KINEMATIC BEHAVIOR OF SOLID PARTICLES IN ENCLOSED LID-DRIVEN CAVITY USING CIP METHOD

ALI AKBARI SHELDAREH

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

Faculty of Mechanical Engineering
Universiti Teknologi Malaysia

FEBRUARY 2012

To my beloved family and friends

And my respectful supervisor

Thank you for all kindness and sacrifices you made for me.

ACKNOWLEDGEMENTS

In the name of Allah, the most Gracious, the Most Merciful.

I have faced difficulties during preparation of this thesis, but with the help and support of Dr.Nor Azwadi Che Sidik, I have overcome all of them. I sincerely thank Dr. Nor Azwadi Bin Che Sidik for his guidance, help and motivation. I have learnt a lot from him and his positive contributions and responsibility were a boon to me.

I would like to thank my Supervisor Dr. Nor Azwadi Bin Che Sidik, the Head of Department of Postgraduate studies in the Mechanical Department for his warm assistance in finalizing the research problem and giving useful comments. A great deal of appreciation also goes to the Faculty of Mechanical Engineering (FKM). I am especially grateful to my fellow course mate Arman Safdari, for his sincere assistance, useful insights, and his positive contributions.

Nevertheless, I would like to thank to my family for loving me and being supportive in the duration of completing this thesis. I wish to thank my parents, Dr.Fereydoun Akbari Sheldareh and A. Ghanbari, for their love, support and kindness. They have always been a source of inspiration and encouragement for me. I wish to thank my brothers Arash and Amir Hossein who are always giving me positive energy and kindness, brothers who are the apple of my eyes.

ABSTRACT

The Cubic Interpolated Pseudo-Particle Navier Stokes equation (CIP-NSE) was applied to investigate the two-dimensional laminar square lid driven cavity flow of water at Reynolds number 1000. CIP-NSE scheme was used to solve hyperbolic term of the vorticity transport equation. In the CIP-NSE, 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 CIP-NSE provided a very good agreement with the established benchmark results by previous researchers. The Runge-Kutta method has been used to calculate the velocity and position of the particle with the effects of Drag force and Gravitational forces. The hard sphere model has been applied to show the collisions effect on particles in the Lid-Driven cavity. The main result achieved from the investigation is that, as the density of particles increases the number of particles collision in first seconds of the investigation decreases and the number of particles settled on the floor of the cavity increases, so for higher density of particles there have been large number of particles settlement on the floor and the collision at starting of investigation decrease as the particles moves slower, and for the lighter particles and lower density of particles number of collision at starting of investigation in more as the particles are lighter and move faster but the particles settlement on the floor of cavity are less in compare to higher density of particles. All simulation have been done for four different density of particle which are 1000, 1200, 1700, and 2000 (kg/m^3).

ABSTRAK

Persamaan Navier-Stokes untuk penentuan *cubic pseudo-particle* (CIP-NSE) telah digunakan untuk mengkaji alirain air dalam rongga berpandukan penutup dua matra dengan nombor Reynolds bersamaan dengan 1000. Kaedah CIP-NSE digunakan untuk menyelesaikan istilah hiperbolik bagi persamaan perjalanan vortex. Bagi CIP-NSE, kecerunan dan nilai vortex di nod ditentukan dan fungsi aliran ditentukan oleh persamaan vortex. Keputusan daripada simulasi berangka CIP-NSE didapati hampir serupa dengan keputusan daripada penyelidik sebelum ini. Kaedah Runge-Kutta telah digunakan untuk meramal kelajuan dan kedudukan zarah dengan mengamibil kira daya rintangan dan daya tarikan graviti. Model sfera keras telah digunakan untuk menunjukkan kesan perlanggaran ke atas zarah di dalam rongga berpandukan penutup. Kajian ini telah membuktikan bahawa semakin tinggi ketumpatan zarah, kadar perlanggaran zarah di awal kajian semakin rendah. Disebabkan ketumpatan yang tinggi, zarah-zarah akan tenggelam ke dasar rongga tersebut, dan zarah tersebut bergerak dengan perlahan, seterusnya menyebabkan kadar perlanggaran zarah yang rendah. Zarah yang ringan dan berketumpatan rendah mempunyai kadar perlanggaran yang tinggi kerana zarah yang ringan bergerak dengan lebih pantas dan seterusnya menghasilkan lebih banyak perlanggaran. Kesemua simulasi telah dijalankan untuk empat nilai ketumpatan zarah iaitu 1000, 1200, 1700 dan 2000 (kg/m^3).

