CORE
Metadata, citation and similar papers at core.ac.uk
Provided by UTHM Institutional Repository

SOLUTION TO NAVIER - STOKES EQUATION IN STRETCHED COORDINATE SYSTEM

MOHD ZAMANI BIN NGALI

UNIVERSITI TEKNOLOGI MALAYSIA

PERPUSTAKAAN UTHM
30000001866473

cy890/2

PSZ 19:16 (Pind. 1/97)

UNIVERSITI TEKNOLOGI MALAYSIA

BORANG PENGESAHAN STATUS TESIS[υ]

JUDUL : **SOLUTION TO NAVIER-STOKES EQUATION IN STRETCHED COORDINATE SYSTEM**

SESI PENGAJIAN : 2005/2006

Saya **MOHD ZAMANI BIN NGALI**

(HURUF BESAR)

mengaku membenarkan tesis (~~PSM~~/Sarjana/~~Doktor Falsafah~~)* ini disimpan di Perpustakaan Universiti Teknologi Malaysia dengan syarat-syarat kegunaan seperti berikut :

1. Tesis adalah hak milik Universiti Teknologi Malaysia.
2. Perpustakaan Universiti Teknologi Malaysia dibenarkan membuat salinan untuk tujuan pengajian sahaja.
3. Perpustakaan dibenarkan membuat salinan tesis ini sebagai bahan pertukaran di antara institusi pengajian tinggi.
4. **Sila tandakan (✓)

☐	SULIT	(Mengandungi maklumat yang berdarjah keselamatan atau kepentingan Malaysia seperti yang termaktub di dalam AKTA RAHSIA RASMI 1972)
☐	TERHAD	(Mengandungi maklumat TERHAD yang telah ditentukan oleh organisasi/ badan di mana penyelidikan dijalankan)
✓	TIDAK TERHAD	

(TANDATANGAN PENULIS)

Alamat Tetap:

LOT 683, LORONG HJ. NOH,
KAMPUNG SUNGAI TIRAM,
81800 ULU TIRAM,
JOHOR DARUL TAKZIM.

Tarikh :
28 NOVEMBER 2005

(TANDATANGAN PENYELIA)

Nama Penyelia:

DR. KAHAR BIN OSMAN

Tarikh :
28 NOVEMBER 2005

CATATAN : * Potong yang tidak berkenaan.
** Jika Tesis ini SULIT atau TERHAD, sila lampirkan surat daripada pihak berkuasa/ organisasi berkenaan dengan menyatakan sekali tempoh tesis ini perlu dikelaskan sebagai SULIT atau TERHAD.
υ Tesis dimaksudkan sebagai tesis bagi Ijazah Doktor Falsafah dan Sarjana secara penyelidikan, atau disertasi bagi pengajian secara kerja kursus dan penyelidikan, atau Laporan Projek Sarjana Muda (PSM).

UTM(PS)-1/02

School of Graduate Studies

Universiti Teknologi Malaysia

VALIDATION OF E-THESIS PREPARATION

Title of the thesis : **Solution To Navier-Stokes Equation In Stretched Coordinate System**

Degree: **Master of Engineering (Mechanical-Pure)**

Faculty: **Mechanical Engineering**

Year: **2005**

I **MOHD ZAMANI BIN NGALI**

(CAPITAL LETTER)

declare and verify that the copy of e-thesis submitted is in accordance to the Electronic Thesis and Dissertation's Manual, School of Graduate Studies, UTM

(Signature of the student)

Permanent address:

Lot 683, Lorong Hj. Noh, Sg. Tiram,

81800 Ulu Tiram,

Johor Darul Takzim

(Signature of supervisor as a witness)

Name of Supervisor: **Dr. Kahar Bin Osman**

Faculty: **Mechanical Engineering**

Note: This form must be submitted to SPS together with the CD.

"I/We hereby declare that we have read this thesis and in my/our opinion this thesis is sufficient in terms of scope and quality for the award of the degree of Master of Engineering (Mechanical)".

Signature	: ....
Name of Supervisor	: Dr. Kahar Bin Osman
Date	: 28 November 2005

SOLUTION TO NAVIER-STOKES EQUATION IN STRETCHED COORDINATE SYSTEM

MOHD ZAMANI BIN NGALI

A thesis submitted in fulfilment of the
requirements for the award of the degree of
Master of Engineering (Mechanical)

Faculty of Mechanical Engineering
University of Technology Malaysia

28 NOVEMBER 2005

"I hereby declare that this thesis entitled 'Solution To Navier-Stokes Equation In Stretched Coordinate System' is the result of my own research except those cited in references."

