

Multigrid Computation of Steady and Time-Dependent Incompressible Viscous Flows

A thesis
submitted in partial fulfillment of the
requirements for the degree of
Doctor of Philosophy

by

D. Santhosh Kumar

(Roll No. 05610304)



DEPARTMENT OF MECHANICAL ENGINEERING

Indian Institute of Technology Guwahati

GUWAHATI

OCTOBER 2010

CERTIFICATE

It is certified that the work contained in the thesis entitled **Multigrid Computation of Steady and Time-Dependent Incompressible Viscous Flows** by D. Santhosh Kumar, a student of the Department of Mechanical Engineering, Indian Institute of Technology Guwahati, for the award of the degree of Doctor of Philosophy has been carried out under our supervision and that this work has not been submitted elsewhere for a degree.

Dr. Anoop Kumar Dass
Professor
Department of Mechanical Engineering
IIT Guwahati
Guwahati - 781039, India

Dr. Anupam Dewan (on lien from IIT Guwahati)
Professor
Department of Applied Mechanics
IIT Delhi
New Delhi - 110016, India



Dedicated to
my Grand Mother
Late Smt. Thulasi Ammal

ACKNOWLEDGEMENT

I express my deepest gratitude and hearty thanks to my supervisors Dr. Anoop K Dass and Dr. Anupam Dewan, for their invaluable guidance and constant encouragement. I am deeply inspired by not only their rich experience and knowledge but the way of approaching a problem, immense patience and utmost care. I am highly indebted to them for their untiring devotion and willingness. During my project work, they provided me with the most invaluable ideas, books and journals that are incorporated in the thesis played most important role in completing the thesis successfully. I enjoyed each and every moment working under their supervision and learnt a lot many things from them, which will be an asset for my future research.

I express my sincere gratitude and hearty thanks to Dr. Manab Kumar Das who guided me for the first two years of my research period. I am thankful to my doctoral committee members Dr. A.K. De, Dr. J. C. Kalita and Dr. Anugrah Singh for giving their valuable suggestions and encouragement which helped to shape myself to meet the requirement of a successful researcher. I would like to express my sincere thanks to Prof. D. Chakraborty (H.O.D Mechanical Engineering Department) for providing all the facilities needed during my research work and financial support extended by the Mechanical Engineering Department. I take this opportunity to thank all the faculty members for valuable suggestions and cooperation during my course work. I specially thank Mr. Amal Kalita providing good facilities in CAD lab and providing me a separate computer for my research work.

I am sincerely thankful to the institute for providing me with sufficient funds and grants to have this research work done successfully. I am thankful to staff of Computer Center who have provided me the computational facilities. I am thankful to other faculty members and staff directly or indirectly motivated me to complete the thesis work.

I would like to express my sincere gratitude to my parents Mr. A. Deenadayalan and Mrs.

D. Nalini, my brothers Mr. D. Jagannath and Mr. D. Kamalakkannan and sister-in-laws Mrs. J. Kalyani and Mrs. K. Rekha whose blessings, prayers and never ending support is the real impetus that continuously motivates me to produce my best.

I should take this opportunity to thank my friend Mr. S. Jayavel who insisted me to apply for Ph.D in IIT Guwahati.

I am sincerely thankful to Mr. E. Vishnu, Mr. K. Suresh Kumar and Mr. M. Yousuf for teaching me the basic ideas of CFD and lots of useful discussions during my research work. I would like to thank Mr. S. P. Lakshmanan and Mr. A. Satheesh, with whom I had academic discussions, shared my personal feelings and they provided constant encouragement and continuous help during my entire stay in IIT Guwahati. I am very much thankful to Mr. D. Senthil and Mr. Krishnamoorthy for providing various softwares. I am thankful to Mr. C. Subramanian, Mr. Suresh Pandian, Mr. M. Eswaran, Mr. Siva Pradeep, Mr. Shyam, Mr. Anbu, Mr. Ramakrishnan, Mr. P. Vijay, Mr. Monash, Mr. D.A. Perumal, Mr. Gopinath , Mr. P. Saravanan, Mr. Anto Pradeep, Mr. Suresh Babu, Mr. Ratnakar Das, Mr. Anandraj, Mr. Ashok, Ms. Sumitra Shukla and Ms. Arpananath with whom I had a healthy discussion and making my stay at IIT Guwahati a pleasant one. I would like to thank my friends Mr. S. Jaya Prakash, Mrs. D. Swarnalatha, Mr. Buchibabu, Mrs. Usha, Mr. MuthtamilSelvan and my fiancée Ms. P. Sangeetha whose continuous support and motivation to complete the thesis work.

Abstract

The present thesis deals with the development of multigrid-assisted incompressible viscous-flow solvers for both steady and unsteady flows and their application to several problems of varying complexities to throw light on various aspects of the working of the multigrid method. Some of the applications deal with problems to which multigrid method was not used earlier.

Multigrid methods, which are based on many levels of grids, have the ability to overcome the slow convergence characteristic of the single-grid methods for large-scale problems. This method has been established as a powerful tool for accelerating numerical convergence and, thus reducing the computational time. The two versions of multigrid methods are linear multigrid method, known as the correction scheme applicable to linear equations and nonlinear multigrid method also known as the full-approximation storage (FAS) multigrid method applicable to nonlinear equations.

In the computation of transient viscous flows the pressure-Poisson equation has to be solved accurately at every time-step and it is imperative that this computation is carried out with good time wise efficiency. Use of multigrid here to compute the solution of the pressure-Poisson equation is justified at the beginning by showing that for a heat conduction problem, linear multigrid is better than the point successive overrelaxation (PSOR) method when Gauss-Seidel method is used to solve the discretized algebraic system. In the transient solvers the convective terms are discretized to third-order accuracy and the viscous terms are discretized to fourth-order accuracy, and a fractional-step technique is used to obtain second-order temporal accuracy. Finite-difference discretization with transformation from the physical space to the computational space is carried out on graded Cartesian meshes to obtain better scale resolution in shear layers, so that the transient computations enjoy the advantages of good grid economy provided by the graded mesh and good time efficiency provided by the multigrid. Some of the transient computations include (i) computation of

unsteady and asymptotically obtained steady flow in a single-sided lid-driven square cavity including periodic flows that develop at higher Reynolds numbers like $Re = 8200$ and 10000 as shown here. (ii) Computation of transient flow in a hitherto-unexamined configuration of two-sided lid-driven square cavity that involves gradual development of a free ‘shear layer’ with associated vortices. (iii) Computation of steady flow in the backward-facing step in a channel through time-marching, that shows that 2D flows at $Re = 800$ and 1000 appear to be steady. (iv) Computation of flow past a square prism confined in a channel, which - beyond a certain Reynolds number - involves periodic vortex shedding giving rise to the so-called von Kármán vortex street. Periodicity of the flow is established through phase plots, power spectrum analysis and temporal plots of lift and drag coefficients.

A computer code having similar features as the 2D code described above is also developed to compute 3D flows especially in geometries that allow graded Cartesian meshes to resolve sharp gradients in shear layers by locally clustering the grid there. First, computations of transient and steady flows are carried out for the single-sided lid-driven cubical cavity. The flow patterns are studied at different locations and planes. The time evolution of Taylor-Gortler like vortices is also captured at different instants of time. After having used the code to compute the flow in the single-sided cubical cavity to establish its credibility, it is then used to compute the transient and steady-state flows in a hitherto unexplored configuration, namely, the two-sided cubical cavity flow induced by the motion of two facing walls in the same direction. The grid here is clustered not only near the walls but also in the free-shear layer midway between the moving walls to obtain a good grid-economy. Various observations are made about this hitherto untested flow configuration. Though not tested, the 3D transient code appears to have the potential of being used for DNS of flows in some Cartesian geometries.

Also developed is a full-approximation storage (FAS) based multigrid solver for the 2D Navier-Stokes equations. The code is based on the SIMPLE algorithm on a colocated grid arrangement using the finite volume method. The transport equations are discretized using the QUICK scheme for the convective terms and the central-difference scheme for the viscous

and the pressure gradient terms. The code is first validated against the existing results for a single-sided lid-driven square cavity flow. It is then applied to the two- and four-sided lid-driven cavity flows that involve multiple steady solutions. In the two-sided square cavity two of the adjacent walls move with an equal velocity and in the four-sided cavity all the four walls move in such a way that parallel walls move in the opposite directions with the same velocity. The strategy employed to obtain the nonunique steady solutions is also described in the thesis.

Further we have investigated the relative performance of two models, namely Brinkman-extended Darcy model (BDM) and Brinkman-Forchheimer Darcy model (BFDM) in a problem involving mixed convection in a square cavity filled with a porous medium using the FAS multigrid method. The left and right walls of the cavity filled with a porous medium, moving in the opposite directions, are maintained at different constant temperatures while the top and bottom walls are thermally insulated. The direction of the plate movements have been so chosen as to oppose the buoyancy effects so that interesting flow patterns develop especially when both the effects have similar levels of influence.

Results of the computations are carefully substantiated through a comparison exercise wherever possible. Reliability of the computational results for untested flow configurations are commented upon. The performance of the linear and nonlinear multigrid is quantified through speed-ups given in tables at several places.

Contents

1	Introduction	1
1.1	Background	1
1.2	Literature Review on Multigrid	3
1.3	Motivation and Objectives	6
1.4	The Present Thesis	7
1.5	Organization of the Thesis	9
2	Multigrid Theory	10
2.1	Introduction	10
2.2	Linear Two-Grid Algorithm	13
2.2.1	Restriction and Prolongation	14
2.3	Multi-level Scheme	20
2.4	Multigrid Implementation to Heat Conduction Problem	22
2.5	Results and Discussions	25
2.6	Non-linear Multigrid Method	28
3	A Multigrid-Accelerated Code on Graded Cartesian Meshes for 2D Time-Dependent Incompressible Viscous Flows	31
3.1	Introduction	31
3.2	Governing Equations in the Physical Plane	35
3.2.1	Governing Equations in the Transformed Plane	36
3.3	Fractional-Step Procedure	37

3.3.1	Poisson Solver	38
3.4	Numerical Method	39
3.5	Computations, Results and Discussions	40
3.5.1	Transient Flow in a Single-Sided Lid-Driven Square Cavity	41
3.5.2	Transient Flow in a Two-Sided Lid-Driven Square Cavity	56
3.5.3	Backward Facing Step Flow	62
3.5.4	Flow Past a Square Prism	67
3.6	Conclusions	79
4	A Multigrid-Accelerated Three-Dimensional Transient-Flow Code and its Application to a New Test Problem	81
4.1	Introduction	81
4.2	Governing Equations	83
4.2.1	Governing Equations in Transformed Plane	84
4.3	Computations, Results and Discussions	85
4.3.1	Transient Flow in a Single-Sided Lid-Driven Cubic Cavity	85
4.3.2	Two-Sided Lid-Driven Cubic Cavity: A New Test Problem	105
4.4	Conclusions	111
5	Multiple Stable Solutions for Two- and Four-Sided Lid-Driven Cavity Flows using FAS Multigrid Method	113
5.1	Introduction	113
5.2	Governing Equations and Numerical Method	115
5.2.1	Discretization Method	116
5.2.2	Description of Multigrid Algorithm	117
5.3	Results and Discussion	124
5.3.1	Application to Lid-Driven Cavity Flow	124
5.3.2	Multiple Solution for Two- and Four-Sided Lid-Driven Cavity Flows .	129
5.3.3	Multigrid Performance	141

5.4	Conclusions	142
6	Analysis of Non-Darcy Models for Mixed Convection in a Porous Cavity using Multigrid Approach	145
6.1	Introduction	145
6.2	Problem Description	147
6.3	Numerical Method	150
6.4	Results and Discussion	152
6.4.1	Flow Due to Forced Convection	154
6.4.2	Flow Due to Natural Convection	166
6.5	Conclusions	172
7	Summary and Conclusions	174
A	Transformation of Governing Equations	178
B	Derivation of Pressure-Poisson Equation	180
C	Power Spectrum Analysis	181

List of Figures

2.1	(a) Fine grid (b) Coarse grid.	12
2.2	Grid arrangement in two-level scheme.	13
2.3	Location of fine and coarse grid points for a 2D square domain.	15
2.4	Damping of errors using a 2-level (64 and 32 points) multigrid method. . . .	19
2.5	(a) V-Cycle (b) W-Cycle (c) FMV-Cycle.	21
2.6	Schematic representation of two-dimensional square rod.	22
2.7	Grid arrangements in the 4-level multigrid method.	23
2.8	(a) Temperature distribution for a 2D heat conduction problem on 129×129 grids (b) residual history for different combinations of 2-level MG (c) effects of multigrid levels (d) Comparison of residual fall for GS, SOR and MG method. 27	
3.1	Schematic diagram of single-sided lid-driven square cavity.	41
3.2	A 65×65 grid in (a) physical plane and (b) computational plane.	43
3.3	4-level V-cycle multigrid.	43
3.4	Convergence of multigrid on a 129×129 grid for $Re = 3200$ at $t = 0.1$: (a) effect of multigrid levels, (b) pressure error history.	44
3.5	Speed-up for multigrids with a 2- and 4(optimum)-level.	44
3.6	Time-step-independence study: (a) horizontal velocity along vertical centre- line and (b) vertical velocity along horizontal centreline.	45
3.7	Tertiary vortex at the bottom right corner of the cavity at $Re = 1000$: (a) uniform grid and (b) nonuniform grid.	46

3.8	Comparison of steady-state velocity profiles: (a) horizontal velocity along the vertical centreline and (b) vertical velocity along the horizontal centreline for the single-sided lid-driven cavity.	47
3.9	Comparison of steady-state centreline u and v profiles with Botella and Peyret [11] for the single-sided lid-driven cavity at $Re = 1000$	47
3.10	Time evolution of streamlines for the single-sided lid-driven square cavity at $Re = 1000$	49
3.11	Steady-state streamline patterns for the single-sided lid-driven square cavity on a 129×129 clustered grid.	50
3.12	Streamline pattern for $Re = 8200$	51
3.13	(a) Horizontal velocity history and (b) power spectrum at monitoring point $(14/16, 2/16)$ for $Re = 8200$	52
3.14	(a) Horizontal velocity history , (b) power spectrum and (c) phase portrait on 129×129 grid at monitoring point $(2/16, 14/16)$ for $Re = 10000$	53
3.15	Evolution of streamlines during one time period for $Re = 10000$ on grid 129×129	53
3.16	Two-sided lid-driven square cavity flow: (a) flow configuration (b) grid (129×129).	57
3.17	Steady-state streamline patterns for different Reynolds numbers for the two-sided lid-driven square cavity on a 129×129 clustered grid.	58
3.18	Steady-state velocity profiles: (a) horizontal velocity along the vertical centreline and (b) vertical velocity along the horizontal line at $y=0.75$, on a 129×129 clustered grid.	59
3.19	Time evolution of streamlines for the two-sided lid-driven square cavity at $Re = 1000$	61
3.20	Configuration of backward-facing step problem.	63
3.21	Steady-state streamlines for the backward-facing step flow at various Reynolds numbers.	64

3.22	Steady-state velocity-vector plots for the backward-facing step flow at various Reynolds numbers.	65
3.23	The reattachment length as a function of Reynolds number.	66
3.24	Comparison of u profile for $Re = 800$ at $x = 7$ and $x = 15$ with the result of Gartling [40] and Kalita and Gupta [58].	67
3.25	Steady-state pressure contours for the backward-facing step flow at various Reynolds number.	68
3.26	Pressure distribution along upper and lower channel walls at $Re = 800$	69
3.27	At $Re = 1000$, steady-state: (a) streamline pattern (b) velocity-vector plot (c) pressure contours.	69
3.28	(a) Configuration of the flow past a square prism (b) Grid arrangement for flow past a square prism.	71
3.29	Streamline patterns for flows past a square prism at different Reynolds numbers.	72
3.30	Vorticity contours for flows past a square prism at different Reynolds numbers.	73
3.31	Temporal evolution of streamline patterns for the flow over a square prism at $Re = 100$ over one period.	75
3.32	Temporal evolution of vorticity contours for the flow over a square prism at $Re = 100$ over one complete vortex shedding cycle.	76
3.33	(a) Variation of drag and lift coefficients, (b) power spectrum and (c) phase diagram at monitoring point $(53/4, 14/4)$ for $Re = 100$	78
4.1	(a) Schematic of lid-driven cubic cavity (b) nonuniform grid arrangement for lid-driven cubic cavity.	86
4.2	Pressure error history at $t = 0.1$ for the Poisson equation at $Re = 1000$	87
4.3	Grid-independence study: steady-state (a) horizontal velocity along vertical centreline (b) vertical velocity along horizontal centreline, at $z = 0.5$ and $Re = 1000$ for the single-sided cubic cavity.	88

4.4	Time-step independence study: (a) horizontal velocity along vertical centreline (b) vertical velocity along horizontal centreline at $z = 0.5$ and $Re = 1000$ for the single-sided cubic cavity.	89
4.5	Comparison of steady-state (a) horizontal velocity along the vertical centreline (b) vertical velocity along the horizontal centreline for $Re = 100, 400$ and 1000 at $z = 0.5$ for the single-sided cubic cavity with Ku <i>et al.</i> [67].	90
4.6	Comparison of steady-state centreline u and v profiles with Albensoeder and Kuhlmann [2] for the single-sided cubic cavity at $Re = 1000$	91
4.7	Evolution of projected streamlines for the single-sided cubic cavity at different instants of time at $Re = 400$ and $z = 0.5$	93
4.8	Evolution of projected streamlines for the single-sided cubic cavity at different instants of time at $Re = 1000$ and $z = 0.5$	94
4.9	Steady state projected streamline patterns for the single-sided cubic cavity for different Reynolds numbers at symmetry plane $z = 0.5$	95
4.10	Steady state velocity vectors for the single-sided cubic cavity for different Reynolds numbers at symmetry plane $z = 0.5$	95
4.11	Steady state pressure contours for the single-sided cubic cavity for different Reynolds numbers at symmetry plane $z = 0.5$	96
4.12	Comparison of 2D and 3D centreline velocity profiles for the single-sided cubic cavity for u and v at different instants of time at $Re = 400$ and $z = 0.5$	97
4.13	Comparison of 2D and 3D centreline velocity profiles for the single-sided cubic cavity for u and v at different instants of time at $Re = 1000$ and $z = 0.5$. . .	98
4.14	Comparison of steady state velocity profiles for the single-sided cubic cavity at different span-wise aspect ratios for $Re = 400$ and $Re = 1000$	99
4.15	Comparison of mid-span ($z = 0.5$) steady-state streamline pattern for the single-sided cubic cavity for span-wise aspect ratio 1:1 and 2:1.	100
4.16	Projected streamline patterns for the single-sided cubic cavity in different x -planes at $Re = 1000$	102