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
FDM	-	Finite Difference Method
FEM	-	Finite Element Method
FVM	-	Finite Volume Method
CIPNSE	-	Cubic Interpolated Pseudo-particle Navier-Stokes Equation Method

LIST OF SYMBOLS

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
W	-	Gravitational force
F_b	-	Bouyancy force
F_D	-	Drag force
ρ_p	-	Particle density
g	-	Gravity
C_D	-	Coefficient of drag
m_j	-	Mass of particle
F_{pj}	-	External force
ω_j	-	Angular velocity
T_j	-	Particle torque
u_j	-	X direction velocity of particle
ν_j	-	Y direction velocity of particle
$V_1^{(0)}$	-	Pre-collision velocity
V	-	Post-collision velocity
G	-	Collisional relative velocity
e	-	Restitution coefficient
a	-	Particle radius

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

CHAPTER 1

INTRODUCTION

1.1 Introduction

One of the biggest inventions of mankind is the computer. Nowadays, the lack of a computer may cause many problems. The world is changing rapidly, with the computer's evolution as evidence. Since its invention in the early 20^{th} century, the computer started off as big as a house and was incapable of rapid calculations.

However it was not the end of the story; it was just the beginning of a great invention. After years of struggle and improvements made by companies, they have improved the computer in many aspects such as the size, weight and the performance. People used to need days or months to execute a task on an old version of the computer; the same task can be done in mere minutes on today's computers.

Researchers and accountants benefit a lot from the improvements of today's computer performance. They save more time and can perform more tasks in mere minutes, hence they will have more time doing other tasks and research.

Centuries before the invention of the computer, researchers can only count on experimental data and results to comprehend the actions of the flow of fluids and derive many other correlations and relationship. An example of the relationship which is famous and widely used is the Reynolds Number (Re) which was discovered once hundreds of successful experiments and investigations have been done.

The experiments were conducted by Osborn Reynolds in the 1880s constructing the founding of the dimensionless Reynolds number, Re, as the important parameter for the resolve of the flow system in pipes, whether turbulent or not [1] the next amazing and successful experiment ever accomplished in last few decades was the airplane which was designed and developed in 1903. Oliver and Wilbur Wrights were the team successfully lead the world into a new aspect and those conquests were attained so many praiseworthy experience and experiments.

Clearly, the most challenging part of experimentation is to develop an effective data which need a large number of experiments. The outcome from experiment is very hopeful because it is the real thing that is really happening. By some means, it is difficult when showing an experiment since the preparation of the composition and devices is boring if it does not follow the instruction in a correct way.

As the world technologically advances, computers also improved. CFD or Computational Fluid Dynamic is one of the applications which can be presented by a computer. CFD simulates fluid flow, and hence is a great tool to help solve problems in fluid flow. Many simulations were done using the CFD and it has been a great help for engineers and scientists. CFD is an easier and more cost-effective alternative to conducting an experiment, which could be expensive and time-consuming.

Furthermore, the use of CFD is current and will always produce a good result if the formulation, especially the numerical simulation, was correctly selected and evaluated. More researches were carried out, and a large amount of numerical method was applied using a computer.

Generally, solutions for fluid dynamics can be introduced through experiments where many relationships are established, and can be classified into three major categories which have more relationships.

