NUMERICAL SOLUTION OF CONVECTIVE HEAT TRANSFER IN A POLAR CAVITY

ALFRED SAGAYAM A/L VILAVANDRAN

A thesis submitted in fulfilment of the

requirements for the award of the degree of

Master of Engineering (Mechanical Engineering)

Faculty of Mechanical Engineering

Universiti Teknologi Malaysia

JAN 2012

ACKNOWLEDGEMENT

The successful completion of my project can be attributed to the combined efforts made by and the contribution made in one form or the other, by the individuals I hereby acknowledge. First of all, I express our heartfelt gratitude and thanks to my supervisor Prof Amer Nordin Darus, for encouraging me to move ahead and for being with me right from the beginning of the project and guiding me at every step.

I am grateful to my fellow course meets for his timely help. And last but not the least I also thank my parents and friends for being with me all the time and giving me moral support.

ABSTRACT

Numerical solution of 2-D polar cavity is very less investigated analytically, numerically and experimentally. The investigation is mainly to understand the flow phenomena in the domain. One of the difficulties that occur for the numerical investigation is domain setup for complex geometry. The numerical solution for polar cavity is made possible by ANSYS FLUENT. Steady state incompressible ideal gas is considered for simplicity and other mechanical and thermal properties of fluid are constant with respect to temperature and pressure. The cavity's stationary walls such as inner radial wall and side wall kept as isothermal wall, while outer radial wall is set in motion in circumferential direction. The physical characteristic of flow phenomena in the polar cavity is analysed for different Reynolds numbers and different angles (30^0, 60^0 and 90^0). Based on the mean velocity results, convective heat transfer coefficient in the driven curve cavity is computed. At end of the study, it is expected the convective coefficient increases with respect to the Reynolds numbers and cavity angle. The results are verified based on the numerical solution found in the in the published literature.

ABSTRAK

Penyelesaian secara berangka, analitik mahupun secara ujikaji masalah rerongga kutub memang agak kurang diberi perhatian. Penelitian ini dilakukan adalah untuk memahami gelagat aliran di dalam rerongga tersebut. Salah satu masalah yang dihadapi dalam penyelesaian secara berangka ialah dalam membangunkan domin penyelesaian geometri yang komplek ini. Penyelesaian berangka masalah aliran di dalam reronga kutub ini telah mampu dilaksanakan dengan ANSYS FLUENT. Untuk memudahkan penyelesaikannya aliran bendalir dianggap sebagai gas unggul lagi tidakbolehmampat dengan sifat mekanikal dan termalnya kekal malar terhadap suhu dan tekanan. Semua dinding dinding rerongga dianggap isotermal kecuali dinding atas yang bergerak. Ciri ciri fizikal fenomena aliran di dalam rerongga kutub ini dianalisis pada pelbagai magnitude nombor Reynolds dan sudut bukaan yang berlainan. Berasaskan hasil halaju purata dinding yang bergerak, koeffisien pemindahan haba olakan di dalam rerongga kutub tersebut ditentukan secara berangka. Kajian ini menunjukkan bahawa nilai koeffisien pemindahan haba olakan di dalam rerongga kutub bertambah dengan pertambahan magnitude numbor Reynolds dan sudut bukaan rerongga tersebut. Keputusan ini disahkan kebenarannya melalui perbandingan nilai koeffisien pemindahan haba olakan masalah serupa yang diberi dalam literatur yang telah diterbitkan.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

α - Thermal diffussity

β - Coefficient of thermal expansion

c_p - Specific heat at constant pressure

θ - Cavity angle

υ - Viscosity

ρ - Density

Φ - Dissipation

D - Hydrodynamic length

h - Coefficient of convection

k - Coefficient of conduction

L - Reference length; $L=R_1$

T - Dimensionless temperature

T_w - Wall temperature

T_1 - Entrance temperature

R_1 - Inner radius

R_2 - Outer radius

V_w - Speed of the outer radial wall

u - Radial component of velocity

v - Tangential component of velocity

l - Characteristic length

z - Z-axis

y - Y-axis

x - X-axis

Re - Reynolds number

AP - Aspect ratio

Pr - Prandtl number

Ec - Eckert number
ADI - Alternating-direction implicit
N-S - Navier-stokes
FDM - Finite Different Methods and
FEM - Finite element methods

