NUMERICAL ANALYSIS ON THE EFFECTS OF CAVITY GEOMETRY WITH HEAT TOWARDS CONTAMINANT REMOVAL

MOHD NOOR ASRIL BIN SAADUN

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

AUGUST 2012

To my beloved family,

my mum HJH. ZALEHA BINTI HJ. SATER and

my dad HJ. SAADUN BIN HJ. BERHAN,

thank you for your 'doa', support and encouragement

that you have given in my life. I love you all so much.

ACKNOWLEDGMENT

First and foremost, ALHAMDULILLAH I'm grateful to ALLAH SWT on blessing in completing this project.

I would like to take this opportunity to thank my project supervisor, Dr. Nor Azwadi Che Sidik for his invaluable guidance, advising, assisting and support throughout this work from beginning until completely. Under his supervision, many aspects regarding on this project has been explored, and with knowledge, idea and support received from him, this project report can be presented in the given time.

I am also indebted with Universiti Teknikal Malaysia Melaka (UTeM) and Ministry of Higher Eduaction (MOHE) Malaysia for funding my Master,s study. Special thanks also given to Universiti Teknologi Malaysia (UTM) for providing relevant literature and facilities to conduct my research efficiently.

Finally, I wish to dedicate my gratitude to all my lecturers involved in teaching my course, thanks for the lesson that been delivered and to all my friends, course mate, and anyone especially Mehran Salehi that has provided whether an idea or support, directly or indirectly, that played a role towards in completing this work.

ABSTRACT

These studies present a computational investigation of fluid-solid interaction flow in a channel. Fluid-solid interaction flow was created in three different shapes of cavity with additional constant heat flux under the floor of a horizontal channel. The flow and solid particle dynamics were explored using Cubic Interpolated Pseudo-Particle (CIP) method and Lagrangian scheme of Newton's law respectively for two main objectives. The first is to demonstrate the validity of the proposed Eulerian-Lagrangian in predicting the main characteristics of fluid-solid interaction flow. The second objective is to shed the light on the dynamics of the solid particle that take place in three types of cavities with heat effect, which has not been fully covered in the literature before. The results show that the particles trajectories are critically dependent on the magnitude of Reynolds numbers and the vortex behavior in the cavity and also buoyancy effect. Good comparisons with the previous studies demonstrate the multidisciplinary applications of this scheme.

ABSTRAK

Kajian ini menerangkan tentang pengiraan aliran cecair di dalam saluran dan interaksi bersama zarah pepejal. Aliran interaksi seperti cecair dan zarah pepejal telah dilakukan di dalam tiga bentuk rongga yang berlainan di mana tenaga fluks haba yang malar ditambahkan di bawah setiap rongga tersebut secara mengufuk. Aliran dan dinamik zarah pepejal telah dikaji menggunakan sisipan sematapadu kaedah Pseudo-Zarah (CIP) dan skim Hukum Newton Lagrangian sebagai dua objektif utama kajian ini dilakukan. Objektif yang pertama adalah untuk menunjukkan kesahihan cadangan atau idea Euleran-Lagrangian dalam meramal ciri-ciri utama aliran interaksi cecair dan zarah pepejal. Objektif kedua adalah untuk menentukan kedudukan pada dinamik zarah pepejal yang berlaku dalam tiga jenis rongga dengan kesan haba yang kebanyakkannya tidak dinyatakan sepenuhnya dalam kajian ilmiah sebelum ini.. Hasil kajian menunjukkan bahawa trajektori zarah adalah amat bergantung kepada magnitud atau nilai nombor Reynolds dan pembentukan vorteks di dalam rongga berbanding kesan keapungan iaitu kesan daripada haba yang diberikan. Perbandingan yang baik dengan kajian sebelum ini menunjukkan skim ini dapat digunakan dalam pelbagai aplikasi dalam bidang kejuruteraan.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF ABBREVIATIONS

