined BEM-FEM Model for the Velocity-vorticity Formulation of the Navier-Stokes Equations in three-dimensions. *Journal of Fluid Mechanics*, 2000. 24: 307–316.

35. Zunic, Z., Hribersek, M. and Skerget, L. 3D lid driven Cavity Flow by Mixed Boundary and Finite Element Method. *European Conference on Computational Fluid Dynamics*, 2006: 1–12.

36. Shala, M., Pericleous, K. and Patel, M. A Two Dimensional Unstructured Staggered Mesh Method with Special Treatment of Tangential Velocity. *Proceedings of the 4th WSEAS International Conference on Fluid Mechanics and Aerodynamics, Elounda, Greece*, 2006: 48–53.

37. Sun, K., Pyle, D. L. and Baines, M. J. Velocity Proles and Frictional Pressure Drop for Shear Thinning Materials in Lid Driven Cavity with Fully Developed Axial Flow. *Chemical Engineering Science*, 2006. 61: 4697–4706.

38. Kumar, D. S., Kumar, K. S. and Das, M. K. A Fine Grid Solution for a Lid Driven Cavity Flow Using Multigrid Method. *Engineering Applications of Computational Fluid Mechanics*, 2009. 3: 336–354.

39. Boppana, V. B. L. and Gajjar, J. S. B. Global Flow Instability in a Lid Driven Cavity. *International Journal for Numerical Methods in Fluids*, 2010. 62: 827–853.

40. Rahman, M. M., Alim, M. A. and Sarker, M. A. Numerical Study on the Conjugate Effect of Joule Heating and Magnetohydrodynamics Mixed Convection in an Obstructed Lid Driven Square Cavity. *International Communication in Heat Mass Transfer*, 2010. 37: 524–534.

41. Omari, R. CFD Simulations of Lid Driven Cavity Flow at Moderate Reynolds Number. *European Scientific Journal*, 2003. 9: 22–35.

42. Takami, H. and Kuwahara, K. Numerical Study of Three-Dimensional Flow within a Cubic Cavity. *Journal of the Physical Society of Japan*, 1974. 37: 1695–1698.

43. Ku, H. C., Hirsh, R. S. and Taylor, T. D. A Pseudospectral Method for Solution of the three-dimensional Incompressible Navier-Stokes Equations. *Journal of Computational Physics*, 1987. 70: 439–462.

44. Perng, C. and Street, R. L. Three-Dimensional Unsteady Flow Simulations: Alternative Strategies for a Volume-Averaged Calculation. *International Journal for Numerical Method in Fluids*, 1989. 9: 342–362.

45. Loukopoulos, V. and Tzirtzilakis, E. Biomagnetic channel flow in spatially varying magnetic field. *International Journal of Engineering Science*, 2004. 42: 571–590.

46. Biringen, S. and Chow, C. Y. *An Introduction to Computational Fluid Mechanics by Example.* New Jersey: John Wiley. 2011.

47. Tryggvason, G. A Code for a Navier-Stokes Equations in Velocity-Pressure Form. *http://www.nd.edu/ gtryggva/CFD-Course/.20 October 2013*: 1–6.