There are three different categories of solution for fluid dynamics problems: the first one is through experiments, where the problem will be investigated in an experimental manner and in a sample mode; the second category is a theoretical solution, which deals with the fact that most problems dealing with fluid dynamics have its own assumptions and mathematical equation that will result in analytical solutions. The final category of solution is the recently generated and recently used method known as CFD, which stands for Computational Fluid Dynamics. This particular classification is shown in Figure 1.1

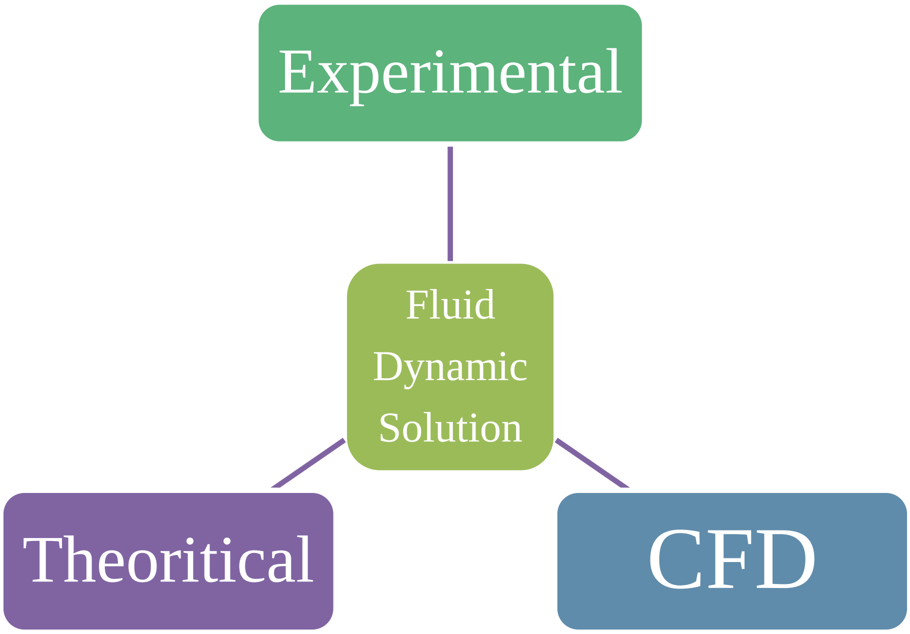

Figure 1.1 Classification of Fluid Dynamics Solution

1.2 Computational Fluid Dynamic (CFD)

Computational Fluid Dynamics (CFD) is one of the branches of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. The CFD has become an essential tool in solving problems governing the Navier-Stokes equation and the continuity equation, or any equation which are derived from these equations.

CFD works by showing on a computer how fluid behaves. One method is to divide the spatial domain into small cells to form a volume mesh or grid, and then apply a suitable algorithm to solve the equations of motion (Euler equations for inviscid, and Navier-Stokes equations for viscous flow). In many instances, other equations are solved simultaneously with the Navier-Stokes equations.

Other equations that may be included are those describing species concentration (mass transfer), chemical reactions and heat transfer, among others. More advanced codes allow the simulation of more complex cases involving multi-phase flows (e.g. liquid/gas, solid/gas, liquid/solid), non-Newtonian fluids (such as blood), or chemically reacting flows (such as combustion).

The basic approach in the use of CFD includes preprocessing, simulation, and post-processing. In preprocessing, the geometry of the problem is defined and the volume occupied by fluid is divided into meshes. During this process, both physical modeling and boundary conditions are defined.

Simulation begins after the process and the equations are solved iteratively. Post-processing is where the postprocessor is used for the analysis and visualization of the result. Computational fluid dynamics, usually abbreviated as CFD, is a branch of fluid mechanics which solves fluid dynamics problems by using numerical methods. In this method, computers play an important role in computing and calculating the fluid flow problem.

There are many applications of CFD which are useful in the fields of research, education, automotive, design and sports, among others. This thesis focuses on using CFD to solve non-linear partial differential equation (PDE) where the analytical solution typically does not exist. Regardless, some flows with analytical solutions have applied with numerical method for validation purposes.

The base of CFD is the well-known and unsolvable non-linear incompressible full Navier-Stokes equation. There are two types of CFD: the numerical type and another type, which uses computer software to simulate. The second type uses a form of software to simulate or calculate the CFD problem.

Software like FLUENT©, which is very easy to use and can be used to simulate virtually any fluid flow problems, has some disadvantages, such as the user’s lack of knowledge about the equations applied, the assumptions or other criteria. This software is generally used for practical applications and for complicated geometry and complex conditions. In spite of that, FLUENT© software is established when it comes to numerical method but it is not publicized.