4.17	Projected streamline patterns for the single-sided cubic cavity in different y -planes at $Re = 1000$	102
4.18	Projected streamline patterns for the single-sided cubic cavity in different z -planes at $Re = 1000$	102
4.19	Time evolution of projected streamline patterns for the single-sided cubic cavity at $x = 0.5$ and $Re = 1000$	103
4.20	Time evolution of velocity vectors for the single-sided cubic cavity at $x = 0.5$ and $Re = 1000$	104
4.21	Comparison of steady-state centreline u and v profiles with Beya and Lili [10] for a two-sided non-facing lid-driven cubic cavity on a $65 \times 65 \times 65$ grid at $Re = 500$	106
4.22	2D view of the grid in the z -plane and steady-state projected streamline patterns for different Re 's for the facing-wall two-sided cubic lid-driven cavity flow on a $65 \times 65 \times 65$ grid.	107
4.23	Steady-state (a) Horizontal velocity along the vertical centreline (b) Vertical velocity along the horizontal centreline for the facing-wall two-sided cubic lid-driven cavity flow for $Re = 100, 400$ and 600 at $z = 0.5$	109
4.24	Time evolution of projected streamlines for the facing-wall two-sided cubic lid-driven cavity flow at $Re = 600$	110
5.1	Schematic diagram of uniform colocated grid for interpolation formula. . . .	117
5.2	(a) Restriction of mass fluxes and variables.	121
5.3	Grid stencil for prolongation operator.	123
5.4	Comparison of steady-state velocity profiles (a) horizontal velocity along the vertical centreline and (b) vertical velocity along the horizontal centreline for the single-sided lid-driven cavity.	125
5.5	Steady state streamline patterns for the single-sided lid-driven cavity for various Reynolds numbers.	126

5.6	V-cycle multigrid showing iterations at various levels.	127
5.7	Mass residual history of multigrid methods for single-sided lid-driven cavity for various Reynolds numbers	128
5.8	Schematic diagram of the two-sided lid-driven cavity.	130
5.9	Steady state streamline patterns for the two-sided lid-driven cavity for differ- ent Reynolds numbers.	131
5.10	Steady state pressure contours for the two-sided lid-driven cavity for different Reynolds numbers.	133
5.11	Multiple solutions for a two-sided lid-driven cavity at $Re = 2000$	134
5.12	Multiple solutions of pressure contours for a two-sided lid-driven cavity at $Re = 2000$	135
5.13	Schematic diagram of the four-sided lid-driven cavity.	136
5.14	Steady state streamline patterns for the four-sided lid-driven cavity for differ- ent Reynolds numbers.	138
5.15	Steady state pressure contours for the four-sided lid-driven cavity for different Reynolds numbers.	139
5.16	Multiple solutions for a four-sided lid-driven cavity at $Re = 200$	140
5.17	Multiple solutions of pressure contours for a four-sided lid-driven cavity at $Re = 200$	141
5.18	Mass residual history of multigrid methods for a two-sided lid-driven cavity.	142
5.19	Mass residual history of multigrid methods for a four-sided lid-driven cavity.	143
6.1	Schematic diagram of the flow configuration.	148
6.2	Grid-independence study (a) horizontal velocity and (b) temperature along the vertical centre line at $Ri = 0.01$, $Gr = 10^4$ and $Da = 10^{-3}$	151
6.3	Comparison of (a) horizontal velocity and (b) temperature profile along the vertical centre line for various bouyancy ratios (N) with Khanafer and Vafai [63].	151

6.4	(a) Streamline and (b) isotherm patterns for various Rayleigh numbers for natural convection in a square cavity.	153
6.5	(a) Horizontal velocity and (b) temperature along the vertical centre line at different Darcy numbers (Da) for $Ri = 0.01, Gr = 100$ and $\epsilon = 0.9$	155
6.6	BFDM (a) streamline and (b) isotherm patterns for $Ri = 10^{-2}, Gr = 10^2, \epsilon = 0.9$ and for various Darcy numbers (Da).	156
6.7	BDM (a) streamline and (b) isotherm patterns for $Ri = 10^{-2}, Gr = 10^2, \epsilon = 0.9$ and for various Darcy number (Da).	157
6.8	(a) Horizontal velocity and (b) temperature along the vertical centre line at different Darcy numbers (Da) for $Ri = 0.01, Gr = 10^4$ and $\epsilon = 0.9$	158
6.9	BFDM (a) Streamline and (b) isotherm patterns for $Ri = 10^{-2}, Gr = 10^4, \epsilon = 0.9$ and for various Darcy numbers (Da).	160
6.10	BDM (a) Streamline and (b) isotherm patterns for $Ri = 10^{-2}, Gr = 10^4, \epsilon = 0.9$ and for various Darcy numbers (Da).	161
6.11	Local Nusselt number at the cold (left) wall for various Darcy numbers (Da): (a) $Gr = 10^2$ (b) $Gr = 10^4$	162
6.12	Mass residual history of multigrid methods for $Ri = 0.01, Gr = 10^2$ and $\epsilon = 0.9$ (a) $Da = 0.001$ (b) $Da = 0.01$ (c) $Da = 0.1$ for BFDM.	164
6.13	Mass residual history of multigrid methods for $Ri = 0.01, Gr = 10^4$ and $\epsilon = 0.9$ (a) $Da = 0.001$ (b) $Da = 0.01$ (c) $Da = 0.1$ for BFDM.	165
6.14	(a) Horizontal velocity and (b) temperature along the vertical centre line at different Darcy numbers (Da) for $Ri = 100, Gr = 10^2$ and $\epsilon = 0.9$	167
6.15	(a) Streamline and (b) isotherm patterns for $Ri = 100, Gr = 10^2, \epsilon = 0.9$ and for various Darcy numbers (Da).	168
6.16	(a) Horizontal velocity and (b) temperature along the vertical centre line at different Darcy numbers (Da) for $Ri = 100, Gr = 10^4$ and $\epsilon = 0.9$	169
6.17	(a) Streamline and (b) isotherm patterns for $Ri = 100, Gr = 10^4, \epsilon = 0.9$ and for various Darcy numbers (Da).	170

6.18 Mass residual history of multigrid methods for $Ri = 100$, $Gr = 10^4$ and $\epsilon = 0.9$

(a) $Da = 0.001$ (b) $Da = 0.01$ (c) $Da = 0.1$ for BFDM. 172



List of Tables

2.1	Number of equivalent fine grid iterations required for convergence	26
3.1	Performance of multigrid for a 129×129 grid on a uniform grid at various Reynolds number for the single-sided lid-driven square cavity	55
3.2	Performance of multigrid for a 129×129 grid on a nonuniform grid at various Reynolds number for the single-sided lid-driven square cavity	55
3.3	Performance of multigrid on uniform and nonuniform grids at $Re = 1000$ and $t = 10$ for the single-sided lid-driven square cavity	56
3.4	Comparison of C_{Dmean} and Strouhal number	77
4.1	Performance of multigrid on a $65 \times 65 \times 65$ grid at $Re = 1000$ for a single-sided lid-driven cubic cavity	87
5.1	Performance of multigrid method for a single-sided lid-driven cavity flow at various Reynolds numbers.	129
5.2	Performance of multigrid method for a two-sided lid-driven cavity flow at various Reynolds numbers	143
5.3	Performance of multigrid method for a four-sided lid-driven cavity flow at various Reynolds numbers	143
6.1	Comparison of average Nusselt number for natural convection in a square cavity with available literature	152

6.2	Average Nusselt number $\overline{Nu}$ given by Brinkman-Forchheimer Darcy model and Brinkman-extended Darcy model at $Ri = 0.01$	163
6.3	Performance of multigrid for a 130×130 grid for $Ri = 0.01$ and $Gr = 10^4$ and a mass residual of 10^{-10} for BFDM	163
6.4	Performance of multigrid for a 130×130 grid for $Ri = 0.01$ and $Gr = 10^2$ and a mass residual of 10^{-10} for BFDM	166
6.5	Average Nusselt number $\overline{Nu}$ for various Darcy number and Grashof number at $Ri = 100$	171
6.6	CPU time required by the single-grid and multigrid method at $Ri = 100$, $Gr = 10^4$ for various Darcy numbers	171

Nomenclature

$\mathcal{A}_m$	coefficient matrix vector
$\mathcal{S}_m$	source term vector
A, S	approximation to $\mathcal{A}_m, \mathcal{S}_m$
A^f	nonlinear operator
B	blockage ratio
BDM	Brinkman-extended Darcy model
BFDM	Brinkman-Forchheimer Darcy model
c_p	specific heat at constant pressure, kJ/kgK
C_D	drag coefficient
D	width of the square cyliner
Da	Darcy number, $\frac{\kappa}{H^2}$
e	error
f	frequency of vortex shedding
Gr	Grashof number, $\frac{g\beta\Delta TH^3}{\nu^2}$
H	enclosure height
h	mesh size
i	x -direction grid point
j	y -direction grid point
I	restriction and prolongation operator
l	inflow length
C_L	Lift coefficient
Nu	local Nusselt number
$\overline{Nu}$	average Nusselt number
Pr	Prandtl number, $\frac{\nu}{\alpha_e}$

p	dimensionless pressure
$\vec{q}$	velocity vector
Re	Reynolds number of fluid, $\frac{V.H}{\nu}$
Ri	Richardson number, $\frac{Gr}{Re^2}$
St	Strouhal number, $\frac{fD}{u_{max}}$
t	nondimensional time
T	dimensional temperature, °C
T_h	dimensional temperature of hot fluid, °C
T_c	dimensional temperature of cold fluid, °C
V	vertical wall velocity
u, v	dimensionless velocity components along (x, y) axes
u^f	exact solution
v^f	approximation to the exact solution
x^*, y^*	dimensional Cartesian co-ordinates
x, y	dimensionless Cartesian co-ordinates

Greek Symbols

α_e	effective thermal diffusivity of vtf porous medium, m^2/s
β	fluid thermal expansion coefficient
θ	dimensionless temperature, $\frac{T-T_c}{T_h-T_c}$
ν	effective kinematic viscosity ($\frac{\mu}{\rho}$), m^2/s
μ	effective dynamic viscosity, $Pa\ s^{-1}$
ρ	density, kg/m^3
κ	permeability of the porous medium, m^2
ϕ, φ	general variables
λ, ω	under-relaxation factor
σ	capacity ratio, $\left[\epsilon (\rho c_p)_{fl} + (1 - \epsilon) (\rho c_p)_s \right] / (\rho c_p)_{fl}$

Subscripts

c	coarse-grid
E, N	east and north grid points
e	east face
f	fine-grid
fl	fluid
m	grid level
n	north face
nb	neighbour
s	solid
P	main grid point
w	wall

Superscripts

n	time level
$*$	intermediate value

Chapter 1

Introduction

1.1 Background

The principal aim of Computational Fluid Dynamics (CFD) is the computation of flows in various geometries of practical or theoretical importance. It is imperative that the computations should be accurate and carried out with good time-wise efficiency. As the computers are getting more and more powerful, it may appear as though advanced algorithms and convergence-acceleration devices like implicitization, multigrid etc. are not as important now as it was earlier. However, it must be borne in mind that in recent times CFD is increasingly being used as a design tool in many industrial situations, especially in the aerospace design environment. As CFD is used to optimize experimental efforts, in the preliminary design cycles a CFD code requires to be run many tens of times. In this context convergence-acceleration devices become important, since they have the potential of cutting short the design time by months or even years.

The resolution, and hence numerical accuracy, can be improved by refining the grid supporting the solution. However, even if computer memory resources permit such a refinement, convergence towards the exact steady-state numerical solution tends to slow down dramatically with increasing grid density if conventional iterative algorithms such as Jacobi, Gauss-Seidel or ADI methods are used. This is, of course, a particularly serious constraint

in the context of three-dimensional Navier-Stokes calculations. More advanced solvers, such as preconditioned conjugate gradient method and Stones strongly-implicit procedure [102], offer some advantages, but are not always uniformly efficient and suffer from difficulties of being adapted to vector computers.

In all the above mentioned algorithms, the initial rate of convergence for the first few iterations is rapid, but the convergence slowly deteriorates, as the iterations progress. The main problem is the slow smoothing of long-wave (low-frequency) error components relative to a much faster erosion of short-wave (high-frequency) components, having wavelengths comparable to the mesh distance. As the grid is refined, these low-frequency errors dominate the overall rate of convergence. For a solution scheme to offer fast convergence, all error components must decay uniformly fast. There is an increasing amount of evidence that the multigrid method (MGM) goes some way towards meeting the aforementioned objectives. The method is peculiar in that it can use any one of the traditional relaxation methods to achieve convergence rates far above those possible with the same relaxation variant operating on a single grid. This is rooted in the fact that the role of the relaxation scheme is not primarily to reduce the error but rather to smooth it, so that it can be well supported by the grid. The solution progresses along a sequence of coarsening and refining grids, with only a few relaxation sweeps applied on each level to remove high-frequency error components. Relaxation can be effectively performed by a simple point-explicit method, such as the Gauss-Seidel technique which can be easily vectorized. Similarly, inter-grid transfer (interpolation) processes, termed restriction and prolongation, can be effectively vectorized and parallelized, if careful attention is paid to the handling of boundary information.

There are two broad categories of MGM, namely, correction scheme (CS) and full approximation scheme (FAS). The former operate with corrections to the solution on coarse grids, which are eventually added to the absolute fine-grid solution. This method is applicable to linear systems only. In contrast, the latter method can be used to solve nonlinear problems by operating on the absolute solution (rather than its correction) on all grid levels. The Other extensions of multigrid methods include techniques where no partial differential

equation nor geometrical problem background is used to construct the multilevel hierarchy.[9] Such algebraic multigrid methods (AMG) construct their hierarchy of operators directly from the system matrix, and the levels of the hierarchy are simply subsets of unknowns without any geometric interpretation [107]. Thus, AMG methods become true black-box solvers for sparse matrices. However, AMG is regarded as advantageous mainly where geometric multigrid is too difficult to apply. Before describing the multigrid method, the available methods of analysis are briefly surveyed in the present thesis. This survey is not intended to be comprehensive and is intended to indicate the motivation for the developments that are described in this chapter.

1.2 Literature Review on Multigrid

Fedorenko [33,34] is the originator of the idea of multigrid which initially did not draw much attention. Interest in the technique was reawakened later by Brandt. He also gave it a sound mathematical foundation in a path-breaking paper [12] in 1977. He was the first to demonstrate the practical efficiency and generality of multigrid methods. He first introduced the formation of the coarse grid equations by discretizing the governing equations on each grid and interpolating the fine grid residuals to the coarse grid equations and interpolating coarse grid corrections to the fine grid. Since then the multigrid technique has become one of the most extensively used convergence-acceleration tools in CFD. The multigrid method was rediscovered independently by Hackbusch [44] at about the same period as by Brandt [12]. Mathematical convergence proofs for the MGM in the case of elliptic boundary-value problems were obtained by Hackbusch [45]. The potential of the method has been realized and demonstrated in the solution of various elliptic model problems in [115]. An efficient multigrid algorithm for the Poisson equation has been clearly described in [107].

The review article of Stuben and Trottenberg [103] contains detailed analysis of simple model problems, along with some interesting historical notes. Multigrid methods have been analysed, applied and generalized in many ways since their introduction, and are gradually

becoming recognized as a powerful tool in applied mathematics. Foester and Witsch [35] developed a package of multigrid solvers for solving a simply structured elliptic boundary value problems and showed superior performance of multigrid method to that of fast direct solvers. Fuchs [39], in 1981, developed a numerical solution of 2D Navier-Stokes equation with the emphasis on multigrid. He examined the smoothing properties of different relaxation schemes as well as the effect of stretched grids on multigrid performance. Miller and Schmidt [79] demonstrated the benefits of using multigrid methods in solving the Poisson and Navier-Stokes equations.

Patankar and Spalding's [86] SIMPLE algorithm is one of the most widely used methods for calculating incompressible flows. The attraction of this method is mainly in its simplicity and the easy extension to problems involving heat and mass transfers and turbulence models. In 1988, Sivaloganathan and Shaw [99] first applied multigrid method to SIMPLE using a staggered grid. They adopted the full-approximation storage multigrid method and achieved a large reduction in CPU time. Since then application of multigrid method in SIMPLE pressure correction smoothers has increased. Grid-coarsening process is difficult in staggered grid arrangement because of the use of three different sets of cells. Hence a colocated grid layout, developed by Rhie and Chow [93], to eliminate pressure wiggles is used in the present work. Hortman and Peric [49] presented a finite volume multigrid procedure for the prediction of laminar natural convection flows, enabling efficient and accurate calculations on very fine grids. They used the second-order central-differencing for convection and diffusion fluxes and the strongly implicit procedure was used to relax the discretized governing equations on all grid levels and implemented full-approximation storage method in a full-multigrid fashion. Lien and Leschziner [75] used full approximation storage multigrid method on a colocated finite volume solver for laminar and turbulent flows. A dramatic reduction in the computational time using multigrid method was showed in [49, 75].

Johansson and Davidson [53] developed a three-dimensional SIMPLEC code on colocated grid arrangement using V-cycle. They concluded that multigrid acceleration is highly effective in 2D and 3D laminar and turbulent flows. However, the CPU-time of the multigrid

calculations is linearly dependent on the number of nodes. The effectiveness is not greatly affected by grid non-uniformity. However, it tends to reduce with increasing Reynolds number. Lilek *et al.* [76] reported on the analysis of efficiency and accuracy of a full-multigrid SIMPLE algorithm for 3D flows using colocated grids and central-difference discretization for both convective and diffusive terms. They showed that the central differencing scheme offers good convergence properties and high accuracy even at large Peclet numbers. They demonstrated, for lid-driven cavity, that iterative convergence errors are efficiently reduced when using multigrid method. Yan and Thiele [117] developed a modified full-multigrid algorithm in which only residuals are restricted and not the variables from the fine to coarse grids by summation. This simplifies the multigrid strategy and the structure of the program. They solved fluid flows with heat transfer using SIMPLE algorithm on colocated grids and achieved substantial speed-up for natural convection in an eccentric annulus.

Cornelius *et al.* [24] presented a numerical method for the efficient calculation of three-dimensional, incompressible, turbulent flow in curvilinear co-ordinates. They used full approximation storage method based on SIMPLE algorithm with finite volume discretization. They concluded that multigrid method is an appropriate approach to get solutions for fine grids in acceptable run times. Recently, Cheikh *et al.* [19] presented a benchmark solution for the time-dependent natural convection flows using multigrid method. They used projection method [81, 88] for solving the Navier-Stokes equations and the multigrid method is used to solve the pressure-Poisson equation. They demonstrated the potential of the multigrid method for obtaining very accurate solution on very fine grids and showed that it is a good acceleration technique. Vanka [110, 111] assessed the performance of block-implicit multigrid calculation procedure for computation of steady viscous internal flows. He demonstrated the potential of the multigrid technique to calculate complex fluid flows efficiently. Jameson [51, 52] used multigrid time-stepping algorithm for solving the Euler equations and demonstrated variety of rapidly convergent methods.

Two basic conclusions that can be derived from the above review are the convergence of the MGM is independent of the mesh size and that the amount of computational effort

needed for solving a discrete system of N unknowns is proportional to N . It is promising to obtain solutions on very fine grids in acceptable CPU time. It is important to understand not only the basic principles of the multigrid solution procedures but also to develop, as general as possible, user-oriented codes for the solution of specific equations. In the present work we discuss some important points for the development of Navier-Stokes solvers using multigrid. Two different types of computer codes are developed here. The first code is based on the finite-difference method using staggered grid arrangement on graded Cartesian meshes. Here the unsteady incompressible 2D and 3D Navier-Stokes equations are solved by using the fractional-step method and the pressure-Poisson equation is solved by using the linear multigrid method at every time-step. The other code is developed for the finite-volume-based SIMPLE algorithm to solve 2D steady incompressible Navier-Stokes equations using the full-approximation storage multigrid method for problems involving fluid-flow and heat transfer that also include porous media flows. This is perhaps the first instance in which full-approximation storage multigrid method is used in porous media flows.