2D	-	Two dimensional
CIP	-	Cubic Interpolated Pseudo-particle
CFD	-	Computational Fluid Dynamics
PDE	-	Partial Differential Equation
NSE	-	Navier-Stokes Equation
FDM	-	Finite Difference Method
CIPNSE	-	Cubic Interpolated Pseudo-particle Navier-Stokes Equation Method

LIST OF SYMBOLS

AR	-	Aspect Ratio
D	-	Channel diameter
P	-	Pressure
u	-	Velocity in x direction
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
L	-	Height of cavity
Re	-	Reynolds Number
t	-	Time
T	-	Dimensionless time
u_∞	-	Maximum inlet velocity
μ	-	Dynamic viscosity
ρ	-	Density
υ	-	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

LIST OF APPENDIXE

CHAPTER 1

INTRODUCTION

1.1 Overview

Flow past cavities have received great attention in the past decades in both experimental and numerical investigations due to its relevance to many practical engineering applications. For high speed examples of this flow type are flows over aircraft hulls, weapon bays, wheel wells and gas turbine channels while electronic devices on printed circuit boards are some low speed. The shape of cavities used in power-engineering equipment varies vastly. Some shape can be existing for the real application is rectangular, semicircular and also triangular. The effects of shape cavity with experiment have done by Ozalp et al. [1] with measured velocities by PIV. From Ozalp findings, maximum Reynolds stresses and turbulence intensity values were observed in the lid section of the cavity at the centerline position while rectangular and triangular cavities cause much greater turbulence compared to semicircular cavity shape.

On the other hand, hydrodynamics cleaning of components, parts and pipelines has become widely accepted as an alternative method of cleaning process. However, other problems arise when the presence of distinct cavities and steps due to poorly fitted components or connections in the pipelines. These problems contributed to the accumulation of contaminants and cleaning of these can lead to quite difficult problems. The hydrodynamic removal of a contaminated fluid from a cavity on the

floor of a duct has been studied thoroughly using numerical and experimental by Fang [2]. It was shown that the cleaning of the foulant with the same density as the fluid in the duct is more pronounced during the unsteady start-up of the duct flow and the rate of cleaning decreases as the flow reaches a steady state. The cleaning process is enhanced as the cavity aspect ratio (width/depth) is increased and as the duct Reynolds number increases.

In recent years, the study of convective heat transfer in a cavity has increased. Simultaneous convection of buoyancy and forced convection is called as combined or mixed convection, which is of great interest in engineering applications such as nuclear reactors, lakes and reservoirs, cooling process of electronic devices, solar applications, combustion chambers, food processing and float glass production in industry. The applications of convective heat transfer problems are numerous in the chemical and food processing industry. Relatively few studies, however, deal with the problem of hydrodynamic removal under the effect of convective heat transfer. Hence, the extension to such problem is important in hydrodynamic cleaning.

1.2 Objectives Of The Study

This study numerically reviews the current state of knowledge about the flow in the channel with cavity filled of solid particles. The main objective of this research is to investigate the behaviour of macroscopic rigid particles removal in term of percentage until reach steady state in a fully two-dimensional viscous flow with using CIP approached as a numerical scheme. Secondly, to determine the characteristics of the flow in the channel using three different shape cavities rectangular, semicircular and triangular. Thirdly, to examine the effect of the mixed convection heat transfer to the cavity with the major concern herein is vortex structure and the rate and percentage of contaminant removal.

1.3 Scope

The scope of this research only focuses into simulation of flow in an open cavity containing an incompressible fluid and solid particles. The fluid phase was modelled using the CIP approach where the solid particles are treated as points moving as a result of the fluid motion in the computational domain. The scopes for this particular research are bounded by these three matters and therefore will be followed throughout the research which are:

- Solving the advection equation with the application of CIP method
- Comparison of results with different cases
- The test case is flow in different shape of cavity, focusing on streamline plots, isotherm and time.

1.4 Limitation of The Study

In this study, the flow in the channel and cavity are two-dimensional analysis, steady, laminar, incompressible, mixed convection flow and only one way interaction between fluids onto particle to be considered. The fluid properties are assumed to be constant and the radiation effects are taken as negligible. The aspect ratio for all cases is fixed into 2.