CHAPTER 1

INTRODUCTION

1.1 Overview

Computational Fluid Dynamics, (CFD) become upsurge of interest by means of high performance computing hardware and user friendly interfaces have led to diverge of fluid analyzing. The CFD techniques have implemented into design, research and development, and manufacture of aircraft and automotive. As the CFD is appropriate a vital component in the flow analysis, driven cavity flow have always been the attention due to the simplicity of geometry and boundary condition. Contempt its simplicity, it is helped to shows attractive flow features as the geometries and Reynolds numbers varies. With available tremendous of computer-based simulation encouraging to analyze the different geometries and varies Reynolds numbers to present different flow features with high accuracy and numerical efficiency. The majority of cavity driven flows for steady incompressible flow were done by numerical studies. These shows discrepancies in studies, however, this study can be continued to computational of heat transfer in the cavity.

1.2 Background Study

The 2-D incompressible cavity flow has been studied numerically and experimentally. The 2-D cavity might be square, rectangular, triangular, and cone shapes. It can be polar cavity. They are numerous important engineering application this gadget. Most of the literatures presented the numerical studies about physical flow patterns inside the cavity for various Reynolds numbers. Among the literatures are Erturk E. and Gokcol O. (2005), Henri C.H and Saad M. (2005), Baytas A.C. and Liaqat A. (2000), Glowinski R., Guidoboni G. and Pan T.-W (2005), Xu H., Zhang C. and Barron R. (2005), Cheng M. and Hung K.C. (2005), Povitsky A. (2005), where each of them present different geometries and the results with different numerical approach.

In 1977, Ghia U. Ghia K.N and Studeruss C.J, computed 3-D laminar incompressible flow in straight polar cavity by numerical method. In the study some assumption have been done which is polar duct’s axial flow direction the governing differential equations are parabolic, while for cross-flow the equation remain as elliptic for the computation of 3-D polar duct. To accomplish the assumption, they have made some limitation on their analysis such that entrance flow Reynolds number cannot be too low and the axial velocity must not be negative values everywhere.

The applications of the cavity flow are turbo machinery flow in the blade passages, heat exchangers and blood flow in the human arterial system. In engineering, examiner really wants to study the fluid motion in a cavity in order to optimize the effectiveness of flow in that cavity. It shows that the driven cavity flow analysis is very important in our daily life.

1.3 Problem Statement

Polar cavity driven flow is very less investigated due to its complex geometry and complex mathematical modeling. Eventually, investigator cannot examine the fluid motion inside of driven cavity. Based on literature, the last was analyzed by Ghiak.U, et al. (1977) and it is not examined for high Reynolds numbers.

Incompressible flow in polar cavity became very subjective to study the fluid motion in engineering application such as in pump's impellers, turbo machinery's blade passages, rotating heat exchangers and blood flow in the human arterial system and etc. Besides, the coefficient of forced convective heat transfers cannot to be compute due to unknown mean velocity at cavity.

With these considerations, a study in polar driven cavity is conducted to study the flow physical and to calculate mean velocity of fluid inside cavity and the rate of heat transfer between fluid and the wall.

1.4 Objective

The objective of this study is to analyze numerically the effect fluid flow physical; especially the mean velocity of fluid on coefficients of convective heat transfer for varies Reynolds no and cavity angles.

1.5 Significance of the Project

One of the important of this project is to study the physical properties of flow and understand the rate of heat transfer from body to fluid. Most of today's technologies operates with higher frequency (rotation or translation or even in microprocessor) which dissipate heats. The faster heat release system helps a system to operate effectively and efficiently. Besides, this numerical study contributes to development department of design to create very effective channel to release heat from particular engineering application, especially turbo machinery and motors. With this particular design concept will improve the reliable of component or equipment or an integrated system. To succeed the goal at first have to study the flow physical and the rate of heat transfer at polar cavity.