Signature	: ....
Name of Author	: MOHD ZAMANI BIN NGALI
Date	: 28 NOVEMBER 2005

To my beloved family,

The lover in you who brings my dreams comes true.

To my baby Lubna, who have brought a new level of love, patience and understanding into our lives.

ACKNOWLEDGEMENTS

I would like to express my sincere appreciation to Dr. Kahar Bin Osman for all his encouragement, support and guidance during the course of this Master project. Thanks also to all the lecturers involved during the accomplishment of this project. Not forgotten all my colleagues for their time, encouragement and support.

A heart felt gratitude to my parents and my brother for being very understanding, and also for giving my own family their helping hand every time needed through out the course. And last but not least, a special thanks to my wife, Junita and my doughter, Lubna for being there with me for all the cry and laughter in completing this project.

ABSTRACT

Solution to Navier-Stokes equation by Splitting method in physical orthogonal algebraic curvilinear coordinate system, also termed *'stretched coordinate'* is presented. The unsteady Navier-Stokes equations with constant density are solved numerically. The linear terms are solved by Crank-Nicholson method while the non-linear term is solved by the second order Adams-Bashforth method. The results show improved in comparison of efficiency and accuracy with benchmark steady solution of driven cavity by Ghia et al. and other first order differencing schemes including splitting scheme in Cartesian coordinate system. Enormous improvements from the original Splitting method in Cartesian coordinate observed where accurate solutions are obtained in minimum 17 X 17 from 33 X 33 resolution for Re = 100, 47 X 47 from 129 X 129 resolution for Re = 400 and 65 X 65 from 259 X 259 resolution for Re = 1000.

CONTENTS

LIST OF TABLES

LIST OF FIGURES

CHAPTER I

INTRODUCTION

1.1 Overview

Fluid dynamics essentially deals with motion of liquids and gases, which appear to be continuous in its macroscopic structure. All the variables are considered to be continuous functions of spatial coordinates and time. The Navier-Stokes equations are able to model weather or the movement of air in the atmosphere, ocean currents, water flow in a pipe, as well as many other fluid flow phenomena.

The original Navier-Stokes equations are directly simplified by an assumption of constant density. Another simplification that commonly applied in construction of computational solution is to set all changes of fluid properties with time to zero. This is called steady solution where the Navier-Stokes equations become simpler with only steady forms are considered. A problem is termed steady or unsteady depending on the frame of reference. For instance, the flow around a ship in a uniform channel is said to be steady from the passengers' point of view, but unsteady by observers on the shore. Fluid dynamicists often transform problems to frames of reference in which the flow is steady in order to simplify the problem.

Over the last three decades, the use of CFD techniques in solving fluid flow and its applications has grown from being able to model only steady single phase, low Reynolds number flows to its current level of use in a wide range of applications. This level of growth has been enhanced by the advances in computer technology which have vastly reduced the computational times for all computations and simulations as well as increasing the size of problems which can be solved.

The application of Navier-Stokes equation in solving fluid flow has also evolved throughout this period of time with numerical method as one of the most inspiring technique that been explored. Numerical methods for 2-D steady incompressible Navier-Stokes (N-S) equations are often tested for code validation on a very well known benchmark problem, the lid-driven cavity flow. Due to the simplicity of the cavity geometry, applying a numerical method on this flow problem in terms of coding is quite easy and straight forward. Despite its simple geometry, the driven cavity flow retains a rich fluid flow physics manifested by multiple counter rotating re-circulating regions on the corners of the cavity depending on the Reynolds number. In the literature, it is possible to find different numerical approaches which have been applied to the driven cavity flow problem.

Amongst the numerous studies that use different types of numerical methods on the driven cavity flow found in the literature, priority is given for comparable methods with first order accuracy discretization scheme, Reynolds number ranging from 100 to 1000 and employ either Cartesian or algebraic stretched grid only. Some of the comparison works are the Upwind scheme, first suggested by Courant, Isaacson and Rees [10], the hybrid scheme, developed by Spalding [11], the power law scheme, described by Patankar [12] and the exponential scheme , also described by Patankar[9].