1.3 Motivation and Objectives

The motivation behind the present work is to gain insight into the problem faced, intricacies and the techniques needed to simulate various flow situations using a given numerical algorithm. The intention is to gain computational experience and acquire the expertise needed to tackle complex problems requiring both intensive and exhaustive computational work. The main aim of the present work is to develop a multigrid-accelerated code which has promising potential to provide time accurate solutions for two and three-dimensional incompressible viscous fluid flow problems. Also intended is to examine the performance of full-approximation storage multigrid method to porous cavity flows to which they have not been applied so far. Another objective is to examine the applicability of multigrid code to a hitherto unexplored problems and situations involving fluid flow and heat transfer. All the numerical investigations have been carried out with the help of computer codes developed

by the author.

1.4 The Present Thesis

In the present thesis, a multigrid code is developed for linear equations and the effectiveness of the method is demonstrated by solving 2D heat conduction problem. The convergence history of the test problem shows that the multigrid algorithm requires far less computational effort than the Gauss-Seidel and successive over-relaxation method. This linear multigrid code can be assimilated easily into the fractional-step procedure to solve pressure-Poisson equation which has to be solved at every time-step in the computation of incompressible transient viscous flows.

The important task of the present work is concerned with the development of a multigrid-assisted solver for the 2D and 3D time-dependent incompressible Navier-Stokes equations on graded Cartesian meshes. As the finite difference method is used to discretize the governing equations on nonuniform staggered grids, a transformation of the governing equations from the physical space to the computational space is performed. To obtain the second-order time accuracy a fractional-step method is employed. The convective terms are discretized using a third-order accurate upwind scheme and the viscous terms are centrally differenced to fourth-order accuracy. The main aim of the present code is to efficiently capture both transient and steady-state solutions of four different time-dependent fluid flow problems. The code is first applied to compute unsteady flow in a single-sided lid-driven cavity and then to obtain steady flow in the same configuration by time-marching. The results obtained are in excellent agreement with the established numerical results. Because of clustering accurate results are produced with good grid-economy, which further adds to the computational efficiency. The performance of the multigrid method is seen to be impressive. The total computational time is effectively reduced to one-tenth compared to that for the single-grid method.

The code is further tested to capture periodic flow in the lid-driven cavity for $Re = 8200$ and 10000. After having thus gained an insight into various aspects of the code including

its accuracy and capability, it is then used to capture the transient flow features that evolve with time in a 2D two-sided lid-driven cavity. It involves gradual development of a free ‘shear layer’ with associated vortices. Another issue addressed in the present work is the comparative performance of the multigrid algorithm on uniform and nonuniform grids of equal sizes. Computations are also carried out for the much-examined backward-facing step flow. This problem is solved in the Reynolds number range of 100 to 1000. The present results are compared with the results reported in the literature. The capability of the 2D transient code is better realized when it is used to compute flow past a square prism and the famous von Kármán street is simulated very successfully for $Re = 100$.

Another significant contribution of the present thesis is the development of a 3D transient code for the NS equations on nonuniform grids with transformation. The pressure-Poisson equation requires solution at every time-step and this is usually the most intensive computation task. Therefore multigrid method is employed to accelerate the convergence of the Poisson equation. To test the efficiency of the code, it is used to solve the 3D lid-driven cavity problem. For the same problem, the method accurately computes both transient and steady state results. The results compare very well with established numerical results. The code developed is then used to compute transient and steady-state flows in a hitherto unexplored configuration, namely, the two-sided cubical cavity flow induced by the motion of two facing walls in the same direction. Various observations are reported about this untested flow configuration.

The next major task in the present thesis is to develop a FAS multigrid code for solving incompressible Navier-Stokes equations applicable to various flows including that in a porous media. We have used full-approximation storage multigrid method to compute these flows. The convective terms in the transport equations are discretized using the Quadratic Upwind Interpolation for Convective Kinematics (QUICK) [48] scheme on an orthogonal collocated grid. The coupling of the velocity and the pressure terms of momentum equations is solved by SIMPLE algorithm [86]. To test the efficiency of the code, it is first used to solve the 2D lid-driven cavity problem. The FAS multigrid method achieved appreciable acceleration of

convergence. Subsequently it is employed to study the multiple steady solutions in the two- and four-sided lid-driven cavity flows. In the last part of the present work the developed code is applied to heat transfer problems. First it is applied to the standard problem of steady-state natural convection in a square-cavity. Subsequently the code is employed to investigate the performance of Brinkman-Forchheimer Darcy model (BFDM) and Brinkman-extended Darcy model (BDM) in a mixed convection square cavity filled with a porous medium. The influence of the key parameters, namely Darcy number (Da) and Grashof number (Gr) on the flow and heat transfer is examined. The issue of reliability of the results is addressed. Wherever possible, the predictions are validated by careful comparison with established results.

1.5 Organization of the Thesis

The thesis is organized in seven chapters. Chapter 1 presents an introduction to multigrid method and the relevant literature survey. Chapter 2 discusses the multigrid theory and the performance of multigrid method for linear equations. The development of multigrid code for 2D transient incompressible flows on nonuniform grids using transformation is described in Chapter 3 and its application to lid-driven cavity, two-sided lid-driven cavity, backward step-facing flow and flow over the square prism is also discussed in this chapter. Chapter 4 discusses the transient solutions of 3D single-sided and two-sided lid-driven cavities and a comparison of 2D and 3D transient solutions is also presented in this chapter. Chapter 5 describes the application of full-approximation storage multigrid method to the single-sided lid-driven cavity and to the two-sided and four-sided lid-driven cavities flow problems. Multiple solutions are also presented in this chapter. The performance of non-Darcy models, namely, Brinkman extended Darcy model and Brinkman-Forchheimer Darcy model, for mixed convection in a porous cavity are analysed using the multigrid method in Chapter 6. Chapter 7 highlights the various achievements of the work and throws light on possible future works. Three appendices are added to improve the clarity of the thesis.

Chapter 2

Multigrid Theory

2.1 Introduction

This chapter is intended both to serve as a simple introduction to multigrid method and to emphasise its potential for solving problems in computational fluid dynamics. The partial differential equations are the heart of most mathematical models used in engineering physics, giving rise to extensive computations. In order to capture the true physics of the problems large number of grid points are required. The advancement in computational power, in both speed and storage, of modern supercomputers has brought solutions of problems hitherto considered too large to compute within the scope of these computing machines. Over the past two decades, several new techniques for solving fluid flow problems have been developed. These advancements have also brought other challenges to light. The conventional solution procedures, which seek to solve the problem on a single distribution of grid points (single-level methods), have the tendency to prohibitively slow down the convergence for problems involving a large number of grid points. This problem is compounded in the case of three-dimensional situations, where a nominal grid refinement in any coordinate direction leads to tremendous increase in the total number of grid points. The main problem in the iterative methods like Jacobi, Gauss-Seidel etc. is the slow smoothing of long-wave error components relative to the rapid damping of short-wave components. Therefore, there is a need of a

solution scheme, “multigrid method”, which decays all error components uniformly fast. The multigrid methods, which are based on many levels of grids, have overcome this slow convergence characteristic of the single-grid methods for large-scale problems. Multigrid methods have been established as a powerful tool for accelerating the numerical convergence and, thus reducing the computational time. The multigrid method has its origin in early work of Fedorenko [33,34] and has been extensively studied and developed by Brandt [12]. Since then the popularity of the multigrid method has grown enormously, particularly within the context of CFD. Further details on the multigrid method can be found in [12,14,106]. The present chapter outlines the main principles and the practical utility of multigrid methods. Also the relative performance of multigrid method with Gauss-Seidel and successive over-relaxation method in the numerical solution of the Laplace equation is examined in some details.

Multigrid method is one of the most effective way to accelerate convergence and is a very fast linear iterative solver based on the multilevel paradigm. This method iteratively solves a system of discrete (finite-difference or finite-element or finite-volume) equations on a given grid, by constant interactions with a hierarchy of coarser grids, taking advantage of the relation between different discretizations of the same continuous problem. The typical application of multigrid is in the numerical solution of elliptic partial differential equations in two or more dimensions. Multigrid algorithms are not difficult to program, if the various grids are suitably organized. Multigrid can be applied in combination with any of the common discretization techniques like Jacobi or Gauss-Seidel (GS) methods which provide excellent smoothing of the local error.

Fig. 2.1 demonstrates that the smooth components of the error in the fine grid projected on coarser grid becomes more oscillatory. Assume that a relaxation scheme on the fine grid with $N = 12$ shown in Fig. 2.1(a) removes the oscillatory components of the error leaving relatively smooth error. These smooth components are then projected directly to the coarser grid with $N = 6$ and the smooth wave on the fine grid looks more oscillatory on the coarser grid (Fig. 2.1(b)). This suggests that when relaxation begins to stall, signaling

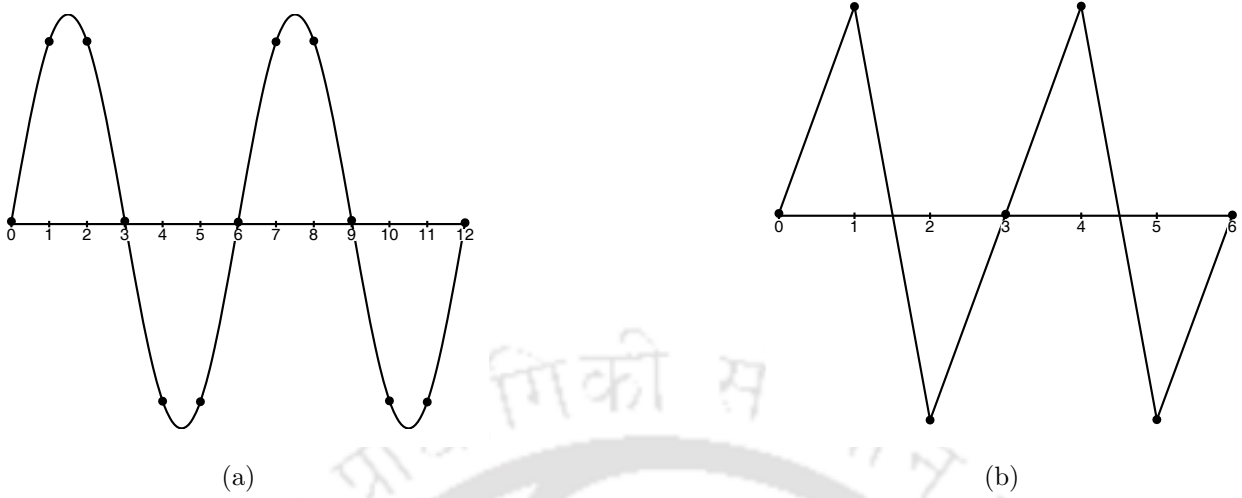


Fig. 2.1: (a) Fine grid (b) Coarse grid.

the predominance of smooth error modes, it is advisable to move to a coarser grid, on which those smooth error modes appear more oscillatory and relaxation will be more effective.

Multigrid method is broadly classified into two categories, namely, correction scheme (CS) and full approximation storage (FAS) method. The CS is applicable to linear equations only; they operate with corrections to the solution on coarse grids, which are eventually added to the absolute fine-grid solution. The FAS is used for solving non-linear problems. It solves equations for approximations to the solutions rather than for corrections at each grid. The three important steps that should be carried out in order to apply multigrid using a coarse grid are:

- i) Transfer of the solution and residuals from the fine to the coarse grid.
- ii) Calculation of new solution (corection/approximation) on the coarse grid.
- iii) Solution interpolation from the coarse to the fine grid.

The above method is called the two-grid multigrid method. At this point, it is important to introduce two terms which are frequently used hereafter - restriction and prolongation. Restriction is defined as the transfer of variables from the fine to the coarse grid denoted by the operator I_f^c and prolongation is defined as the transfer of variables from the coarse to the fine grid denoted by the operator I_c^f with the suffixes f and c standing for fine and coarse grids. In the next section the two-grid multigrid algorithm for solving linear equation

is discussed.

2.2 Linear Two-Grid Algorithm

A two grid algorithm for linear problems consists of smoothing on the fine grid, approximation of the required correction on the coarse grid, prolongation of the the coarse grid correction to the fine grid and again smoothing on the fine grid (Fig. 2.2). Consider a

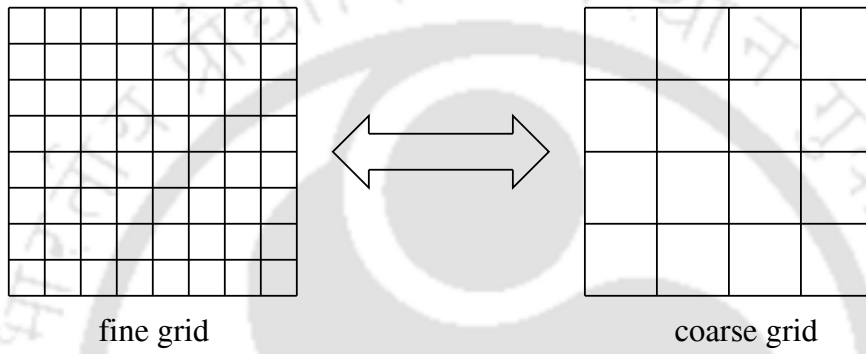


Fig. 2.2: Grid arrangement in two-level scheme.

two-dimensional Poisson equation

$$\nabla^2 \phi = S \quad (2.1)$$

where S denotes the source term. Eq. (2.1) after discretization by the finite difference method can be written as

$$L\phi_{i,j} = S_{i,j} \quad (2.2)$$

where L denotes the discretization operator. A two grid correction scheme proceeds as:

1. Iterate Eq. (2.2) n times on a fine grid.
2. Compute residual $R_{i,j}^f = L\phi_{i,j} - S_{i,j}$ and store at each point. This residual is restricted to the coarser grid to convert the low-frequency error to high-frequency error that can be smoothed on the coarse grid. The restricted residual is denoted by $I_f^c R_{i,j}^f$.

3. The correction equation is given by

$$L(\Delta\phi)_{i,j}^c = -I_f^c R_{i,j}^f \quad (2.3)$$

where $\Delta\phi$ is the correction. Eq. (2.3) is iterated a few times on the coarse grid using zero as the initial guess while keeping the residual fixed at each grid point. The solution $\Delta\phi_{i,j}$ represents the correction to the fine-grid solution.

4. The corrections obtained on the coarse grid are prolonged onto the fine grid and the process is denoted by $I_c^f(\Delta\phi)_{i,j}^c$.

5. Correct the fine grid approximations using,

$$\phi_{i,j}^{new} = \phi_{i,j}^{old} + I_c^f(\Delta\phi)_{i,j}^c \quad (2.4)$$

It is necessary to repeat the above two-level cycle until desired convergence is achieved. It is important to know how the coarse-grid correction equation Eq. (2.3) is obtained. But before that, it is also essential to know how to restrict the computed residuals from the fine to the coarse grids and the prolongation of the corrections from coarse to the fine grids. Therefore, we need an inter-grid transfer mechanisms for transferring information between the grids and this will be discussed in the next subsection.

2.2.1 Restriction and Prolongation

In a finite difference method all nodes on the fine grid coincide with those of the coarse grid and therefore the transfer of variables from the fine to the coarse grid has to be performed by injection, i.e. the coarse grid variables simply take their values directly from the corresponding fine grid points. The arrangement of variables is shown in Fig. 2.3. Thus the restriction operator is denoted as

$$R_{i,j}^c = [I_f^c] R_{2i,2j}^f, \quad 1 \leq i, j \leq \frac{N}{2} - 1 \quad (2.5)$$

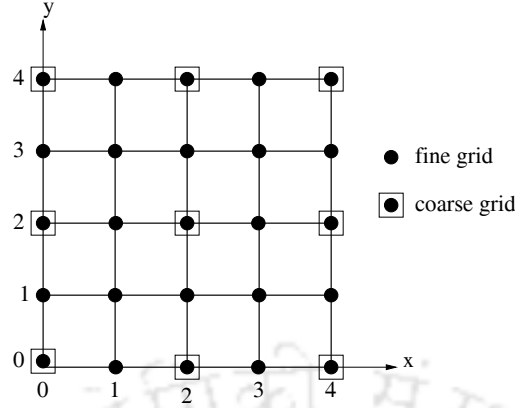


Fig. 2.3: Location of fine and coarse grid points for a 2D square domain.

The transfer of variables from the coarse to the fine grid by the linear interpolation is called prolongation. The interpolation operator is denoted by I_c^f through which the corrections obtained from the coarse-grid solutions are passed onto the fine grid to update the solution on it. The prolongation operator can be expressed as $(\Delta\phi)^f = I_c^f [\Delta\phi]^c$, where

$$(\Delta\phi)_{2i,2j}^f = (\Delta\phi)_{i,j}^c, \quad 0 \leq i, j \leq \frac{N}{2} - 1 \quad (2.6)$$

$$(\Delta\phi)_{2i+1,2j}^f = \frac{1}{2} [(\Delta\phi)_{i,j}^c + (\Delta\phi)_{i+1,j}^c] \quad (2.7)$$

$$(\Delta\phi)_{2i,2j+1}^f = \frac{1}{2} [(\Delta\phi)_{i,j}^c + (\Delta\phi)_{i,j+1}^c] \quad (2.8)$$

$$(\Delta\phi)_{2i+1,2j+1}^f = \frac{1}{4} [(\Delta\phi)_{i,j}^c + (\Delta\phi)_{i,j+1}^c + (\Delta\phi)_{i+1,j}^c + (\Delta\phi)_{i+1,j+1}^c] \quad (2.9)$$

At even-numbered fine grid points both row and column wise (Eq. (2.6)) the values of $(\Delta\phi)_{2i,2j}^f$ are transferred directly from coarse to fine grid (Fig. 2.3) because these points coincide with one another. At odd-numbered columns and even-numbered rows of fine grid points like $(1,0)$, $(1,2)$, etc., the values of $(\Delta\phi)_{2i+1,2j}^f$ are calculated by taking the average of adjacent coarse-grid values Eq. (2.7). Similarly at odd-numbered rows and even-numbered columns of fine grid points such as $(0,1)$, $(2,1)$, etc., the values of $(\Delta\phi)_{i,2j+1}^f$ are calculated using Eq. (2.8). In the remaining odd-numbered (both row and column) fine grid points the values of $(\Delta\phi)_{2i+1,2j+1}^f$ are calculated using Eq. (2.9). It is very important to transfer the corrections accurately back to the fine grid because if interpolation works well the correction

of the fine grid solution would be effective. As promised earlier the formulation of the coarse-grid correction equation (Eq. 2.3) will be given now. Recall that Eq. (2.2) was obtained after the finite difference discretization of Eq. (2.1)

$$L\phi_{i,j} = S_{i,j} \quad (2.2)$$

Let $\phi_{i,j}^k$ represent an intermediate solution (after k iterations), $\Delta\phi_{i,j}$ the correction and $\phi_{i,j}$ the exact numerical solution. Then we can write

$$\phi_{i,j} = \phi_{i,j}^k + \Delta\phi_{i,j} \quad (2.10)$$

Thus the correction is a value that must be added to an intermediate solution in order to obtain the final converged solution. From Eq. (2.10), we can write Eq. (2.2) as

$$L\phi_{i,j}^k - S_{i,j} + L(\Delta\phi)_{i,j} = 0 \quad (2.11)$$

The residual $R_{i,j}$ for an intermediate solution is defined as the unbalance between the two sides of the difference equation (Eq. 2.2) evaluated at every grid point. Accordingly,

$$R_{i,j} = L\phi_{i,j} - S_{i,j} \quad (2.12)$$

at convergence $R_{i,j} = 0$. For an intermediate solution $\phi_{i,j}^k$ Eq. (2.12) can be written as

$$R_{i,j} = L\phi_{i,j}^k - S_{i,j} \quad (2.13)$$

Therefore Eq. (2.11) becomes,

$$L(\Delta\phi)_{i,j} + R_{i,j} = 0 \quad (2.14)$$

Eq. (2.14) is known as the residual or correction equation which is ultimately solved on coarser grid. The first term in Eq. (2.14) can be denoted as $L(\Delta\phi)_{i,j}^c$ for comprehension and the second term $R_{i,j}$ is the restricted residual from fine to coarse grid and may be denoted as $I_f^c R_{i,j}^f$. Hence Eq. (2.14) can be written as

$$L(\Delta\phi)_{i,j}^c + I_f^c R_{i,j}^f = 0 \quad (2.15)$$

This equation is solved iteratively, using $(\Delta\phi)_{i,j}^c = 0$ as the initial guess while keeping $I_f^c R_{i,j}^f$ unchanged, to get the corrections for the fine grid solutions.