1.6 Research Scope

This project focuses on the numerical study of convective coefficient of heat transfer in polar cavity, where the flow profile is taken for consideration to achieve the main objective. The research scopes of this project are as follow;

Design criteria

I. Development of polar cavity as discussed:
Cavity angles (30^0, 60^0, 90^0); dimensionless radius is constant.

II. Identify data input for the software (Gambit and Fluent)

Flow analysis:

III. Analyze the flow physical characteristic; the velocity profile in x and y direction as Reynolds numbers increases.

IV. Analyze the flow in regions with corners and curved boundaries.

Thermal analysis:

V. Analyze temperature distribution inside the cavity

VI. Numerically compute the coefficient of heat transfer for various Reynolds number and cavity angles.

1.7 Research Methodology

To complete this project, few important steps need to be followed. The steps are:

I. Data collection
II. Polar cavity design for angle (30^0, 60^0 ,90^0)
III. Reynolds number ranging (0, 100, 200, 500, 800, 1000, 1200, 1500)
IV. Boundary condition settings
V. Development of meshing
VI. Identify the convergence of results
VII. Analyse velocity profile, stream line and eddy at corners
VIII. Do numerical analysis to calculate mean velocity in cavity
IX. Analyse thermal distribution in cavity
X. Compute the coefficient convective heat transfer
XI. Repeat steps, III to X for greater angle
XII. Discuss the flow phenomena inside cavity, thermal distribution and at last not least the effect of Reynolds number and cavity angle on coefficient of convective heat transfer
XIII. Conclusion

REFERENCES

A.Liaqat and A.C.Baytas (2000). Characteristic of conjugate free convection in a corium pool formed during a serve reactor accident. *Science Direct*.

A.Liaqat and A.C.Baytas (2001). Numerical comparison of conjugate and non-conjugate natural convection for internally heated semi-circular pools. *Science Direct*.

A.Povitsky (2005). Three-dimensional flow in cavity at yaw. *Science Direct*.

C.A. Long, N.D.D. Miché, P.R.N. Childs (2007). Flow measurements inside a heated multiple rotating cavity with axial throughflow. *Science Direct.*

C.H.Bruneau and M.Saad (2005). The 2D lid-driven cavity problem revisited. *Science Direct*.

C. Lei, L. Cheng, K. Kavanagh (1999). A finite difference solution of the shear flow over a circular cylinder. *Science Direct.*

E.Erturk (2005). Discussions on driven cavity flow and steady solutions at high Reynolds numbers. *Science Direct.*

E.Erturk and O.Gokcol (2005). Fine grid benchmark solutions of triangular cavity flow. *International Journal of Numerical Methods for Heat & Fluid Flow.*

Frank P. Incropera. *Heat Transfer*. 6th Edition International Student Version. Wiley. 2011.

Hatice Mercan, Kunt Atalık (2008).Vortex formation in lid-driven arc-shape cavityflows at high Reynolds numbers. *Science Direct.*

H. Xu, C. Zhang and R. Barron (2005). Numerical simulation of cavity flows based on transformed equations. *Science Direct*.

Lin Mu, Xiu Ye (2011). A finitevolumemethod for solving Navier‐Stokes problems.

Science Direct.

M. Cheng and K.C. Hung (2005). Vortex structure of steady flow in a rectangular cavity. *Science Direct*.

R.Glowinski, G.Guidoboni and T.W.Pan (2005). Wall-driven incompressible viscous flow in a two-dimensional semi-circular cavity. *Science Direct*.

Suhas N. Patankar. *Numerical Heat Transfer and Fluid Flow.* Taylor& Francis. 1980.

Timothy J. Baker (2005). Mesh generation: Art or science? *Science Direct*.

U.Ghia and K. Ghia (1977). Three-dimensional laminar incompressible flow in straight polar ducts. *Science Direct*.

U.Ghia and R.K.Goyal (1976). A Study of Laminar Incompressible recirculating flow in a driven cavity of polar cross-section. *In Numerical~Laboratory Computer Methods in Fluid Mechanics.*

Yunus A. Cengel & Michael A. Boles. *Thermodynamic.* Fourth Edition. McGraw-Hill.