Apart from that, literature review also shows that many works have been done on the Navier-Stokes equation especially for steady, highly accurate solution which can be used as accuracy comparison. Barragy & Carey [15] have used a *p*-type finite element scheme on a 257 ×257 strongly graded and refined element mesh. They have obtained a highly accurate (Δh^8 order) solutions for steady cavity flow solutions up to Reynolds numbers of Re=12,500. Wright & Gaskell [16] have applied the Block Implicit Multigrid Method (BIMM) to the SMART and QUICK discretizations. They have presented cavity flow results obtained on a 1024 ×1024 grid mesh for Re < 1,000. Liao & Zhu [17], have used a higher order streamfunction-vorticity boundary element method (BEM) formulation for the solution of N-S equations. They have presented solutions up to Re=10,000 with grid mesh of 257 ×257. Ghia et. al. [1] have applied a multi-grid strategy to the coupled strongly implicit method. They have presented solutions for Reynolds numbers as high as Re=10,000 with meshes consisting of as many as 257 ×257 grid points. Results by Ghia et. al. has frequently used as the benchmark solution of cavity flow.

The use of Curvilinear Grids, also termed Body Fitted Coordinates (BFC), allows the physical domain to be accurately fitted for a large number of cases. The mapping of these grids onto their topologically equivalent Cartesian mesh, with the associated mapping of the transport equations, extends the class of problems to which the numerical method technique can be applied. A similar methodology, in which the transformation to a computational domain is implicit in the discretisation techniques, has been used by Demirdzic and Peric [7] and many other researchers to solve problems with moving boundaries. The problems with this type of approach are that the use of BFC meshes increases the storage requirements and adds considerably to the complexity of the equations being solved. The approximations made to calculate the various terms become significantly more difficult to calculate. This commonly leads to further approximations being made and as a consequence errors become significant if the physical grid differs substantially from the computational Cartesian mesh.

Since this current work is only concern on square driven cavity, algebraic orthogonal curvilinear coordinate or simply termed, 'Stretched Coordinate' is used. Stretched coordinate is selected because it enables direct usage of mathematical models derived in Cartesian coordinate with minimum verifications of the discretization methods. Stretched coordinate also enables mesh clustering that serves very well for lid-driven cavity problem. Further explanation on the advantages of having stretched grid is discussed in section 2.5.

In two dimensional solution of viscous incompressible flow, the pressure term can be eliminated by taking the cross derivative of the momentum equation. The pressure term can also be taken under consideration by velocity-pressure coupling techniques. Some of the popular velocity-pressure coupling methods are Artificial Compressibility method, Fractional-Step method and Pressure Poisson Equation method. The most commonly used velocity-pressure coupling technique is SIMPLE (Semi-Implicit Method for Pressure-Linked Equation). This technique is found to be inefficient since it involve major convergence iteration in determining the pressure values for every main velocity-time iteration. As an alternative, Karniadakis [2] had introduced a new formulation for high-order time-accurate splitting scheme for the solution of the incompressible Navier-Stokes equations.

The pressure in incompressible flow plays a very important particular role as it should always be in equilibrium with the time-dependent divergence-free velocity field, but it does not appear explicitly in the equation imposing such a divergence condition. While it is clear that the governing equation for pressure is a Poisson equation derived from the momentum equation by requiring incompressibility, it is less clear what boundary conditions the pressure should be subject to. In particular, it was argued that in the absence of singularities as time approaching zero value, property derived Neumann and Dirichlet boundary conditions lead to the same solution. However, Neumann boundary conditions are more general and always provide a unique solution for time approaching zero.

In Splitting method which is the method used in this current work, the pressure satisfies a Poisson equation with compatible Neumann boundary conditions. The exact form of this boundary condition is very important not only because it directly affect the overall accuracy of the scheme, but also because it determines the accuracy of the time-stepping algorithm. This is particularly true in simulations of unsteady flows in complex geometry, where a separately solvable second-order pressure equation is still the only affordable approach. In this current work, splitting led to first order accuracy, so that very small time increment steps are required in order to prevent significant time differencing and splitting errors.

In particular, improved pressure boundary conditions of high order in time are introduced for minimum effect of erroneous numerical boundary. A new family of stiffly stable schemes is employed in mixed explicit/implicit time integration rules. These schemes exhibit much broader stability regions as compared to traditional Adam-family schemes. The stability properties remain almost constant as the accuracy of the integration increases, so that robust third or higher order time accurate schemes can readily be constructed.