The basic principle of multigrid will now be explained by studying a one-dimensional (1D) model problem. Of course, 1D problems do not require application of multigrid method, since for the algebraic systems that result from discretization direct solution is efficient. However, in a one dimensional situation multigrid method can be analysed by elementary methods, and their essential principle is easily demonstrated. A numerical experiment has been performed to illustrate that relaxation can be powerful when done on two-grid multigrid method and it has been explained heuristically. Consider a 1D problem governed by the second-order partial differential equation

$$\frac{\partial^2 \phi}{\partial x^2} = 0 \quad (2.16)$$

on a grid with $n = 64$ points. The initial guess is given by

$$\phi_i = \frac{1}{2} \left[\sin \left(\frac{k_1 i \pi}{n} \right) + \sin \left(\frac{k_2 i \pi}{n} \right) \right] \quad (2.17)$$

consisting of $k_1 = 16$ and $k_2 = 40$ modes. The two-grid correction scheme described in Section 2.2 is applied and the smoother used is the weighted Jacobi method (with $\omega = \frac{2}{3}$) is

given by

$$\phi_i^* = (\phi_{i+1}^0 + \phi_{i-1}^0) / 2.0 \quad (2.18)$$

$$\phi_i^1 = (1 - \omega) \phi_i^0 + \omega \phi_i^* \quad (2.19)$$

where ‘1’ denotes the newly updated approximation, ‘0’ the current approximation and ‘*’ the intermediate value. Fig. 2.4 shows the damping of errors for a two-grid correction scheme. Fig. 2.4(a) shows the initial guess with its two modes $k_1 = 16$ and $k_2 = 40$ with Euclidean norm as 3.994. The approximation ϕ_i obtained after one relaxation sweep using weighted Jacobi method is superimposed on the initial guess in Fig. 2.4(b), which shows that much of the oscillatory component of the initial guess has been removed and the norm of the initial error is reduced to 57%, i.e., 2.2879. After three relaxation sweeps on the fine grid, the approximation again superimposed on the initial guess is shown in Fig. 2.4(c). The solution has now become smoother and the norm reduced to 1.475 which is 36% of the initial norm. Further relaxations on the fine grid provide only a marginal improvement at this point. Therefore, it is time to move to the coarse grid. The residuals calculated at all the points on the fine grid are transferred onto the coarse grid containing 32 points. After 1 relaxation sweep on the coarse grid the approximation is overlaid on the initial guess depicted in Fig. 2.4(d). It is clearly seen from Fig. 2.4(d) that we have achieved another reduction in the error by moving to the coarse grid, the norm calculated is 0.8111, which is 20% of the initial error norm. This improvement occurs because the smooth error components, inherited from the fine grid, appear oscillatory on the coarse grid and are, therefore, quickly removed. The error after three coarse grid relaxation sweeps is shown in Fig. 2.4(e). The norm of the error is reduced to 0.289 which is 8% of the initial value. The obtained coarse grid approximation to the correction is now used to correct the fine grid approximation. After three additional fine grid relaxations, the norm of the error is further reduced to 0.1218 which is 3% of the initial error norm as shown in Fig. 2.4(f). If the error is still to be reduced the cycle is repeated. This example demonstrates the powerful damping of errors using multigrid

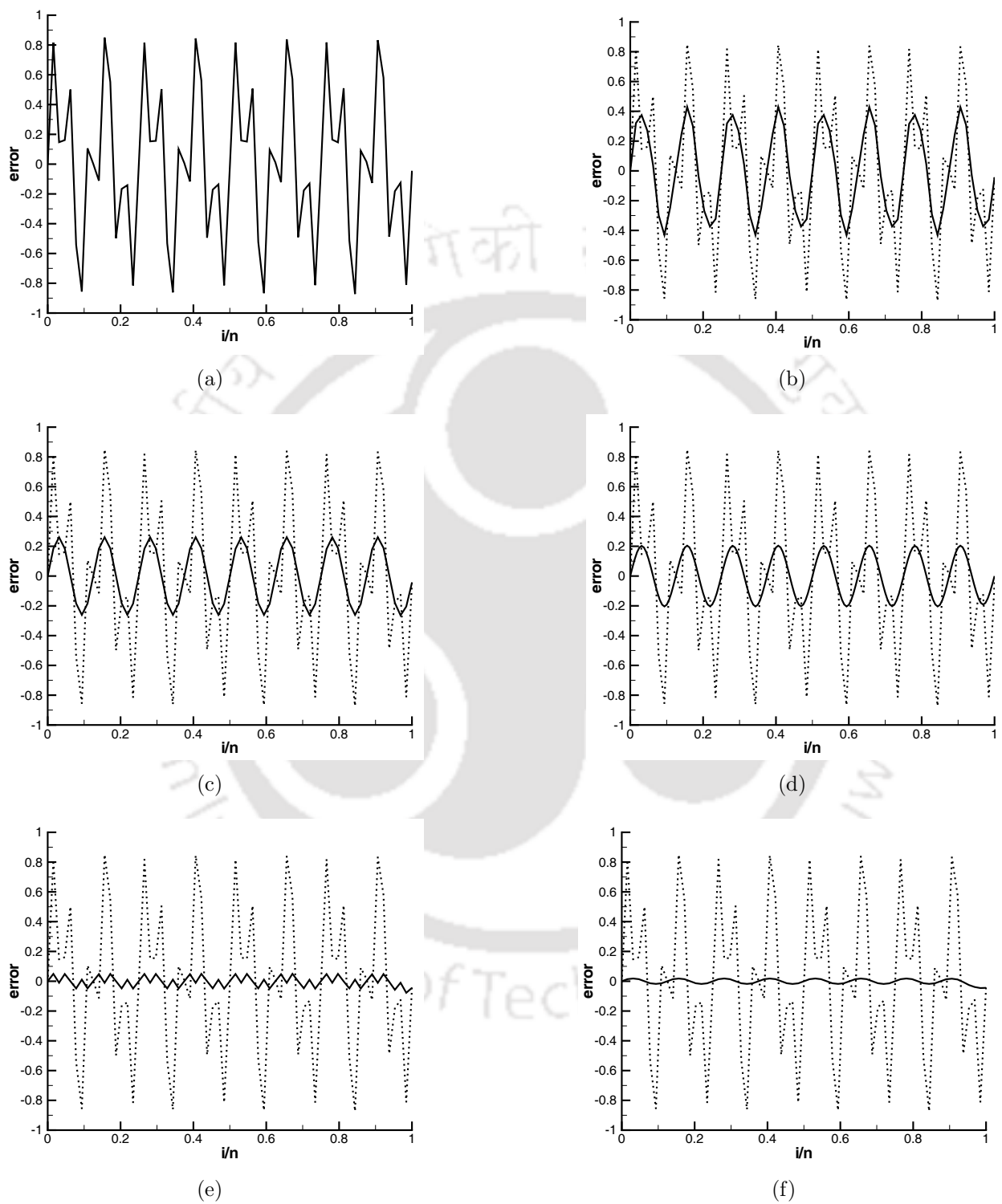


Fig. 2.4: Damping of errors using a 2-level (64 and 32 points) multigrid method.

method.

2.3 Multi-level Scheme

The two-grid algorithm described in Section 2.2 consists of only one coarse grid. In order to get significant improvements in computational economy use of additional levels are required. The extension to additional level is straight forward in principle but requires careful interpretation of the multigrid concept in order to correctly implement the procedure. If multiple coarse grids are present, the steps 2 and 3 of Section 2.2 are repeated until the coarsest grid is reached. It is important to realize, that the residual on the coarse grid formed from Eq. (2.15) is given by

$$R_{i,j}^c = L [\Delta\phi]_{i,j}^c + I_f^c R_{i,j}^f \quad (2.20)$$

In this way, the residual of the finest grid controls the solution on all coarser grids. Once the coarsest grid relaxation is over step 4 in Section 2.2 is successively repeated until the finest grid is reached again. This procedure is known as the V-cycle in which the calculation proceeds from the finest grid down to the coarsest and back to the finest. One sweep of the multigrid method, from the finest to the coarsest grid and back to the finest grid again is called a cycle. There are various multigrid cycles, namely, V-cycle (Fig. 2.5(a)), W-cycle (Fig. 2.5(b)), S-cycle (also called the saw-tooth cycle), Brandt's method (also called R-cycle) and FMV (full multigrid V-cycle) (Fig. 2.5(c)). Only V-, W- and FMV-cycles are shown in Fig. 2.5 with grid spacing $h, 2h, 4h, \dots = 2^{l-1}h$ where l represents number of levels. Among the cycles V- and W-cycles are widely used. W-cycles are usually more robust and they are easier to analyze in the classical multigrid convergence theory. However, they are expensive, and in local refinement applications (or in 1D) they may fail to have optimal work count [95]. W-cycles are especially expensive in parallel algorithms when frequent coarse grid visits lead to poor processor utilization. A full multigrid method starts solving the problem on coarsest

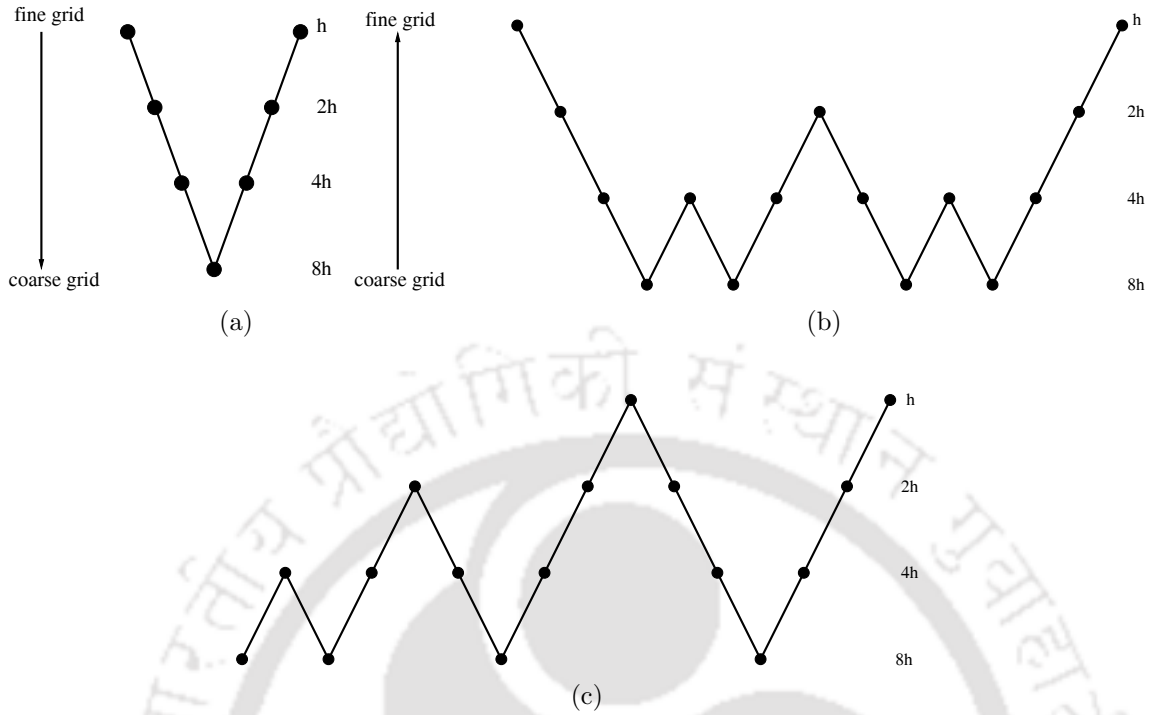


Fig. 2.5: (a) V-Cycle (b) W-Cycle (c) FMV-Cycle.

grid and using that solution as the initial approximation for the next finer level. This process continues until it reaches the finest level where the solution of the problem is required. It should be noted that the order of interpolation has to be chosen in an appropriate way. Either V- or W-cycles can be used in full multigrid method. The optimal choice of number of relaxation sweeps required at any grid level is not explicitly available. Various experiments have to be performed to find it. In the present thesis V-cycle has been used in all the cases. The next section discusses the performance of a 4-level V-cycle scheme applied to solve a two-dimensional heat conduction problem governed by the Laplace equation.

2.4 Multigrid Implementation to Heat Conduction Problem

Laplace equation is the governing equation for the two-dimensional heat conduction problem

$$\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} = 0 \quad (2.21)$$

As a first step towards learning the multigrid method, we have implemented this method to solve the Laplace equation with Dirichlet boundary equations and the computational effort for the multigrid method is compared with Gauss-Seidel and successive over-relaxation (SOR) methods. A square domain is considered (Fig. 2.6) with the boundary conditions

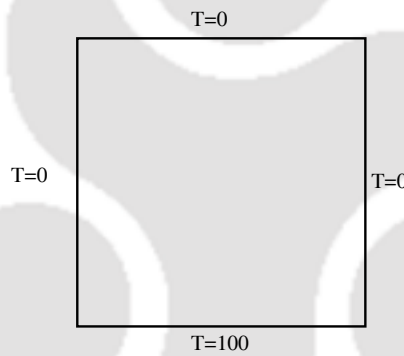


Fig. 2.6: Schematic representation of two-dimensional square rod.

$$T = 100 \quad \text{at} \quad y = 0$$

$$T = 0 \quad \text{at} \quad y = 1$$

$$T = 0 \quad \text{at} \quad x = 0$$

$$T = 0 \quad \text{at} \quad x = 1$$

The Laplace equation (2.21) is numerically solved using the Gauss-Seidel method, Gauss Seidel with SOR and multigrid method with Gauss-Seidel as the smoother. In multigrid method it is convenient to successively halve the number of uniformly distributed grid points on each side of the grid to obtain the coarser grid levels. Hence grids of 3×3 , 5×5 , 9×9 , 17×17 , 33×33 , 65×65 , 129×129 , 257×257 can be used.

The convergence criterion used was $Error < Error_{max}$, where

$$Error = \frac{\Sigma |T_{i,j}^{k+1} - T_{i,j}^k|}{Initial \ L_1\text{-norm of error}}$$

where k represents the iteration level and $Error_{max} = 0.0001$. The standard central-difference discretization for Eq. (2.21) yields,

$$\frac{T_{i-1,j} - 2T_{i,j} + T_{i+1,j}}{(\Delta x^2)} + \frac{T_{i,j-1} - 2T_{i,j} + T_{i,j+1}}{(\Delta y^2)} = 0 \quad (2.22)$$

Let an operator L be defined such that $LT_{i,j}$ is the standard difference representation; then Eq. (2.22) can be written as

$$LT_{i,j} = \frac{T_{i-1,j} - 2T_{i,j} + T_{i+1,j}}{(\Delta x^2)} + \frac{T_{i,j-1} - 2T_{i,j} + T_{i,j+1}}{(\Delta y^2)} = 0 \quad (2.23)$$

Fig. 2.7 shows the grid arrangement in a 4-level multigrid method. Here, level-1 indicates the finest grid, level-2 and level-3 indicates the coarser grid and level-4 indicates the coarsest grid. The steps to be followed for a 4-level multigrid method applied to the 2D heat conduction problem are as follows:

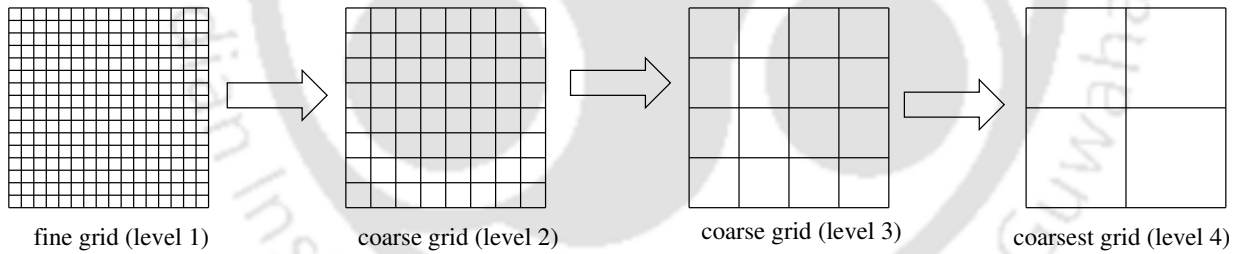


Fig. 2.7: Grid arrangements in the 4-level multigrid method.

1. The difference equation, $LT_{i,j} = 0$ is iterated k times ($k = 3$ or 4) on the finest grid.
2. The residual, $R_{i,j}^1 = LT_{i,j}^k$ is computed and stored at each point. This residual is then restricted by injection to the next coarse grid. The restricted residual is denoted as $I_1^2 R_{i,j}^1$, where I denotes the transfer operator, its subscript indicates the level of origin and its superscript indicates the level of destination and the superscript on the R indicates the grid level upon which the residual is computed.

3. The correction equation

$$L(\Delta T)_{i,j}^2 = -I_1^2 R_{i,j}^1 \quad (2.24)$$

is iterated n times on grid level 2 using zero as the initial guess while keeping the residual fixed at each grid point. The solution after n iterations, $(\Delta T)_{i,j}^2$, represents a correction to the fine-grid solution. This solution, as well as the residual used to obtain it, are stored for future use in the prolongation phase. In order to transfer the problem to a next coarser grid, an updated residual needs to be computed on grid level 2. The updated residual at level 2 is

$$R_{i,j}^2 = I_1^2 R_{i,j}^1 + L(\Delta T)_{i,j}^2 \quad (2.25)$$

where $(\Delta T)_{i,j}^2$ is the solution obtained on the grid level 2 after n iterations. The newly updated residual is then restricted to the next coarser grid (level 3) as $I_2^3 R_{i,j}^2$.