The earlier type of simulation is very notable because who create the codes could understand the simulation, the assumption, boundary conditions and other variables very well. This type of simulation is appropriate for information sharing because many papers are published frequently which touts the use of new methods, for example the Lattice Boltzmann method, Bifurcation method and CIP. The better method is determined by carrying out comparison and validation between the aforementioned methods. The simulation requires the user to be well-versed in programming software such as FORTRAN, C++, and MATLAB; example applications are the simulation of flow over cylinder [2] and the experimental [3]**.**

1.2.1 Governing Equation in CFD

There are many variables and parameters in fluid flow which control the characteristic of the flow. Generally, these parameters are related to the physics of the flow, the nature of the fluid or the surrounding system. Some of those variables which are usually arising in fluid flow are listed:

- Temperature T
- Pressure P
- Velocity u
- Fluid density ρ
- Fluid viscosity, dynamic (μ) , and kinematic (ν)

These are important variables in CFD simulation because they are useful and are generally incorporated in three major governing equations. These governing equations are very important for CFD and also for heat transfer simulation. These equations can also be modified depending on the physics of the fluid flow or based on the assumption which can be made. The equations involved in incompressible fluid flow are:

- The continuity equation (conservation of mass)

$$\nabla . u = 0 \tag{1.1}$$

- The Navier-Stokes Equation (conservation of momentum)

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

- The energy equation (conservation of energy)

$$E_t + \nabla(Eu) = \dot{q} - p.\nabla u + f.u \tag{1.3}$$

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

1.2.2 The Navier-Stokes Equations

The Navier-Stokes equations were named after the French engineer and scientist Claude Louis Henri Navier and the English mathematical physicist George Gabriel Stokes. The equations' essential form was set forth by Navier in 1822; however, the origin of viscous stress was not properly represented. The latter was addressed by others, in particular by Poisson and Saint-Venant, but independently developed by Stokes in 1845.

Stokes constructed a number of solutions to the equations of viscous flow, which confirmed their ability to describe fluid dynamical phenomena.

The equation which describes the motion of fluid substances, i.e. substances which can flow, arise from applying Newton's second law to fluid motion, together with the assumption that the fluid stress is the sum of a diffusing viscous term (proportional to the gradient of velocity), plus a pressure term. The mathematical relationship which governs the fluid flow is the continuity equation and Navier-Stokes equation given by:

$$\nabla . u = 0 \tag{1.4}$$

$$\frac{\partial u}{\partial t} + u . \nabla u = -\nabla P + v \nabla^2 u \tag{1.5}$$

With velocity **u**, pressure *P*, and kinematic shear viscosity. The Navier–Stokes equations are a set of nonlinear partial differential equations which, unlike algebraic υ equations, do not explicitly establish a relation among the variables of interest (e.g. velocity and pressure). Rather, they establish relations among the rates of change.

Navier-Stokes equation is well known in the field of fluid dynamics. The equation is nonlinear and usually the flows that use this equation are considered incompressible. Many fluid flows are governed by this equation because in describing the conservation of momentum, the equation is almost perfect. In the equation lie an unsteady term, a diffusive term, a pressure term, a convective term and the external force which is a complete package for momentum conservation. However, there is no analytical solution to this equation as there are many Partial Difference term in the equation.

During the writing of this thesis, this equation is still not solved but many types of numerical methods were tried out by scientist and engineers and hence produce their own solution of numerical simulation. However, there still are exceptions, because some fluid flows having the analytical solution and this exception will be discussed later in the next chapter.

1.3 Problem Statement

Many classical numerical methods have been applied to investigate the behaviors of particles in a lid-driven fluid cavity by solving the Navier-Stokes equation accompanied with Newton's second law and the CIP method. Yet, these numerical methods are still insufficient; for higher order of accuracy, more grids are needed to satisfy the methods.

- Many numerical solutions are being applied to solve Navier-stokes equation but they still lack accuracy
- Low Mesh Grid has a higher accuracy.
- How to properly describe the flow of a particle within a lid driven fluid cavity.
- Effect of collisions on particle flow and the trajectory of particle.
- The effects of gravitational forces and drag forces and collisions on particle flow.