1.5 Statement of Problems

Knowledge of the flow over open cavity is central in many natural and industrial applications. Examples are car sunroof, aircraft weapon bay, landing gear well. Comprehending the major mean fluid flow phenomena, such as vortex strength and structure, is crucial for the environmental management and design and operation of such engineering equipment. A major limitation, in the problem of fluid flow over a cavity, is the insufficient information about

the vortex structure due to the inflow velocity and also the shape of the cavity. Although there are few fundamental experimental investigations which dictate that formation of vortex enhances the mixing process and strong vortex helps in the process of contaminant removal, more detailed phenomena such as the interaction between the vortex formation and the percentage of contaminant removal and flow parameters on the particles dynamics are still not fully understood. Although there are some reports on non-rectangular cavities, however, the complicated nature of the flow needs further investigations on various parameters which influence the flow characteristics inside cavities.

REFERENCES

[1] C. Ozalp, A. Pinarbasi, B. Sahin. *Experimental measurement of flow past cavities of different shapes*, Experimental Thermal and Fluid Science 34 (2010) 505–515

[2] L.C. Fang, J.W. Cleaver, D. Nicolaou, *Transient removal of a contaminated fluid from a cavity*, Int. J. Heat Fluid Flow 20(1999) 605–613.

[3] E. Erturk, C. Gokcol, *Fourth-order compact formulation of Navier-Stokes equations and driven cavity flow at high Reynolds numbers*, Int. J. Numer. Meth. Fluids 50 (2006) 421–436.

[4] E. Erturk, B. Dursun, *Numerical solution of 2-D steady incompressible flow in a driven skewed cavity*, Z. Angew. Math. Mech. 87 (2007) 377–392.

[5] Y. Striba, *Analysis of the flow and heat transfer characteristics for assisting incompressible laminar flow past an open cavity*, Int. Commun. Heat Mass Transfer 35 (2008) 901–907.

[6] Zdanski, M.A. Ortega, G.C.R. Nide, J. Fico, *On the flow over cavities of large aspect ratio: A physical analysis*. Int Comm in Heat and Mass Transfer 2006; 33 (4): 458-66

[7] C.H. Kuo, C.W. Chang, *Shear-layer characteristics across a cavity with a horizontal top plate*, Fluid Dyn. Res. 22 (1998) 89–104.

[8] C.H. Kuo, S.H. Huang, *Effect of surface mounting of upper plate on oscillating flow structure within cavity*, Exp. Therm. Fluid Sci. 27 (2003) 755–768.

[9] C.H. Kuo, W.I. Jeng, *Lock-on characteristics of a cavity shear layer*, J. Fluid Struct. 18 (2003) 715–728.

[10] A.Y. D'yachenko, V.I. Terekhov, N.I. Yarygina, *Vortex formation and heat transfer in turbulent flow past a transverse cavity with inclined frontal and rear walls*, Int. J. Heat Mass Transfer 51 (2008) 3275–3286.

[11] H. Mercan, K. Atalik, *Vortex formation in lid-driven arc-shape cavity flows at high Reynolds numbers*, Eur. J. Mech. B-Fluid.

[12] M.H. Chang, C.H. Cheng, *Predictions of lid-driven flow and heat convection in an arc-shape cavity*, Int. Commun. Heat Mass Transfer 26 (1999) 829–838.

[13] R. Glowinski, G. Guidoboni, T.W. Pan, *Wall-driven incompressible viscous flow in a two-dimensional semi-circular cavity*, J. Comput. Phys. 216 (2006) 76–91.

[14] C. Migeon, A. Texier, G. Pineau, *Effect of lid-driven cavity shape on the flow establishment phase*, J. Fluid Struct. 14 (2000) 469–488.

[15] C.H. Kuo, S.H. Huang, *Influence of flow path modification on oscillation of cavity shear layer*, Exp. Fluids 31 (2001) 162–178.