4. The correction equation

$$L(\Delta T)_{i,j}^3 = -I_2^3 R_{i,j}^2 \quad (2.26)$$

is iterated n times on grid level 3 using zero as the initial guess. The solution after n iterations can be thought of as a correction to the correction obtained on grid level 2, which of course represents a further correction to the fine-grid solution. This solution and the residual used to obtain it are stored for use in the prolongation phase. A new residual needs to be computed on grid level 3 in order to transfer to the coarsest grid (level 4). The new residual at level 3 is

$$R_{i,j}^3 = I_2^3 R_{i,j}^2 + L(\Delta T)_{i,j}^3 \quad (2.27)$$

where $(\Delta T)_{i,j}^3$ is the solution obtained on grid level 3 after n iterations. The newly

updated residual is again restricted to the coarsest grid (level 4). The correction equation $L(\Delta T)_{i,j}^4 = I_3^4 R_{i,j}^4$ is iterated to convergence on the coarsest grid accurately using zero as the initial guess. As there is no other coarser grid, there is no need to find the residual at this grid level.

5. Now the corrections obtained on the coarsest grid are transferred (prolongated) on to the next finer grid and are represented as $I_4^3 \Delta T_{i,j}^4$. These are added to the corrections $\Delta T_{i,j}^3$ obtained earlier at level 3. The sum of the two corrections is used as the initial guess at each point, relaxed few times using Eq. (2.26). The new solution represents improved corrections. The corrections from level 3 are prolonged onto the next finer grid (level 2). These corrections are added to the already obtained corrections $\Delta T_{i,j}^2$ at level 2. Using these corrections as the initial guess, n iterations are made on grid level 2 using Eq. (2.24). It is to be noted that, no new residuals are computed in the prolongation phase of moving up from the coarser to finer grid.
6. The improved corrections from level 2 are prolonged onto grid level 1 (finest grid) and added to the previous solution obtained on the finest grid $T_{i,j}$. The corrected solution is then relaxed for n sweeps. This completes one cycle. Now check for convergence. If convergence has not occurred, this cycle has to be repeated till desired convergence is reached.

2.5 Results and Discussions

The temperature profile obtained is plotted in Fig. 2.8(a). Table 2.1 compares the performance of multigrid method with the standard Gauss-Seidel (GS) and Gauss-Seidel with optimum over-relaxation (SOR) methods. The computational effort is reported in terms of the equivalent fine-grid sweeps referred as work units (WU) i.e., the fine grid equivalent of the coarse grid iterations are calculated in the multigrid loop so that it can be directly compared with the results obtained by the Gauss-Seidel method. The total work units required

by GS scheme almost proportionally increases with the number of grid points used, whereas the use of SOR scheme substantially reduces the computational effort especially with fine grids. For a 129×129 grid GS method requires 9176 iterations and SOR method requires only 163 iterations. The two-level multigrid is seen to provide a significant reduction in the computational effort but it does not perform quite as well as SOR. But the performance of multigrid with 4-levels is amazing. The number of work units required is seen to be independent of the number of grid points used. It has been seen from Table 2.1 that for a 129×129 grid only 20 equivalent fine grid sweeps were required compared to 9176 standard Gauss-Seidel iterations and 163 optimum SOR iterations. For a two level multigrid (MG-2)

Table 2.1: Number of equivalent fine grid iterations required for convergence

grid size	GS	GS-SOR	MG-2	MG-Max
9×9	67	19	18	16
17×17	197	31	27	18
33×33	716	52	67	18
65×65	2579	91	203	18
129×129	9176	163	654	20

on a 129×129 grids performance is seen to improve by increasing the number of sweeps in the coarser grid. Fig. 2.8(b) shows the performance of the 2-level multigrid grid by varying the number of iterations in the coarser grid. When the number of sweeps are 2 and 5 on the coarser grid, the corresponding equivalent fine grid iterations required are 1527 and 986, respectively. When it is increased to 50, the equivalent fine grid sweeps required for the convergence ultimately reduce to 652. It is observed from Fig. 2.8(c), that using 4-level multigrid is good enough as further increase of levels produce no time-wise gain. Specifically, it was observed that 9176, 652, 71, 22, 20, 20 and 20 work units were required when using 1-, 2-, 3-, 4-, 5- and 6- levels respectively. The convergence history observed with the GS, GS

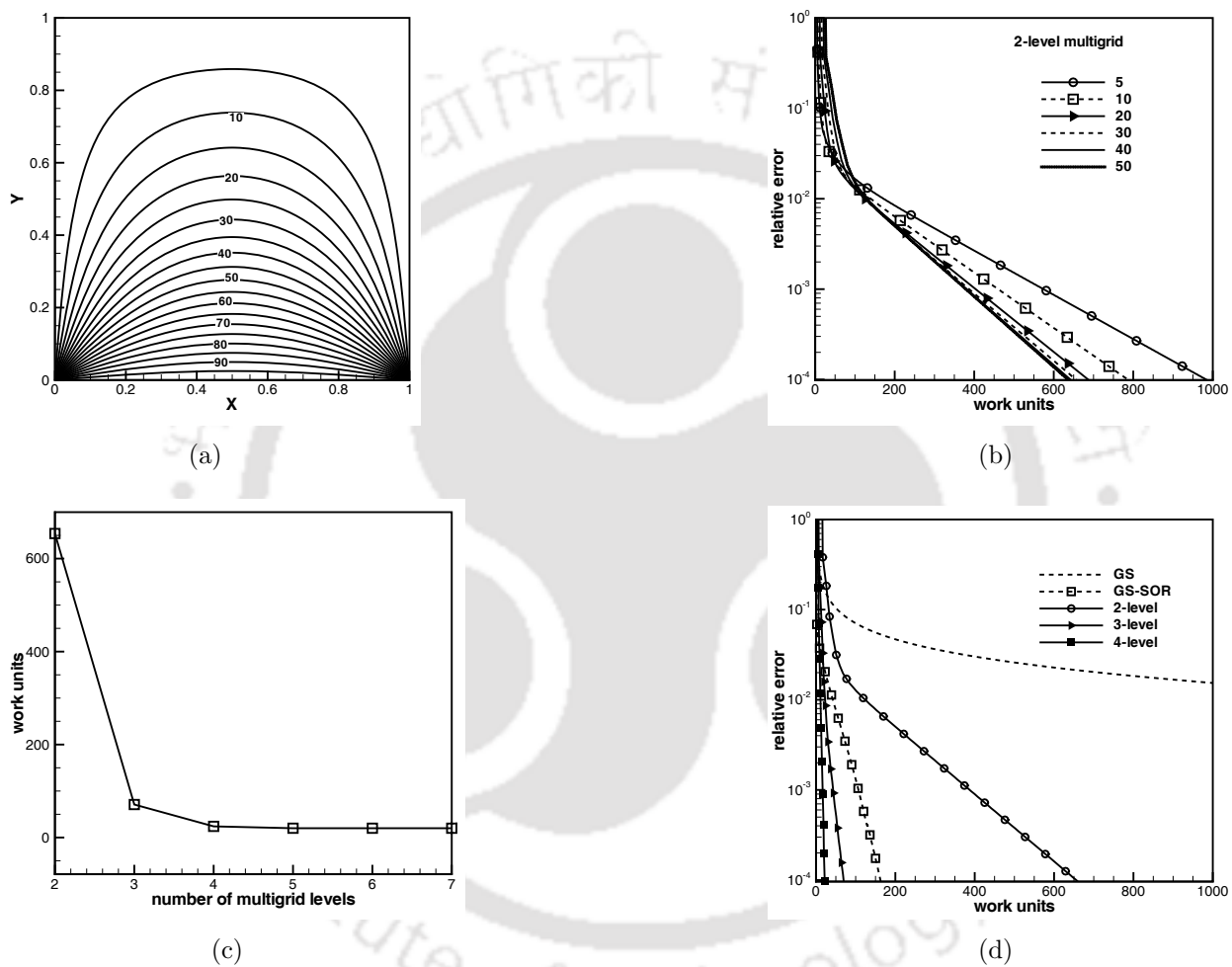


Fig. 2.8: (a) Temperature distribution for a 2D heat conduction problem on 129×129 grids (b) residual history for different combinations of 2-level MG (c) effects of multigrid levels (d) Comparison of residual fall for GS, SOR and MG method.

with SOR and 2-, 3- and 4- level multigrid is shown in Fig. 2.8(d). The convergence curves related to 3- and 4- level multigrid are considerably steeper than those related to GS and SOR. Expectedly multigrid accelerates the convergence of the Gauss-Seidel iterations significantly, which is higher than that achieved by SOR. This exercise is carried out to ascertain that multigrid performs better than SOR, so that it can be used gainfully to numerically solve the pressure-Poisson equation that requires solution at every time step in the transient fluid-flow computation.

2.6 Non-linear Multigrid Method

The multigrid method discussed in Section 2.2 is applicable to linear equations, where we transfer only the residual from one grid to another. If the equation is nonlinear we should transfer both the solution as well as the residual from the fine to coarse grid and operate on the absolute solution on all grid levels. This type of multigrid is known as the full-approximation storage (FAS) multigrid method. The solution procedure for FAS scheme is same as that of the correction scheme. Now we will outline the commonly used FAS multigrid method. Consider a system of nonlinear algebraic equations on the fine grid

$$A^f(u^f) = S^f \quad (2.28)$$

where A^f represents a nonlinear operator, f the fine grid, S the source. Assume v is an approximation to the exact solution u , then the error e and the residual R are given by

$$e^f = u^f - v^f \quad (2.29)$$

$$R^f = S^f - A^f(v^f) \quad (2.30)$$

To derive the residual equation subtract Eq. (2.30) from Eq. (2.28) to obtain

$$A^f(u^f) - A^f(v^f) = R^f \quad (2.31)$$

Using Eq. (2.29), Eq. (2.31) can be written as,

$$A^f(v^f + e^f) - A^f(v^f) = R^f \quad (2.32)$$

Eq. (2.32) is the residual equation which has to be solved on the coarser grid. Therefore, transfer Eq. (2.32) to the coarser grid to obtain

$$A^c(v^c + e^c) - A^c(v^c) = R^c \quad (2.33)$$

where c represents the coarse grid, A^c the coarse grid operator, R^c the coarse grid residual, v^c and e^c denote the coarse grid approximation to v^f and e^f , respectively. Since R^c in Eq. (2.33) is the restricted residual from the fine grid, it can be written as

$$R^c = I_f^c R^f = I_f^c (S^f - A^f(v^f)) \quad (2.34)$$

where I_f^c is the restriction operator. It has already been mentioned that in FAS method not only the residuals but the solutions should also be transferred from the fine to coarse grid. The approximation v^c in Eq. (2.33) is also restricted from the fine grid. Thus it can be represented as

$$v^c = \hat{I}_f^c v^f \quad (2.35)$$

Making use of Eqs. (2.34) and (2.35) in Eq. (2.33) yields

$$A^c(\hat{I}_f^c v^f + e^c) = A^c(\hat{I}_f^c v^f) + I_f^c (S^f - A^f(v^f)) \quad (2.36)$$

where $\hat{I}_f^c v^f + e^c = u^c$ and the solution transfer operator $\hat{I}_f^c$ may differ from the residual-transfer operator I_f^c . Eq. (2.36) is the coarse-grid equation to solve; once e^c is computed it

is prolonged to the fine grid and used to correct the fine grid approximation v^f .

$$v^f \leftarrow v^f + I_c^f e^c \quad (2.37)$$

where I_c^f represents prolongation operator. This is the essence of the full approximation storage multigrid method. A step-by-step procedure and the performance of FAS multigrid method will be discussed in Chapter 5 by applying it to the classical lid-driven cavity problem.



Chapter 3

A Multigrid-Accelerated Code on Graded Cartesian Meshes for 2D Time-Dependent Incompressible Viscous Flows

3.1 Introduction

In the present chapter we have developed a multigrid-accelerated code on graded Cartesian meshes for 2D time-dependent incompressible viscous flows. The unsteady incompressible flows occur frequently in problems of academic interest and engineering applications and one approach adopted for the solution of the unsteady, incompressible Navier-Stokes equations is the fractional-step method [22,64]. In this approach the time integration of the Navier-Stokes equations is carried out by means of the fractional-step procedure, whereby at each time step an incomplete form of the momentum equations is integrated to yield an approximate velocity field, which will in general not be divergence-free. Subsequently a correction is applied to the approximate velocity field to produce a divergence-free velocity field. Integration then proceeds to the next time-step. The correction takes the form of the gradient of a scalar, with

the scalar field obtained by solving a Poisson equation with the divergence of the approximate velocity as the right hand side. The scalar itself is either the pressure or a pressure correction, depending on the exact form of the fractional-step scheme being used. In the present work the scalar is the pressure instead of pressure correction. Such schemes are called projection methods as the correction to the velocity is accomplished via an orthogonal projection onto the divergence-free field.

Traditional fractional-step or projection methods for incompressible Navier-Stokes equation were introduced by Chorin [22,23]. The original Chorin method was modified by Kim and Moin [64] in 1985. They used the explicit Adam-Bashforth scheme for the convective terms and the second-order Crank-Nicholson for the viscous terms. The Poisson equation was solved by a direct method based on trigonometric expansion. In an improvement of the fractional-step method for incompressible Navier-Stokes equations, Le and Moin [73] solved 3D unsteady flows with fractional-step method combined with three-step Runge-Kutta-type scheme and achieved a gain in time over that obtained by Kim and Moin [64]. Time dependent turbulent flows can also be solved by the fractional-step method [21,91]. Fractional-step methods can also be extended to curvilinear coordinate system [116] and these have been modified to solve heat transfer problems [6,20,119,121].

Finite difference method is frequently used in computational fluid dynamics. The method essentially involves setting up a suitable grid in the problem domain, discretizing the governing equations with respect to the grid and solving them numerically. The common practice is to use a uniform grid, though it may not be the most appropriate one for an efficient computation. An accurate spatial resolution of the solution requires that grid points are clustered in the regions of large gradients. Hence nonuniform grids are used for many flow configurations. Finite difference formulations cannot be easily applied on nonuniform grids. The usual approach adopted is to map the physical space with a nonuniform grid on to a computational space with uniform grid where a transformed set of equations is first solved before mapping this solution back on the physical space for interpretation. The disadvantage of this approach is that, there is an increase in the number of terms to be discretized in the

transformed governing equations giving rise to added computations [57]. Many times, however, advantages of using a nonuniform grid outweighs the disadvantages mentioned above. By use of nonuniform grids it is possible to cluster grid lines in the regions of sharp gradients like shear layers and use a relatively coarser grid in the regions involving small gradients, thus obtaining a better spatial scale resolution with a smaller overall grid size. This is the reason why graded Cartesian meshes are used in the present work to have good resolution of the wall-bounded and free shear layers. It has already been mentioned that the transformed governing equations contain a larger number of terms than the original equations, which may result in some increase in the computational cost. Particularly, the pressure-Poisson equation that requires very accurate solution at every time step and consumes more than eighty percent of the total computational time needs special attention and an efficient algorithm for its numerical solution is highly desirable. When the Poisson equation is solved by iterative methods such as the point Gauss-Seidel or line Gauss-Seidel, high-frequency components of the error are effectively reduced, but the low-frequency errors are relatively difficult to remove resulting in a substantial increase in the computational time. Successive overrelaxation (SOR) is known to improve the convergence behaviour of the Gauss-Seidel method. But multigrid, which is arguably the best general convergence acceleration technique [106] has been found to work better than SOR, especially when the grid size is large. This is the reason why multigrid technique is used in the present work to compute the pressure-Poisson equation. In the present work how multigrid method performs in the numerical solution of the pressure-Poisson equation on both uniform and graded Cartesian meshes is examined in some details and also the relative performances of the algorithm for these two types of meshes is studied. Expectedly multigrid accelerates the convergence of the Gauss-Seidel iterations significantly, which in turn brings about a substantial reduction of cost in the overall transient flow computation.

In the present work a finite-difference discretization of the governing equations is carried out on a staggered grid [47, 85, 86]. It is our experience that implementation of the pressure boundary condition plays a vital role in the computational accuracy of transient flows and

that though a colocated grid is possible to use, better results can be obtained using a staggered grid. The convective terms in the Navier-Stokes equations are discretized here using a third-order accurate upwind scheme [60] and the viscous terms are discretized centrally to fourth-order accuracy. As the time accuracy of the fractional-step method used in this work is also of second-order, the transient results produced here have the potential of being very accurate. This aspect of the code is examined by solving four time-dependent fluid flow problems. We first applying it to compute the transient flow in a single-sided lid-driven cavity and then the time-marching steady flow in the same configuration. However, flow does not remain steady for all Reynolds numbers (Re) and as it increases it becomes unsteady and sometimes periodic on the way to turbulence. Computations for two such Reynolds numbers $Re = 8200$ and 10000 at which the flow becomes periodic are also carried out. Close agreement with validated results proves the accuracy of the present computation. The code is then applied to a hitherto unexamined situation of computing the transient flow in a two-sided lid-driven cavity, that involves gradual development of a free shear layer with associated vortices. Although there are no existing results with which the present ones could be compared the computations are lent prestige by the fact that the accuracy of the code has already been substantiated and that it has a good spatio-temporal accuracy. Use of high-precision arithmetic to offset accumulation of round-off errors in the transient computations lends added credibility to the results. Also the steady results, obtained by marching sufficiently in time, are compared with an existing result [87] and a very close agreement is obtained. The third flow configuration considered is the backward-facing step which also provides an excellent test case for the accuracy of our numerical code. This problem is solved in the Reynolds number range of 100 to 1000. The results are compared and excellent matching with existing results. Finally, the capability of our code is better realized when it is used to compute flow past a square prism and the famous von Kármán street is simulated very successfully. Thus it can be summarised that the purpose of the present chapter is to develop an accurate and efficient computational tool for computing incompressible transient-viscous flows using multigrid method and to gain experience about the relative performance of the

algorithm on uniform and nonuniform meshes in the four time-dependent problems.

This chapter is organized in seven sections. Section 3.2 describes the governing equations in the physical (x, y) and the computational (ξ, η) planes. Section 3.3 gives a brief description of the fractional-step method employed and Section 3.4 deals with the numerical method and Section 3.5 with the results and discussions. Finally in Section 3.6 we list our observations and conclusions.

3.2 Governing Equations in the Physical Plane

The unsteady, 2D, incompressible N-S equations in the traditional primitive variable formulation in the non-dimensional form can be written as

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \quad (3.1)$$

$$\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = -\frac{\partial p}{\partial x} + \frac{1}{Re} \left[\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right] \quad (3.2)$$

$$\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} = -\frac{\partial p}{\partial y} + \frac{1}{Re} \left[\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right] \quad (3.3)$$

Here u and v denote the velocities along x and y directions, t the time and p the pressure. In view of Eq. (3.1) solenoidality of the velocity field must be ensured at every instant in a transient computations. It may also be noted that time derivative of pressure does not explicitly appear in the governing equations and hence time-evolution of pressure cannot be computed directly from these equations. Another point worth noting is that the velocities are not dependent on the magnitude of the pressure field but only on its gradient. Values of the pressure to be used in the momentum Eqs. (3.2-3.3) are generally determined up to a constant by solving a pressure-Poisson equation that is obtained by taking the divergence of the vector momentum equations [104]. This equation containing the Laplacian of pressure on the left hand side and a source term containing various velocity gradients on the right,

in two dimensions is

$$\nabla^2 p = \nabla \cdot \mathcal{L}(\vec{q}) \quad (3.4)$$

where $\mathcal{L}(\vec{q}) = \frac{1}{Re} \nabla^2 \vec{q} - \frac{\partial \vec{q}}{\partial t} - (\vec{q} \cdot \nabla) \vec{q}$ and $\vec{q}$ is the velocity vector.