1.4 Objectives of the research

The objective of this thesis is to investigate solid particles behaviors in a lid-driven cavity flow while considering the drag force and gravitational force. In addition, the objective is to observe the effects of collision which is divided to two parts and will be further defined in chapters two and three. This research is mainly based on study of the flow in a square two-dimensional cavity with particle moving and particle collision is limited to hard sphere models only. Meanwhile, the CIP method is applied to solve the Navier-Stokes equation to express the result with less grid structure, which will increase the order of accuracy. The Runge-Kutta method is used to calculate the drag force and gravitational force exerted on the particle.

1.5 Significance of study

Simulation allows scientists to virtually construct the experimental conditions so that they can investigate real conditions without actually experiencing that particular phenomenon. In some cases, it would be quite impossible to perform that experiment with the existing facilities and defined conditions. In the field of computational fluid dynamics, the most interesting areas in this field are description of fluid flow and the prediction and profile of the flow.

Moreover, viscous fluids while in rotary motion have diverse industrial and commercial applications. The main focus for researchers has been lid-driven cavity flows, where the fluid is set into motion by part of the containing boundary. These types of flows are tedious for analyzing fundamental aspects of recirculation fluids: in spite of the apparently simple geometry, lid-driven cavity flows may involve a high degree of complexity. This is an interesting problem, which may yield much information about the interaction between fluid and particle and particle-particle and particle-wall collisions in a wide range of practical configurations. This has not been widely studied before this.

1.6 Scope of the Study

The scope for this particular research is bound by two matters and will therefore be adhered to throughout the research, which are:

Solve the advection equation with the application of CIP for NSE by:

- Comparing the results with practical and simulated benchmarks with over other methods.
- Comparing the dynamics of solid particles with the results that have been revealed so far.
- Simulating multi-particle behavior in a lid-driven cavity while considering the effects of collision and gravitational force and drag force.

Implementation of results verification:

- Two-dimensional incompressible, unsteady, lid-driven cavity.
- Two-dimensional incompressible lid-driven flow in square cavity without particle affecting, focusing on the streamline plots and velocity plots.
- Two-dimensional incompressible lid-driven flow in square cavity representing the dynamics of solid particle, focusing on the orbit of the solid particle.
- Two-dimensional incompressible lid-driven cavity flow with multiple particles and collisions effect and gravitational force and drag force.
- Particle collision is limited to Hard-Sphere model only.
- Gravitational force and drag force are solved using Runge-Kutta method.

REFFERENCES

1. Cengel Y.A *Introduction to Fluid Mechanics*. 5th. ed. Singapore: Mc Graw Hill. 2003
2. Lee K, Yang K.S Flow patterns past two circular cylinders in proximity. *Computers & Fluids* , 2009. 38: 778-788.
3. Xu G, Zhou Z Strouhal numbers in the wake of two inline cylinders. *Experiments in Fluids*, 2004, 37: 248-256.
4. Shankar and Deshpande, *Fluid Mechanics in the Driven Cavity.* Annu. Rev. Fluid Mech, 2000: p. 32, p93.
5. Bruneau C.H., Saad M., *The 2D lid-driven cavity problem revisited,*Computers & Fluids, (2006), 35 326–348.
6. Cheng M., Hung K.C, *Vortex structure of steady flow in a rectangular cavity*, Computers & Fluids., (2006) 35, 1046–1062.
7. Perron S., Boivin S., Herard J.M, *A finite volume method to solve the 3D Navier Stokes equations on unstructured collocated meshes*, Computers & Fluids., (2004), 33 , 1305–1333.
8. Elman H.C.,Howle V., Shadid J.N., Tuminaro R.S., *A parallel block multi-level preconditioned for the 3D incompressible Navier Stokes equations,* Journal of Computational Physics, (2003), 187 ,504–523.
9. Zhang J, *Numerical simulation of 2D square driven cavity using fourth order compact finite difference schemes*, Computers and Mathematics with Applications .,(2003) , 45 ,43–52.
10. Auteri F., Quartapelle L, *Galerkin Spectral Method for the Vorticity and Stream Function Equations.* Journal of Computational Physics, (1999), 149: 306–332.
11. Mei R., Shyy W., Yu D., Luo L.S., *Lattice Boltzmann method for 3-D flows with curved boundary,* Journal of Computational Physics, (2000), 161 ,680 699.