1.2 Objective

A recent attempt to implement Splitting method introduced by Karniadakis et. al [2] in algebraic orthogonal curvilinear coordinate is motivated by the necessity to obtain more accurate and efficient first order accuracy solution of Navier-Stokes equation. First order accuracy scheme is the simplest scheme required for unsteady solution of Navier-Stokes equation. Since efficiency is the most commanding issue in unsteady solution, it is always worthwhile to have less time consuming scheme without sacrificing the accuracy of the solution.

The current work is meant to bring together the advantage of Splitting method as pressure-velocity solver of higher efficiency with the advantage of consuming stretched grid which produce more accurate results in relatively equal number of grid points as compared to Cartesian grid.

The main objectives of the current work can be arranged in more perceptible agreement as below:

i. To develop less mesh sensitive and more efficient numerical Algorithm for unsteady two-dimensional incompressible Navier-Stokes equation.

ii. To introduce Splitting as velocity-pressure coupling method on physical orthogonal algebraic curvilinear coordinates, also termed *'stretched coordinate'* in solving Navier-Stokes equation.

iii. To study the behavior of the developed algorithm in terms of time efficiency, mesh sensitivity, accuracy and its robustness.

iv. To compare the results obtained with previously published results for the traditional driven cavity problem.

CHAPTER II

NUMERICAL SOLUTION TECHNIQUES

2.1 Introduction to Splitting method

In two dimensional solution of viscous incompressible flow, the pressure term can be eliminated by taking the cross derivative of the momentum equation. The pressure term can also be taken under consideration by velocity-pressure coupling techniques. Some of the popular velocity-pressure coupling methods are Artificial Compressibility method, Fractional-Step method and Pressure Poisson Equation method.

The most commonly used velocity-pressure coupling technique is SIMPLE (Semi-Implicit Method for Pressure-Linked Equation). This technique is found to be inefficient since it involves major convergence iteration in determining the pressure values for every main velocity-time iteration.

As an alternative, Karniadakis [2] had introduced a new formulation for high-order time-accurate splitting scheme for the solution of the incompressible Navier-Stokes equations.

The pressure in incompressible flow plays a very important particular role as it should always be in equilibrium with the time-dependent divergence-free velocity field, but it does not appear explicitly in the equation imposing such a divergence condition. While it is clear that the governing equation for pressure is a Poisson equation derived from the momentum equation by requiring incompressibility, it is less clear what boundary conditions the pressure should be subject to. In particular, it was argued that in the absence of singularities as time approaching zero value, property derived Neumann and Dirichlet boundary conditions lead to the same solution. However, Neumann boundary conditions are more general and always provide a unique solution for time approaching zero.

In Splitting method which is the method used in this current work, the pressure satisfies a Poisson equation with compatible Neumann boundary conditions. The exact form of this boundary condition is very important not only because it directly affect the overall accuracy of the scheme, but also because it determines the accuracy of the time-stepping algorithm. This is particularly true in simulations of unsteady flows in complex geometry, where a separately solvable second-order pressure equation is still the only affordable approach. In this current work, splitting led to first order accuracy, so that very small time increment steps are required in order to prevent significant time differencing and splitting errors

In particular, improved pressure boundary conditions of high order in time are introduced for minimum effect of erroneous numerical boundary. A new family of stiffly stable schemes is employed in mixed explicit/implicit time integration rules. These schemes exhibit much broader stability regions as compared to traditional Adam-family schemes. The stability properties remain almost constant as the accuracy of the integration increases, so that robust third or higher order time accurate schemes can readily be constructed.

2.2 *Mathematical preliminaries*

Consider a Newtonian flow with constant material properties, including constant density, governed by the Navier-Stokes and continuity equations. The Navier-Stokes equations for constant density flow, in vector form, are

$$\rho\left(\frac{\partial \bar{v}}{\partial t}+\bar{v}\cdot\nabla\bar{v}\right)=-\nabla p+\mu\nabla^2\bar{v}, \qquad 2.2.1$$

where

$$\bar{v}=u\bar{i}+v\bar{j}+w\bar{k} \qquad 2.2.2$$

is the velocity vector, p is the pressure, μ is dynamic viscosity, ρ is fluid density, and t is time.

The continuity equation for constant density is

$$\nabla\cdot\bar{v}=0 \qquad 2.2.3$$

Consider two-dimensional flow in a rectangle of height, H, and length, L. Dimensionless variables are defined as