It is clear that in the computation of transient flow using the pressure-velocity formulation this Poisson equation has to be solved very accurately at every time step. As opposed to the solution of the transient momentum Eqs. (3.2-3.3), which, is a relatively simple matter with an explicit technique, the solution of the pressure-Poisson equation (3.4) is a nontrivial task that consumes a lot of computational time. For an efficient computation of transient flows using the pressure-velocity formulation it is therefore imperative that this equation is solved with an algorithm involving low computational cost. That is why a multigrid-technique is used to solve this pressure-Poisson equation in the present work. We will see later that the form of the source term in the context of the fractional-step will be somewhat different.

3.2.1 Governing Equations in the Transformed Plane

In order to use nonuniform grids, the governing equations (3.1) - (3.3) have to be transformed to a computational plane in which the computational mesh is uniform and the mathematical characteristics of the equations do not change with the transformation. The transformed version of the governing equations is then solved in the computational plane by a higher order approximation and mapping of the results back to the physical plane is done. In Appendix A we present some details of the transformation. Using these transformations, the governing equations in the computational plane corresponding to the equations (3.1 - 3.3) in the physical plane now become

$$\xi_x \frac{\partial u}{\partial \xi} + \eta_y \frac{\partial v}{\partial \eta} = 0 \quad (3.5)$$

$$\frac{\partial u}{\partial t} + (u\xi_x) \frac{\partial u}{\partial \xi} + (v\eta_y) \frac{\partial u}{\partial \eta} = -\xi_x \frac{\partial p}{\partial \xi} + \frac{1}{Re} \left[\xi_{xx} \frac{\partial u}{\partial \xi} + \xi_x^2 \frac{\partial^2 u}{\partial \xi^2} \right] + \frac{1}{Re} \left[\eta_{yy} \frac{\partial u}{\partial \eta} + \eta_y^2 \frac{\partial^2 u}{\partial \eta^2} \right] \quad (3.6)$$

$$\frac{\partial v}{\partial t} + (u\xi_x) \frac{\partial v}{\partial \xi} + (v\eta_y) \frac{\partial v}{\partial \eta} = -\eta_y \frac{\partial p}{\partial \eta} + \frac{1}{Re} \left[\xi_{xx} \frac{\partial v}{\partial \xi} + \xi_x^2 \frac{\partial^2 v}{\partial \xi^2} \right] + \frac{1}{Re} \left[\eta_{yy} \frac{\partial v}{\partial \eta} + \eta_y^2 \frac{\partial^2 v}{\partial \eta^2} \right] \quad (3.7)$$

3.3 Fractional-Step Procedure

The four step fractional-step method [64] is described here in the uniform and nonuniform Cartesian coordinates. In the first step the momentum equation (3.8) is solved for an auxiliary velocity field $\hat{u}_i$ using the pressure from the previous time step; the convective terms and viscous terms are solved explicitly using the second-order Adams-Bashforth method.

$$\frac{\hat{u}_i - u_i^n}{\Delta t} = -\frac{\partial p^n}{\partial x_i} + \frac{1}{2} [3H(u_i^n) - H(u_i^{n-1})] \quad (3.8)$$

where H is an operator representing the discretized convective and diffusive terms. In the second step Eq. (3.9) is solved to advance the auxiliary velocity field $\hat{u}_i$ to u_i^* which is of course not divergence-free.

$$\frac{u_i^* - \hat{u}_i}{\Delta t} = \frac{\partial p^n}{\partial x_i} \quad (3.9)$$

In the third step, the incompressibility condition is enforced by solving the pressure-Poisson Eq. (3.10), which is obtained by taking the divergence of Eq. (3.11) satisfying continuity for the next time step.

$$\frac{\partial}{\partial x_i} \frac{\partial p^{n+1}}{\partial x_i} = \frac{1}{\Delta t} \frac{\partial u_i^*}{\partial x_i} \quad (3.10)$$

Finally in the fourth step the velocity which satisfies the incompressibility condition is obtained by using the following correction step:

$$\frac{(u_i^{n+1} - u_i^*)}{\Delta t} = -\frac{\partial p^{n+1}}{\partial x_i} \quad (3.11)$$

Similar procedure may be followed in the computational plane. The above four-step fractional-step method for the transformed equations may be written as follows:

$$\frac{\hat{u}_i - u_i^n}{\Delta t} = -\frac{\partial \xi_j}{\partial x_i} \frac{\partial p^n}{\partial \xi_j} + \frac{1}{2} [3H(u_i^n) - H(u_i^{n-1})] \quad (3.12)$$

$$\frac{u_i^* - \hat{u}_i}{\Delta t} = \frac{\partial \xi_j}{\partial x_i} \frac{\partial p^n}{\partial \xi_j} \quad (3.13)$$

$$\frac{\partial^2 \xi_j}{\partial x_i^2} \frac{\partial p^{n+1}}{\partial \xi_j} + \frac{\partial \xi_j}{\partial x_i} \frac{\partial \xi_k}{\partial x_i} \frac{\partial^2 p^{n+1}}{\partial \xi_k \partial \xi_j} = \frac{1}{\Delta t} \left(\frac{\partial \xi_j}{\partial x_i} \frac{\partial u_i^*}{\partial \xi_j} \right) \quad (3.14)$$

$$\frac{(u_i^{n+1} - u_i^*)}{\Delta t} = -\frac{\partial \xi_j}{\partial x_i} \frac{\partial p^{n+1}}{\partial \xi_j} \quad (3.15)$$

The Poisson equation in the transformed plane is given by Eq. (3.14), which is solved using the multigrid method.

3.3.1 Poisson Solver

The fractional-step method described in the previous section is required to solve the Poisson equation Eq. (3.10) on a nonuniform grid, which is equivalent to solving Eq. (3.14) on the uniform transformed grid at each time step to satisfy the divergence-free condition. In this case the pressure-Poisson equation is derived by taking the divergence of the intermediate velocity field Eq. (3.11) subjected to continuity constraint for the next time step (given in Appendix B). On the physical plane the Poisson equation is given as

$$\frac{\partial^2 p}{\partial x^2} + \frac{\partial^2 p}{\partial y^2} = \frac{1}{\Delta t} \left(\frac{\partial u^*}{\partial x} + \frac{\partial v^*}{\partial y} \right) \quad (3.16)$$

On the transformed space this Poisson equation, given below, has more number of terms

$$\xi_{xx} \frac{\partial p}{\partial \xi} + \xi_x^2 \frac{\partial^2 p}{\partial \xi^2} + \eta_{yy} \frac{\partial p}{\partial \eta} + \eta_y^2 \frac{\partial^2 p}{\partial \eta^2} = \sigma \quad (3.17)$$

where, σ is the source term and that has the form

$$\frac{1}{\Delta t} \left(\xi_x \frac{\partial u^*}{\partial \xi} + \eta_y \frac{\partial v^*}{\partial \eta} \right) \quad (3.18)$$

The pressure-Poisson equations Eq. (3.16) or Eq. (3.17) are normally solved by the Gauss-Seidel method. The iterative solution of the pressure-Poisson equation is the most time consuming part in the overall solution procedure. For high Reynolds number flows and highly stretched grids, it is very time consuming to obtain the converged solution of the pressure-Poisson equation up to the machine accuracy. Therefore the use of a convergence enhancing scheme such as the multigrid method is desirable. In the initial stages of the computation, the solution of the Poisson equation alone takes approximately 95% of the total CPU time required to update all the primitive variables over one time step. Even though this percentage drops down in the later states of computations, there is a strong need to use some convergence-acceleration technique. Multigrid method has been chosen to accelerate the convergence rate, as it is highly suitable for elliptic equations such as the Poisson equation. As the Poisson equation is a linear equation, correction scheme is used in the present work which has been described in Section 2.2.

3.4 Numerical Method

The governing equations are discretized in space, using a finite difference formulation on a staggered, Cartesian grid [47], where velocities and pressure are calculated at different locations. The main advantage of the staggered grid arrangement is the strong coupling between pressure and velocities without requiring special interpolation techniques. This helps avoid convergence problems and oscillations in pressure fields. Another advantage of using

staggered grid for incompressible flows is that the pressure boundary conditions are not required when the momentum equations are evaluated. The boundary conditions for velocity are straightforward, and are directly imposed on the walls of the cavity whereas the pressure boundary conditions are somewhat tricky. There are no natural boundary conditions for pressure as in the case of velocity. Since pressure is coupled with the velocity, Peyret and Taylor [88] state that, “The primary difficulty with this approach (primitive variable formulation) is specification of boundary condition for pressure”. We can derive the pressure boundary condition from momentum equation by substituting velocities. The boundary conditions thus obtained are called *dynamically consistent pressure boundary conditions*. This is naturally true for colocated grid but for staggered grid there is no need of such a pressure boundary condition. For the pressure-Poisson equation the pressure gradient normal to the walls is assumed zero, i.e.,

$$\frac{\partial p}{\partial x_i} = 0 \quad (3.19)$$

The convective terms of the momentum equations are approximated with Kuwahara’s third-order upwind scheme [60] (Eq. 3.20) and viscous terms are discretized with a fourth-order central-difference scheme (Eq. 3.21) as given below:

$$\begin{aligned} \left(u \frac{\partial u}{\partial \xi} \right)_{i,j} &= u_{i,j} (u_{i-2,j} - 8u_{i-1,j} + 8u_{i+1,j} - u_{i+2,j}) / 12\Delta\xi \\ &\quad + |u_{i,j}| (u_{i-2,j} - 4u_{i-1,j} + 6u_{i,j} - 4u_{i+1,j} + u_{i+2,j}) / 4\Delta\xi \end{aligned} \quad (3.20)$$

$$\left(\frac{\partial^2 u}{\partial \xi^2} \right)_{i,j} = (-u_{i-2,j} + 16u_{i-1,j} - 30u_{i,j} + 16u_{i+1,j} - u_{i+2,j}) / (12(\Delta\xi)^2) \quad (3.21)$$

3.5 Computations, Results and Discussions

The code developed is second-order accurate in time, third-order accurate in the spatial discretization of the convective terms and fourth-order accurate in the spatial discretization

of the viscous terms. In order to evaluate the performance of the code in regard to time-wise efficiency and accuracy, it is applied to the four time-dependent flow configurations to compute transient viscous flows which asymptotically reach steady state if the solution is advanced sufficiently long in time.

3.5.1 Transient Flow in a Single-Sided Lid-Driven Square Cavity

This problem, over the years, has become the most frequently used benchmark problem for the assessment of numerical methods, particularly the steady-state solution of incompressible fluid flows governed by the Navier-Stokes equations [7,8,11,16,17,41,56]. This problem is of significant scientific interest because it displays almost all fluid mechanical phenomena for incompressible viscous flows in the simplest of geometrical settings. The geometry and the boundary conditions are shown in Fig. 3.1 where the top wall is moving and the remaining three walls are stationary. The no-slip boundary conditions have been employed on all the

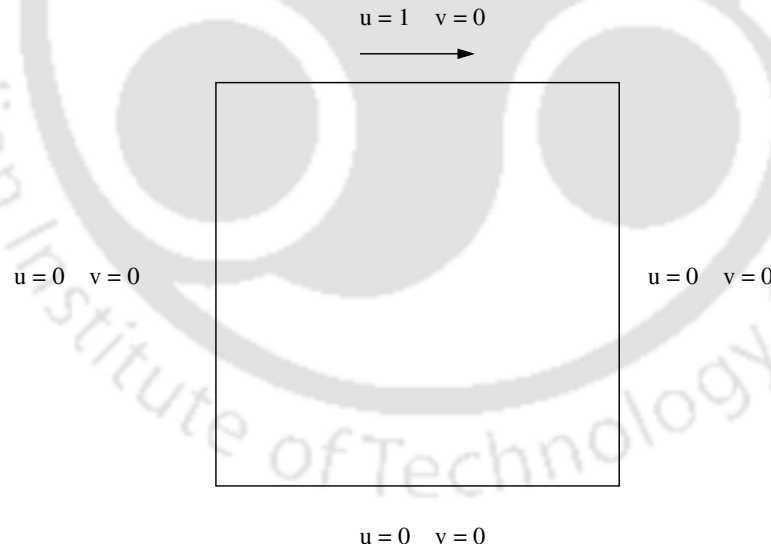


Fig. 3.1: Schematic diagram of single-sided lid-driven square cavity.

four walls. The dimensionless boundary conditions are given as

$$x = 0 : u = 0.0, \quad v = 0.0$$

$$x = 1 : u = 0.0, \quad v = 0.0$$

$$y = 0 : u = 0.0, \quad v = 0.0$$

$$y = 1 : u = 1.0, \quad v = 0.0$$

The moving wall generates vorticity which diffuses inside the cavity and this diffusion is the driving mechanism of the flow. At high Reynolds numbers, several secondary and tertiary vortices begin to appear, whose characteristics depend on Re . Because of the presence of large gradients near the walls, the grids are clustered there to capture the smaller spatial scales in these regions.

Grid Used

In this case to transform the clustered centro-symmetric grid in the physical plane (x, y) (Fig. 3.2(a)) to a uniform grid in the computational plane (ξ, η) (Fig. 3.2(b)) the following stretching function [57] is used:

$$x = \xi - \frac{\lambda}{2\pi} \sin(2\pi\xi) \tag{3.22}$$

$$y = \eta - \frac{\lambda}{2\pi} \sin(2\pi\eta) \tag{3.23}$$

where λ denotes the stretching parameter with $0 < \lambda < 1$. It may be noted that $\lambda = 0$ results in a uniform grid and $\lambda = 0.9$ produces dense clustering. In the present case λ is taken as 0.6.

Multigrid Solution of the Pressure-Poisson Equation

In the present study a 4-level V-cycle multigrid shown in Fig. 3.3 has been used in all the computations to solve the pressure-Poisson equation and the number of sweeps used



Fig. 3.2: A 65×65 grid in (a) physical plane and (b) computational plane.

in various grid-levels is given inside the corresponding circles. For a 2-level scheme on a

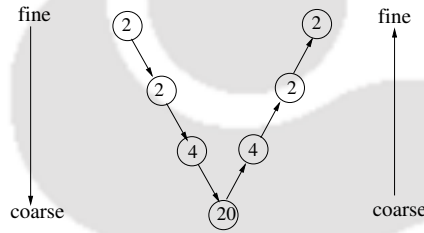


Fig. 3.3: 4-level V-cycle multigrid.

129×129 grid at $Re = 3200$, performance is seen to improve by increasing the number of sweeps in the coarser grid. When the number of sweeps increases from 7 to 60, the number of equivalent fine grid sweeps required for the convergence ultimately reduce from 927 to 321, which represents a reduction of computational effort to nearly one-third for the 2-level scheme. For $Re = 3200$ on a 129×129 grid, it is observed that using four or five level of grids is good enough as further increase of levels produce no time-wise gain. Specifically, it was observed that 3736, 321, 164, 36, 29, 29 and 29 work units (WU) (equivalent fine-grid sweeps) were required when using 1-, 2-, 3-, 4-, 5-, 6- and 7-levels respectively (Fig. 3.4(a)). To gain an insight into the convergence behaviour after around 100 time steps, at $t = 0.1$ a plot between the pressure error and number of iterations required to solve the pressure-Poisson equation is shown in Fig. 3.4(b). Before examining overall quantitative performance data, it is instructive to consider the convergence histories in Fig. 3.4(b) obtained with the single

grid and with sequences of 2-, 3- and 4-levels. Expectedly the convergence curves relating

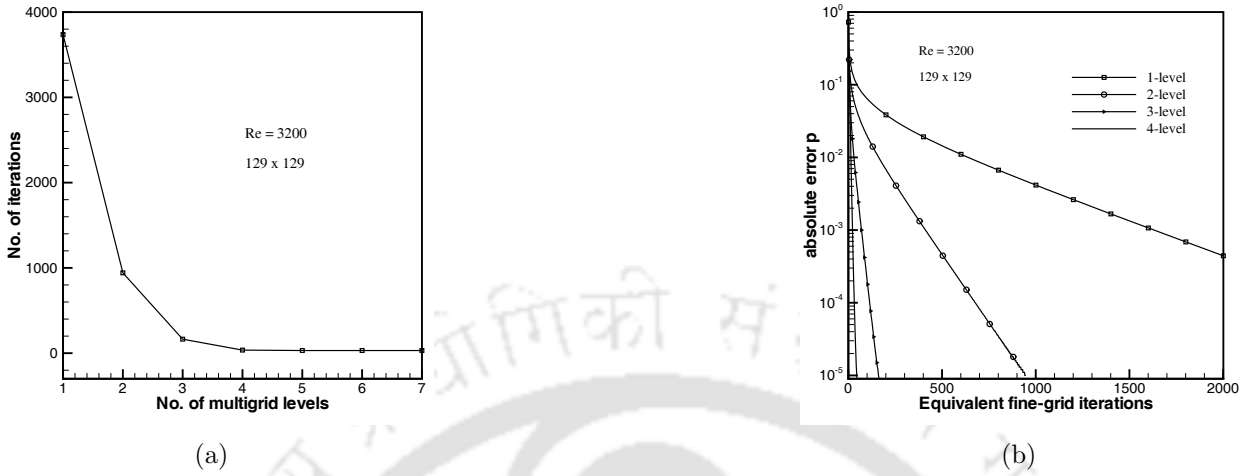


Fig. 3.4: Convergence of multigrid on a 129×129 grid for $Re = 3200$ at $t = 0.1$: (a) effect of multigrid levels, (b) pressure error history.

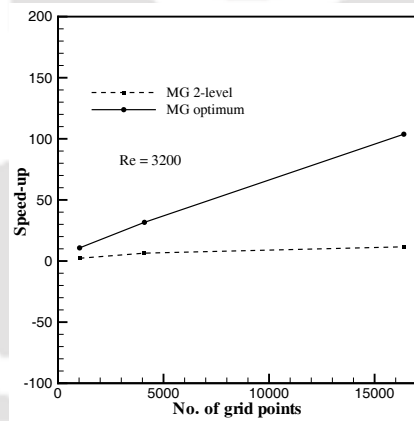


Fig. 3.5: Speed-up for multigrids with a 2- and 4(optimum)-level.

to multigrid are considerably steeper than that relating to the single grid. The ratio of the computational effort in work units for the single grid to that for the multigrid may be termed speed-up. As the convergence limit of error is lowered speed-up is seen to increase. This is because on a single grid error-reduction rate becomes smaller and smaller as the iteration progresses whereas multigrid maintains more or less the same error-reduction rate throughout. The performance of the multigrid with the optimum number of levels is truly amazing. The number of sweeps is seen to be nearly independent of the number of grid points used. Only 29 sweeps were required for the finest grid compared to 3736 for the conventional

Gauss-Seidel method. This is a reduction in effort by a factor of 128. The multigrid with the optimum number of levels, 4 in the present case, requires only $1/11^{th}$ as much effort as the multigrid with a 2-level scheme. The speed-ups obtained by 2- and 4-level multigrids are plotted against grid nodes in Fig. 3.5. We note that this convergence rate is achieved on a uniform Cartesian grid. A similar behavior has been experienced on a graded Cartesian grid.