12. Wright J.A., Smith R.W., *An edge-based method for incompressible Navier–Stokes equations on polygonal meshes*, Journal of Computational Physics, (2001), 169 24–43.
13. Liu C.H., Leung D.Y.C., *Development of a finite element solution for the unsteady Navier Stokes equations using projection method and fractional! Scheme*, Computer Methods in Applied Mechanics and Engineering, (2001), 190, 4301–4317.
14. Boivin S., Cayre F., Herard J.M., *A finite volume method to solve the Navier-Stokes equations for incompressible flows on unstructured meshes,* International Journal of Thermal Science, (2000), 39 ,806–825.
15. Neofytou,P.,*A 3rd order upwind finite volume method for generalized Newtonian fluid flows,* Advances Engineering Software,(2005), 36, 664–680.
16. Tai C.H., Zhao Y., Liew K.M., *Parallel multigrid computation of unsteady incompressible viscous flows using a matrix-free implicit method and high resolution characteristics based scheme*, Computer Methods in Applied Mechanics and Engineering, (2005), 194, 3949–3983.
17. Albensoeder S., Kuhlmann H.C., and Rath H.J., *Multiplicity of Steady Two-Dimensional Flows in Two-Sided Lid-Driven Cavities*. Theoret. Comput. Fluid Dynamics, (2001), 14: 223–241.
18. Albensdoer S., Kuhlmann H.C., *Linear stability of rectangular cavity flows driven by anti-parallel motion of two facing walls,* Journal Of Fluid Mechanics, (2002), 458, 153–180.
19. Albensdoer S., Kuhlmann H.C, *Three dimensional instability of two counter rotating vortices in a rectangular cavity driven by parallel wall motion,* European Journal of Mechanics B/Fluids, (2002), 21, 307–316.
20. Povitsky A., *Three-dimensional flow in a cavity at yaw*, Nonlinear Analysis , (2005), 63 1573–1584.
21. Sheu T.W.H., Tsai S.F., *Flow topology in a steady three-dimensional liddriven cavity*, Computers & Fluids, (2002), 31 ,911–934.
22. Oosthuizen P.H, D. Naylor. *Introduction to Convective Heat Transfer Analysis*. Singapore : Mc Graw Hill. 1999.
23. Thomas S. *Applied Dimensional Analysis and Modeling*. US: Elsevier, 2007.

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

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

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

27. Agrawal L., Mandal J.C., Marathe A.G., *Computations of laminar and turbulent mixed convection in a driven cavity using pseudo compressibility approach,* Computers & Fluids, (2001) 30 ,607–620.

28. Zhou Y., Zhang R., Staroselsky H., Chen H., *Numerical simulation of laminar and turbulent buoyancy-driven flows using a lattice Boltzmann based algorithm,* International Journal of Heat and Mass Transfer, (2004) 47 ,4869–4879.

29. Alleborn N., Raszillier H., DurstF., *Lid driven cavity with heat and mass transport,* International Journal of Heat and Mass Transfer,(1999) 42,833–853.

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

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

32. Tu J, Yeoh G.H, and Liu C, *Computational Fluid Dynamics: A Practical Approach*. Elsevier. 2008.

33. Finnemore E.J, and Franzini J.B, *Fluid Mechanics with Engineering Application*. Mc Graw Hill. 2002.

34. Grebel, W.P. *Advanced Fluid Mechanics*. Academic Press. 2007

35. Panton, R. L. *Incompressible flow*. USA : John Wiley & Son. 1984.

36. 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.

37. Barragy E and Carey F.C. *Stream Function-Vorticity Driven Cavity Solutiion using p Finite Elements*. Computers & Fluids, 1997. 26(5): 453-468.

38. Albensoeder S, Kuhlmann H.C, and Rath H.J. *Multiplicity of Steady Two-Dimensional Flows in Two-Sided Lid-Driven Cavities*. Theoret. Comput. Fluid Dynamics, 2001. 14: 223–241.