[16] S.M. Grace, W.G. Dewar, D.E. Wroblewski, *Experimental investigation of the flow characteristics within a shallow wall cavity for both laminar and turbulent upstream boundary layers*, Exp. Fluids 36 (2004) 791–804.

[17] D. Rockwell, J.-C. Lin, Experimental data for computational fluid dynamic simulations of fluid oscillators: cavity flow, vols. 1–2, Fluid Mechanics Labs., Rept. R01, Dept. of Mechanical Engineering and Mechanics, Lehigh Univ., Bethlehem, PA, 1996.

[18] W.L. Oberkampf, L.I. Crow, *Numerical study of the velocity and temperature fields in a flow-through reservoir ASME*, J. Heat Transfer 98 (1976) 353–359.

[19] C.K. Cha, Y. Jaluria, *Effect of thermal buoyancy on the recirculating flow in a solar pond for energy extraction and heat rejection ASME*, J. Solar Energy Eng. 25 (1984) 428– 437.

[20] M. Sparrow, F. Samie, *Interaction between a stream which passes through an enclosure and nature convectionwithin the enclosure*, Int. J. Heat Mass Transfer 25 (1982) 1489–1502.

[21] P.H. Oosthuizen, J.T. Paul, *Mixed convection heat transfer in a cavity, in Fundamentals of Forced and Mixed Convection*, ASME HTD 42 (1985) 159–169.

[22] M.H. Chang, C.H. Cheng, *Predictions of lid-driven flow and heat convection in an arc-shape cavity*, Int. Commun. Heat Mass Transfer 26 (1999) 829–838

[23] E. Papanicolaou, Y. Jaluria, *Mixed convection from an isolated heat source in a rectangular enclosure*, Numer. Heat Transfer A 18 (1990) 427–461.

[24] M.K. Moallemi, K.S. Jang, *Prandtl number effects on laminar mixed convection heat transfer in a lid-driven cavity*, Int. J. Heat Mass Transfer 35 (1992) 1881-1892

[25] W.L. Chien, H.Rising, J.M. Ottino, *Laminar mixing and chaotic mixing in several cavity flows*. J. Fluid Mech. 170 (1986) 355–377.

[26] P.N. Shankar, *Three-dimensional Stokes flow in a cylindrical container*. Phys. Fluids 10 (1998) 540–549.

[27] T.Y. Hwu, *Fluid stirrings in a circular cavity with various driven boundaries*. Eur. J. Mech. B, Fluids 25 (2006) 192–203.

[28] F. Pan, A. Acrivos, *Steady flows in rectangular cavities*. J. Fluid Mech. 28 (1967) 643–655.

[29] H.S. Rhee, J.R. Koseff, R.L. Street, *Flow visualization of a recirculating flow by rheoscopic liquid and liquid crystal techniques*. Exp. Fluids 2 (1984) 57–64.

[30] A.K. Prasad, J.R. Koseff, *Reynolds number and end-wall effects on a lid-driven cavity flow*. Phys. Fluids A 1 (1989) 208–218.

[31] J.L. Guermond, C. Migeon, G. Pineau, L. Quartapelle, *Start-up flows in a three-dimensional rectangular driven cavity of aspect ratio 1:1:2 at Re = 1*000. J. Fluid Mech. 450 (2002) 169–199.

[32] C. Migeon, *Details on the start-up development of the Taylor–Gortler-like vortices inside a square-section lid-driven cavity for 1000 Re = 3200*. Exp. Fluids 33 (2002) 594–602.

[33] C. Migeon, G. Pineau, A. Texier, *Three-dimensionality development inside standard parallelepipedic lid-driven cavities at Re = 1000*. J. Fluids Struct. 17 (2003) 717–738.

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

[35] S.J. Tsorng, H. Capart, J.S. Lai, D.L. Young. *Three-dimensional tracking of the long time trajectories of suspended particles in a lid-driven cavity flow*. Exp. Fluids 2006; 40(2): 314-28.

[36] 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.