Grid- and Time-Step-Independence Study

Fig. 3.6(a-b) respectively show the horizontal velocity along the vertical centreline and vertical velocity along the horizontal centreline of the cavity at instant $t = 20.0$ captured with three different time steps $\Delta t = 0.001, 0.0005$ and 0.0001 for $Re = 3200$. These profiles plotted for a 129×129 grid show no discernable differences, proving the time-accuracy of the present computations together with the fact that the transient results are also physically meaningful. Therefore, a time step of 0.001 has been used for all subsequent computations. Fig. 3.7(a-b) show the steady state vortices at the bottom right corner of the cavity for



Fig. 3.6: Time-step-independence study: (a) horizontal velocity along vertical centreline and (b) vertical velocity along horizontal centreline.

uniform and nonuniform grids, respectively at $Re = 1000$. It is clear that without clustering in the vicinity of the walls a uniform grid cannot predict appropriate tertiary corner vortices at the bottom right corner (Fig. 3.7(a)), whereas a nonuniform grid is able to resolve the

tertiary vortices accurately (Fig. 3.7(b)). This is the justification for using a nonuniform grid with associated complexities, in that it allows more number of points to be put in the regions of sharp flow gradients thus obtaining better scale resolution. Fig. 3.8(a-b) present



Fig. 3.7: Tertiary vortex at the bottom right corner of the cavity at $Re = 1000$: (a) uniform grid and (b) nonuniform grid.

a comparison of the steady-state horizontal velocities on the vertical centreline and the vertical velocities on the horizontal centreline of the square cavity with those obtained by Ghia *et al.* [41] for both uniform and nonuniform grids. Both the results agree well with those reported in [41] and there is negligible deviation of velocity profiles between the uniform and nonuniform grids. To lend further credibility to the present computations we also compare the present centreline u and v velocity profiles at $Re = 1000$ with the accurate results of Botella and Peyret [11] in Fig. 3.9. The high accuracy of the present computations are brought out by the close matching of the results.

Transient Results for $Re = 1000$

For transient computations, in order to give a divergence-free initial velocity field, x -component of velocity u and y -component of velocity v are both taken as zero at all grid nodes except on those on the moving top wall where u is taken as 1 and v as zero. Fig. 3.10 shows the time evolution of the streamline pattern for $Re = 1000$ till the steady state is reached. At time $t = 1.25$ (Fig. 3.10(a)) a recirculation zone is seen near the top right corner of the

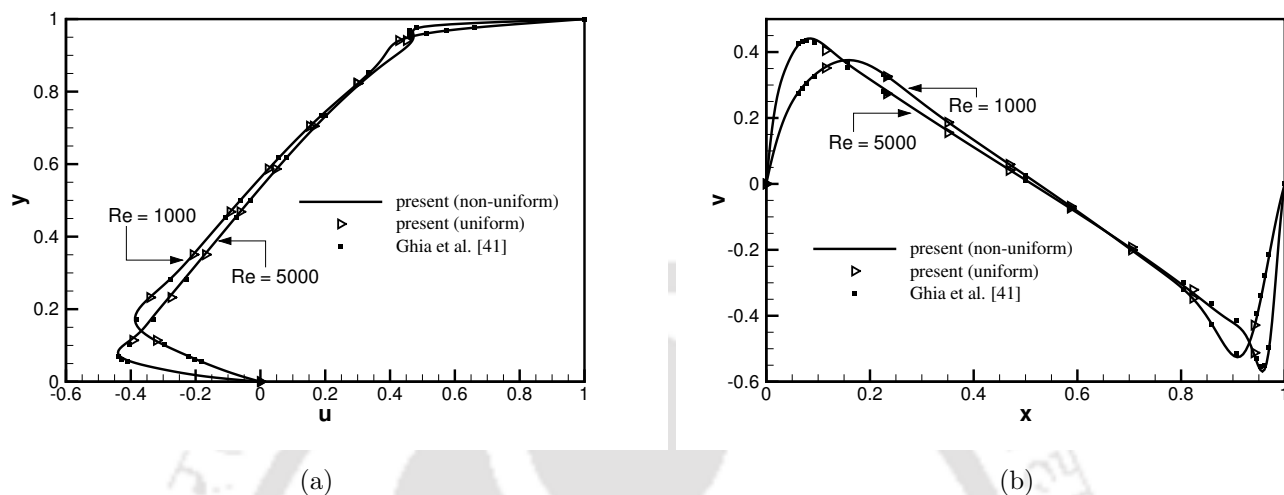


Fig. 3.8: Comparison of steady-state velocity profiles: (a) horizontal velocity along the vertical centreline and (b) vertical velocity along the horizontal centreline for the single-sided lid-driven cavity.

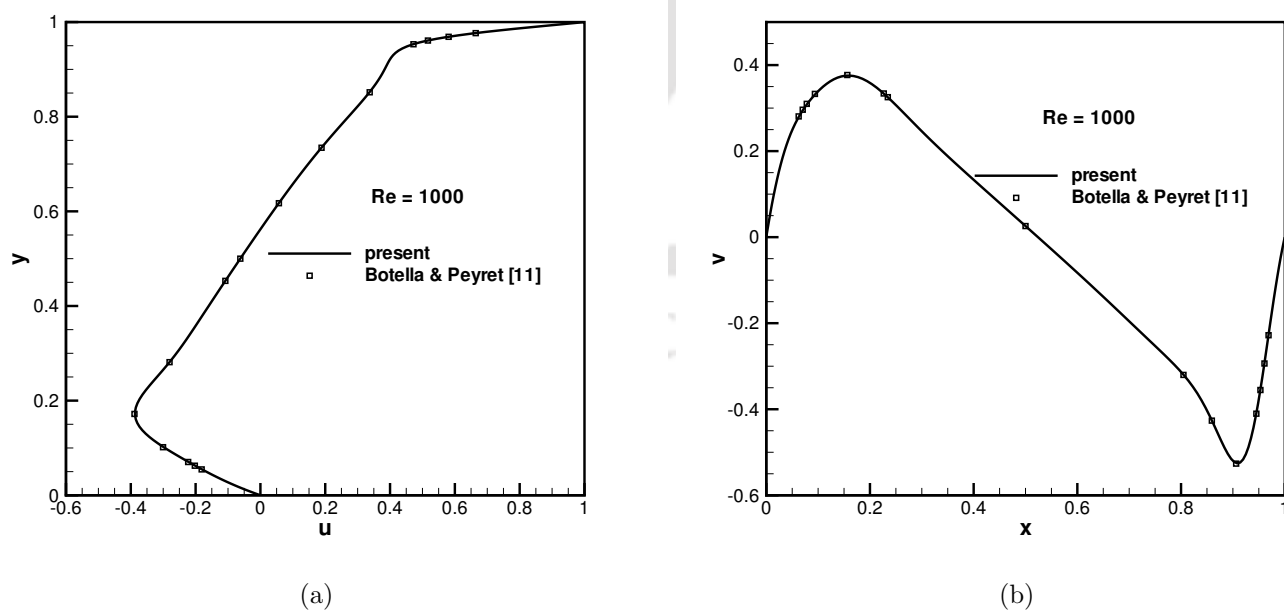


Fig. 3.9: Comparison of steady-state centreline u and v profiles with Botella and Peyret [11] for the single-sided lid-driven cavity at $Re = 1000$.

cavity. From Fig. 3.10(a-f) it is seen that this primary vortex core gradually moves towards the geometric centre of the cavity. At $t = 5.0$ (Fig. 3.10(c)) a small secondary recirculation zone at the bottom right corner is seen and another such zone slightly below the centre of the right wall, which with time moves downwards and merges with the secondary vortex at the bottom right corner. Also seen at this instant is the emergence of an incipient vortex at the bottom left corner, which grows bigger with time. At $t = 10.0$ (Fig. 3.10(d)) it is seen that after the two secondary vortices merge at the bottom right corner the resultant vortex there is much larger and the secondary vortex at the bottom left corner also grows bigger. At time $t = 40.0$ (Fig. 3.10(e)) the bottom left secondary vortex is seen to be much bigger although there is not much change in the bottom right vortex. Steady state, for a mass residual of 10^{-5} , is obtained at $t = 104$ (Fig. 3.10(f)) where, compared with $t = 40.0$, some minor readjustment is seen to take place especially at the top left corner. By examining the evolution of the streamline pattern two important observations can be made. As mentioned earlier the primary vortex core generated at the top right gradually moves with time towards the geometric centre of the cavity though it does not actually reach there even at the steady state for $Re = 1000$ (Fig. 3.10(f)). Secondly when the top wall moves from the left to right, spots of important near-wall changes gradually move with time from the top left corner in a clockwise sense along the cavity walls.

Asymptotically Obtained Steady Solutions

The steady-state streamline patterns obtained by time-marching with the present code for $Re = 1000, 3200, 5000$ and 7500 are shown in Fig. 3.11, which agree quite well with those in [41]. It may be concluded that though we use a 129×129 grid, as a result of grid-clustering near the wall and good accuracy of the spatial discretization, the results obtained are very accurate. Apart from the changes of the near-wall vortex pattern with Re , another conclusion that can be made is that as the Reynolds number increases the proximity of the primary vortex core and the geometric centre of the cavity increases.

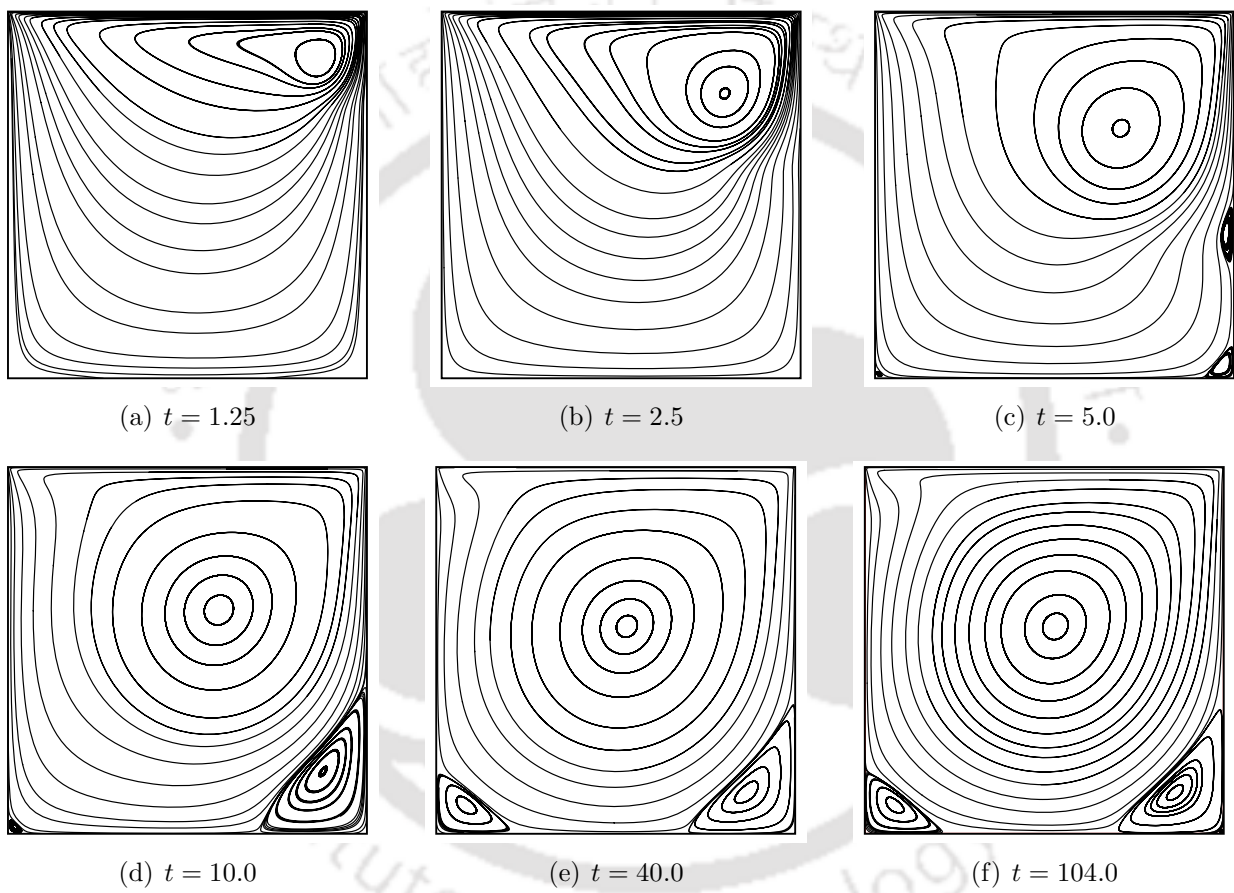


Fig. 3.10: Time evolution of streamlines for the single-sided lid-driven square cavity at $Re = 1000$.

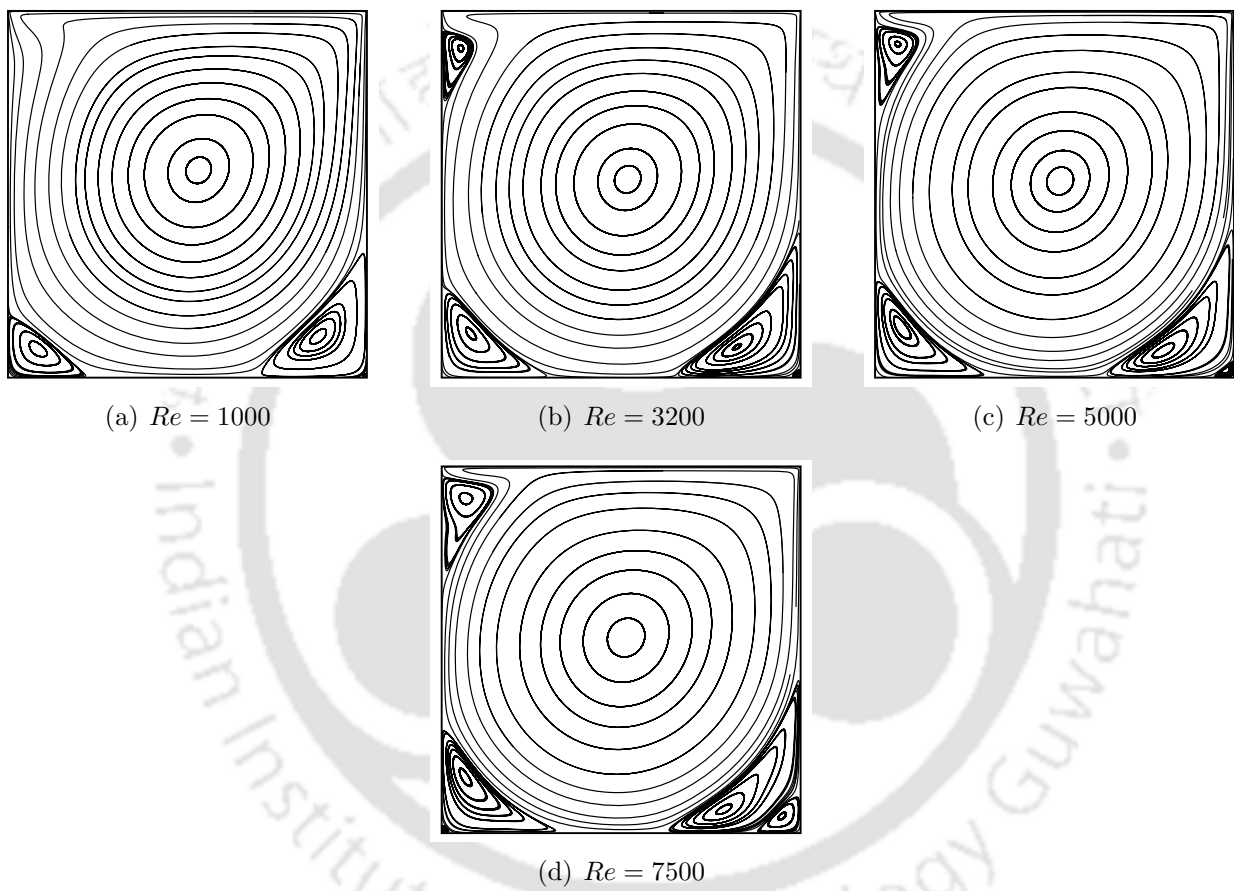


Fig. 3.11: Steady-state streamline patterns for the single-sided lid-driven square cavity on a 129×129 clustered grid.

Periodic Solutions

Until $Re = 7500$, we see that the transient computations asymptotically approach a steady solution on both 129×129 and 257×257 grids. To lend credibility to our unsteady code, we carried out our computations for still higher Reynolds numbers to capture periodic solutions reported in the literature [17]. It has been found from earlier works [7, 17, 37] that the first Hopf bifurcation takes place around $Re = 8000$. In order to find the periodic solution, computations are carried out first for $Re = 8200$. The numerical solution develops a time periodic pattern on a 257×257 grid, whether uniform or nonuniform. The streamline pattern for $Re = 8200$ is shown in Fig. 3.12. The pattern plotted at different instants of time

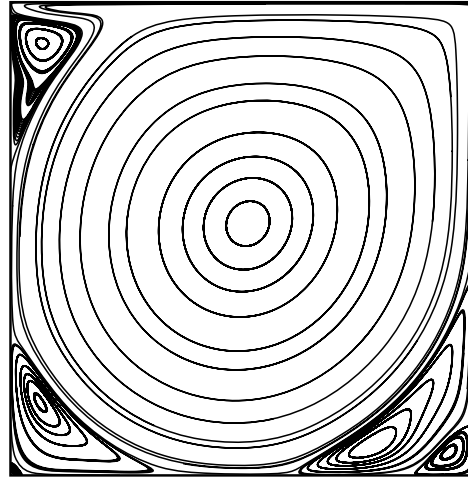


Fig. 3.12: Streamline pattern for $Re = 8200$.

looks almost identical as the fluctuations are small in magnitude. Probing further, however, reveals that small periodic oscillations occur in the solution. The time evolution of u , the x -component of velocity at the monitoring point $(14/16, 2/16)$ is plotted in Fig. 3.13(a) and a Fourier analysis performed for this signal is shown in Fig. 3.13(b). From this figure, the frequency is seen to be 0.45 and has only one dominant harmonic and one subharmonic, confirming that the solution is periodic. The behaviour is exactly identical for other quantities such as y -component of velocity, pressure and vorticity. Moreover similar results are obtained at the other points considered $(2/16, 2/16)$, $(2/16, 14/16)$ and $(14/16, 14/16)$ with some variations in amplitude of the signals. At $Re = 8200$ these points are chosen so as to

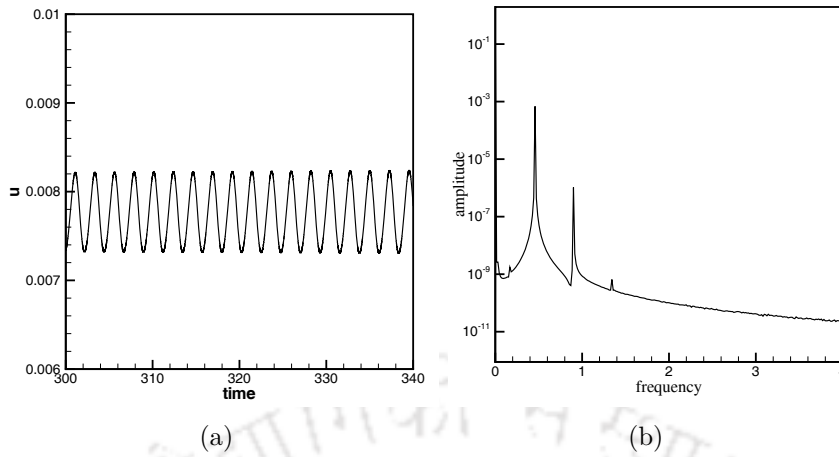


Fig. 3.13: (a) Horizontal velocity history and (b) power spectrum at monitoring point (14/16, 2/16) for $Re = 8200$.