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

40. 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.

41. 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.

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

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

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

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

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

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

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

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

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

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

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

53. White C.M., *The equilibrium of grains on the bed of a stream*, Proceedings of the Royal Society of London. Mathematical physical sciences, (1940),332–338.

54. Sommerfeld M., Huber N., *Experimental analysis and modeling of particle–wall collisions*, International Journal of Multiphase Flow, (1999), 25 1457–1489.

55. Sommerfeld M., *Modeling of particle–wall collisions in confined gas particle flows,* International Journal of Multiphase Flow, (1992), 18 905–926.

56. Frank T., Schade K.P., Petrak D., *Numerical simulation and experimental investigation of a gas solid two phase flow in a horizontal channel,* International Journal of Multiphase Flow, (1993), 19 187–198.

57. Kuru W.C., Leighton D.T., McCready M.J., *Formation of waves on a horizontal erodible bed of particles,* Journal of Multiphase Flow, (1995), 21 (6) 1123–1140.

58. Doron P., Barnea D., *Pressure drop and limit deposit velocity for solid liquid flow in pipes*, Chemical Engineering Science, (1995),50 (10) 1595–1604.

59. Hayden, K.S., K. Park, and J.S. Curtis, *Effect of particle characteristics on particle pickup velocity.* Powder Technology, 2003. 131(1): p. 7-14.

60. Portela L.M., Oliemans R.V.A., *Eulerian Lagrangian DNS/LES of particle turbulence interactions in wall bounded flows*, International Journal for Numerical Methods in Fluids, (2003), 26 719–727.

61. Crowe C, Sommerfeld M, Tsuji Y, *Multiphase Flows with Droplets and Particles*, CRC Press LLC, 1998.

62. Sundaram S, Collins L.R, Numerical considerations in simulating a turbulent suspension of finite-volume particles, Journal of Computational Physics 124 (1995) 337–350.

63. Kosinski, P., A. Kosinska, and A.C. Hoffmann., *Simulation of solid particles behaviour in a driven cavity flow*. Powder Technology (2009) 191 p. 327–339.

64. Pawel Kosinski , AlexC.Hoffmann, *An extension of the hard-sphere particle–particle collision model to study agglomeration,* Chemical Engineering Science 65 (2010) 3231–3239

65. Crowe C, Sommerfeld M, and Tsuji Y. *Multiphase Flow with Droplets and Particles*. CRC Press, 1998.

66. Castellanos A, Valverde J.M, and Quintanilla M.A.S. *Aggregation and sedimentation in gas-fluidized beds of cohesive powders*. Physical Review E, 64:Art. No. 041304, 2001.

67. Castellanos A, Valverde J.M, and Quintanilla M.A.S. *Aggregation and sedimentation in gas-fluidized beds of cohesive powders*. Physical Review E, 64:Art. No. 041304, 2001.

68. Balakin, B., Alex C. Hoffmann, Pawel Kosinski, *The collision efficiency in a shear flow*, chemical engineering science (2011) , 10.1016/2011.09.042

69. Hoomans B.P.B, Kuipers J.A.M, Briels W.J, and van Swaaij W.P.M. *Discrete particle simulation of bubble and slug formation in a two-dimensional gas-fluidised bed: a hard sphere approach*. Chem. Eng. Sci.51:99–108, 1996.

70. Visser J. An invited review: *Van der Waals and other cohesive forces affecting powder fluidization*. Powder Technol., 58:1–10, 1989.

71. Brilliantov N.V, Albers N, Spahn F, and Poschel T. *Collision dynamics of granular particles with adhesion*. Physical Review E, 76, 2007.

72. Tsorng S. J. ,Capart H. , Laie J. S.,*Three dimensional tracking of the long time trajectory of suspended particle in a lid driven cavity flow,* experiments in fluid , (2006) 40,314-328

73. Pawel Kosinski, Alex C. Hoffmann, *An extension of the hard-sphere particle-wall collision model to account for particle deposition,* Phys. Rev. E. 79, 2009, 061302-1–061302-11