analyse the behaviour of the solution in every part of the cavity. The variation in streamline pattern is not illustrated because the variations in velocities are small and there are no easily distinguishable differences in the patterns. It is expected that further increase in Re is likely to increase the amplitude of the oscillations. We therefore, carried out the computations at $Re = 10000$. This value was the subject of discussion for quite a long time, as the question was to know whether a stable steady solution is obtained or not for this Reynolds number. Though some of the previous works [8, 32, 41, 98] reported stable steady solutions for this value, there are some others [7, 17, 37] that reported existence of periodic solution at this Reynolds number. Therefore to explore the actual potential of the present code to compute transient flows we decided to use it to re-examine this interesting and somewhat controversial issue. The x -component of velocity plotted in Fig. 3.14(a) with time indicates that the solution reaches a stable periodic state. A power spectrum plotted in Fig. 3.14(b) using Fourier analysis confirms that the solution is periodic with one dominant harmonic and two subharmonics. After the flow reaches a stable periodic state, a phase diagram (Fig. 3.14(c)) is drawn between u and v at the monitoring point (2/16, 14/16). These plots indicate a perfect periodic pattern for the solutions obtained by us. To demonstrate the global periodic nature of the flow, in Fig. 3.15 the streamline pattern is plotted at discrete time intervals for roughly a period ($\Delta t = 2.3$) of the oscillations. Clearly the streamline pattern is repeated

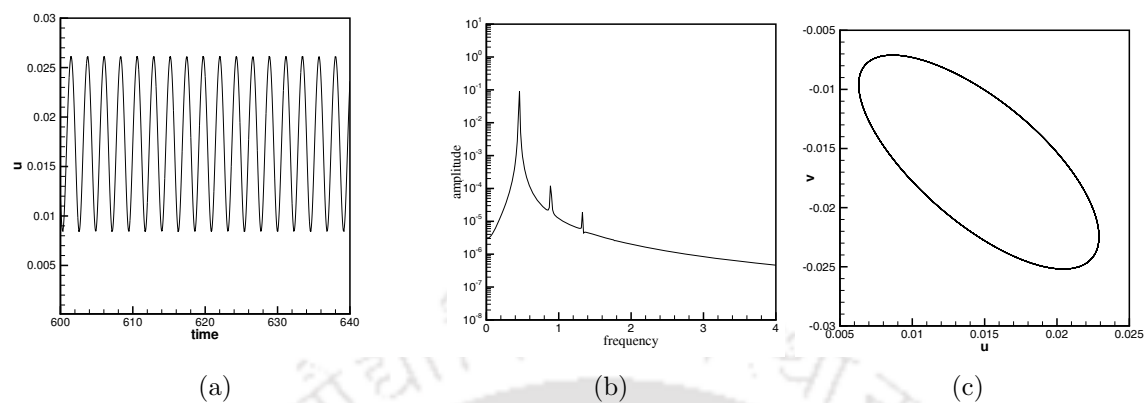


Fig. 3.14: (a) Horizontal velocity history, (b) power spectrum and (c) phase portrait on 129×129 grid at monitoring point $(2/16, 14/16)$ for $Re = 10000$.

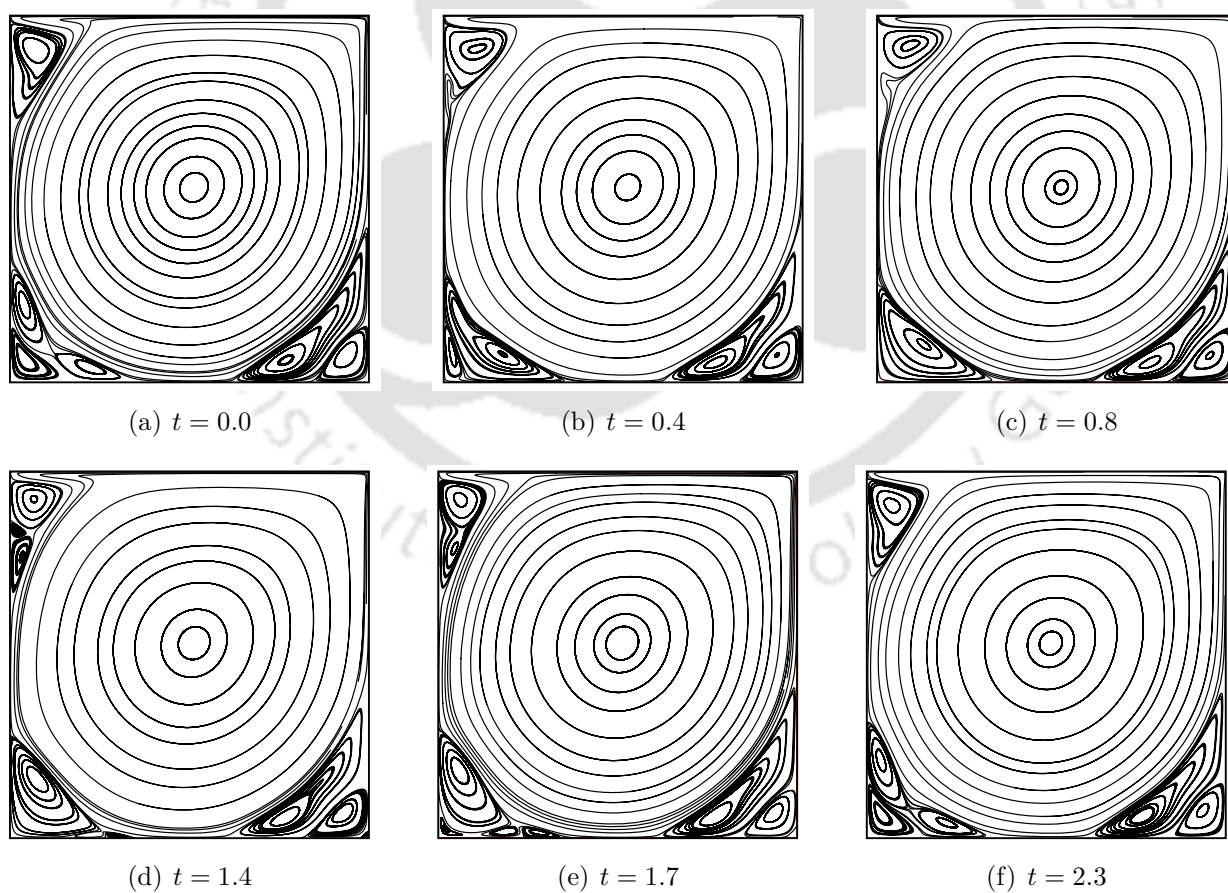


Fig. 3.15: Evolution of streamlines during one time period for $Re = 10000$ on grid 129×129 .

after a period. Indeed, the primary vortex is still attached to the three walls of the cavity but the secondary vortices are unstable. Also we observe that there are persistent oscillations at the top left and bottom left secondary vortices. The other significant changes during one period are the periodic appearance and disappearance of two tertiary vortices at the bottom left and the top left. These observations strongly indicate the global periodic nature of the solution in the entire domain, which substantiates the observations in [7, 17, 37]. It may be noted the main aim of the present work is to explore and establish the potential of the code and by capturing this periodic flow behaviour the code has established its accuracy and ability. Another aim of the present work is to carry out a comparison exercise to evaluate the performance of multigrid over single grid and also of multigrid for the uniform and nonuniform grids.

Multigrid Performance

Tables 3.1 and 3.2 give the CPU times of multigrid and single grid computations to reach the asymptotic steady state and the time-wise speed-up achieved by multigrid for various Reynolds numbers on both uniform and nonuniform grids of size 129×129 . For the same fall of residual, time-gain by multigrid is impressive. The time-wise speed-up achieved by multigrid at ‘steady state’ is ten or more for the various Reynolds numbers on the uniform grid, which for the nonuniform grid is about ten or slightly less. The fact that on the nonuniform grid multigrid performance is somewhat poorer can be attributed to the appearance of additional terms in the transformed pressure-Poisson equation for the nonuniform grid. It is also observed that work units required to reach the steady state increase as Re increases for both single-grid and multigrid. This can be attributed to the fact that high Reynolds number flows contain multiplicity of scales, which introduce high frequency errors into the computational process. The time-wise speed-up achieved by multigrid is also seen to decrease as Re increases for both uniform and nonuniform grids. Another point worth noting is that the CPU time and speed-up appearing in Tables 3.1 and 3.2 give the most conservative picture of the multigrid performance and the speed-up achieved for initial instants of time is

considerably higher. This fact is observed from Table 3.3 where the speed-ups achieved by multigrid at $t = 10$ have been listed for both uniform and nonuniform grids. This is because as the steady state is approached pressure does not change much with time and multigrid does not exert as much influence on the convergence rate as it does on earlier instants of time. All the CPU times given in Tables 3.1, 3.2 and 3.3 corresponds to the computations carried out on a Dual Xeon 3.2 GHz based machine.

Table 3.1: Performance of multigrid for a 129×129 grid on a uniform grid at various Reynolds number for the single-sided lid-driven square cavity

Reynolds number	CPU time (minutes)		Speed-up (SG/MG)
	Single-grid	MG 4-level	
1000	77.12	6.18	12.25
3200	162.48	13.48	11.8
5000	390	36.48	10.6
7500	699.42	68.36	10.2

Table 3.2: Performance of multigrid for a 129×129 grid on a nonuniform grid at various Reynolds number for the single-sided lid-driven square cavity

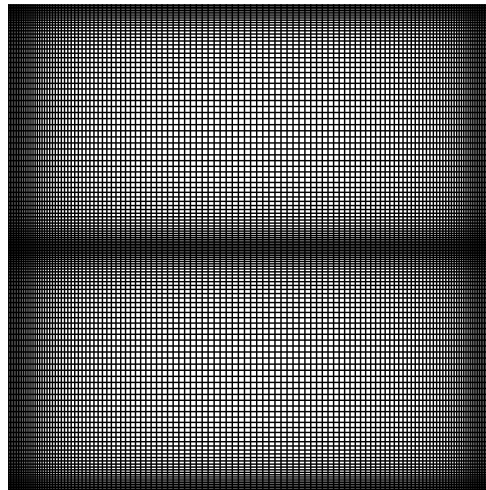
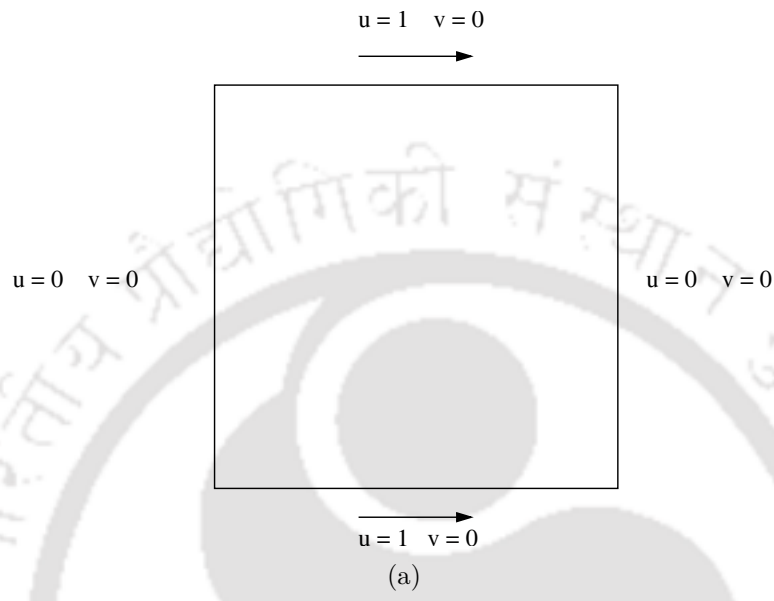
Reynolds number	CPU time (minutes)		Speed-up (SG/MG)
	Single-grid	MG 4-level	
1000	157.46	16.06	9.8
3200	387.12	41.12	9.4
5000	535.18	64.03	8.3
7500	716.18	88.03	8.1

Table 3.3: Performance of multigrid on uniform and nonuniform grids at $Re = 1000$ and $t = 10$ for the single-sided lid-driven square cavity

Grid	CPU time (minutes)		Speed-up (SG/MG)
	Single-grid	MG 4-level	
uniform grid	49.18	1.42	29
nonuniform grid	61.12	2.54	21

3.5.2 Transient Flow in a Two-Sided Lid-Driven Square Cavity

The validation exercises, results and discussions in the Subsection 3.5.1 establish the accuracy and the capability of the present code to compute transient viscous flows. After having thus gained confidence in the code it is then applied to compute the transient flow in a configuration for which no transient results exist. The problem consists of a square cavity filled with a stationary fluid that is set into motion by the sudden simultaneous movement of the top and the bottom walls from the left to right (Fig. 3.16(a)). A similar problem involving rectangular cavities has been studied by Kuhlmann *et al.* [68] to produce multiple steady solutions. Perumal and Dass [87] used lattice Boltzmann method to compute steady solution for the problem, with which the present steady results obtained asymptotically by marching with time are compared. The reason for selecting this flow configuration for numerical study is that it is fraught with interesting flow features including gradual development of a free shear layer and associated off-corner secondary vortices. Use of a grid (Fig. 3.16(b)) with appropriate clustering not only to capture the wall bounded shear layers but also the free-shear layer and use of multigrid suggests that the present computations combine accuracy, space-grid economy and time-wise efficiency. Computations are carried out for $Re = 100, 400, 1000$ and 2000 . Fig. 3.17 shows the steady-state streamline patterns asymptotically obtained from the present transient computations. These patterns agree quite well with those reported in [87]. Fig. 3.17(a) shows that at $Re = 100$ two counter-rotating primary vortices appear and there is no trace of any near-wall secondary vortices. At $Re = 400$ (Fig. 3.17(b)), apart from the



(b)

Fig. 3.16: Two-sided lid-driven square cavity flow: (a) flow configuration (b) grid (129×129).

primary vortices a counter-rotating pair of secondary vortices symmetrically placed about the horizontal cavity centreline is seen to appear near the centre of the right wall. Also a free shear layer about this centreline is now more prominent. Fig. 3.17(c-d) shows the streamline patterns at $Re = 1000$ and 2000 respectively. It is seen that from $Re = 400$ to $Re = 1000$ the size of the secondary vortices increases significantly whereas from $Re = 1000$ to $Re = 2000$ the increase in size is very small. It is also seen that with an increase in Re the distance of the top and bottom primary vortex cores from the top right and bottom right corners increases and at $Re = 2000$ they take position close to the centres of the top half and the bottom half of the cavity. The counter-rotating pairs of primary and secondary vortices maintain their symmetry about the horizontal centreline for all the Reynolds numbers. Fig. 3.18(a) plots the x -component of velocity along the vertical centreline ($x = 0.5$) of the

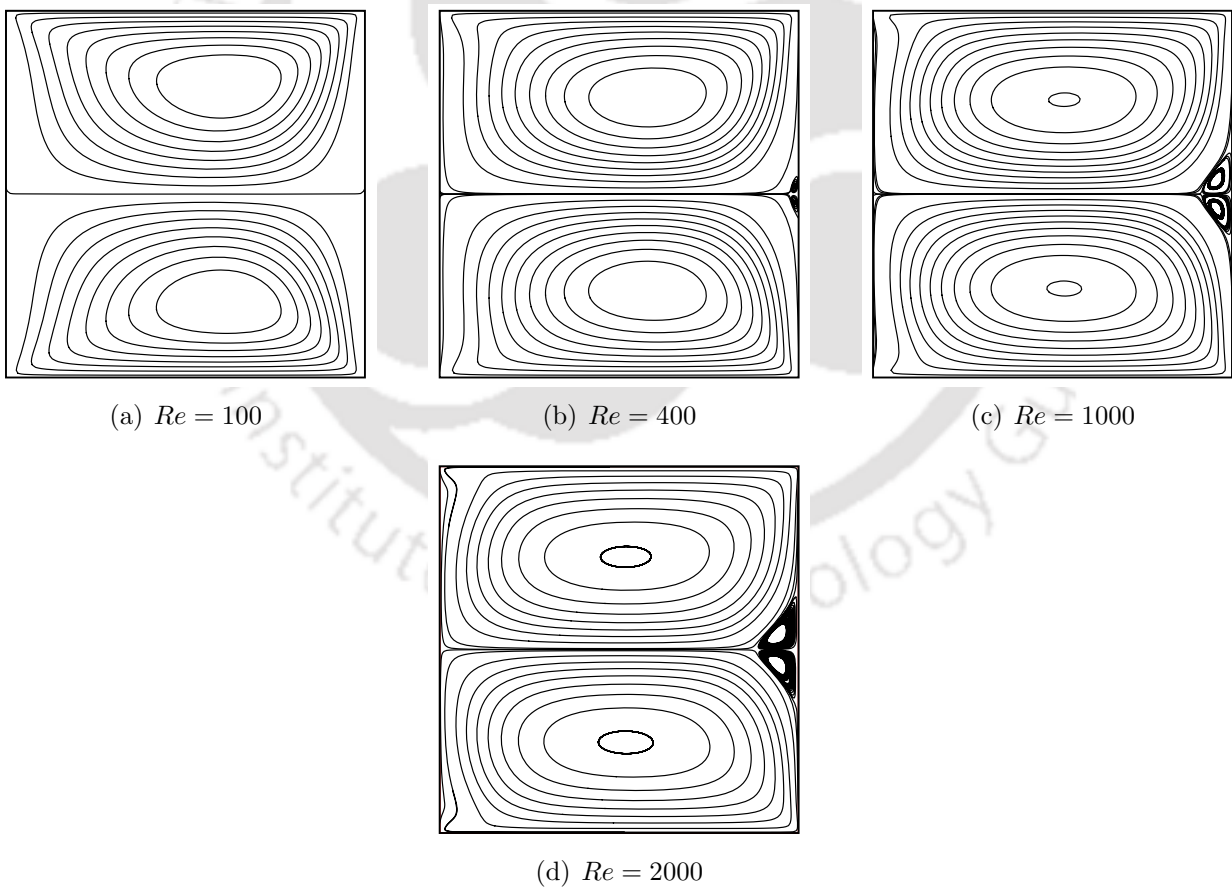


Fig. 3.17: Steady-state streamline patterns for different Reynolds numbers for the two-sided lid-driven square cavity on a 129×129 clustered grid.

cavity at $Re = 100$ and $Re = 1000$ and Fig. 3.18(b) plots the y -component of velocity along the horizontal line $y = 0.75$ at the same Reynolds numbers. Plotted alongside these curves are those reported in [87]. Close agreement of the present steady results with the very carefully produced results in [87] shows that the spatial accuracy and resolution obtained by the present code on a relatively coarse clustered grid is quite high. Another conclusion that can be drawn from the present computation concerns the existence of a stable steady solution at $Re = 2000$. Kuhlmann *et al.* [68] points out that for a two-sided lid-driven rectangular cavity with parallel motion of the walls and a certain aspect ratio greater than unity and beyond, multiple steady solutions exist; whereas for a two-sided lid-driven square cavity there are no multiple solutions and steady solutions are unique. However, as steady equations were numerically solved to produce the results reported in [87] it was not very clear until now whether there exists a unique steady and stable solution for the present flow-configuration at $Re = 2000$. As the present code with proven ability to produce periodic solutions is seen to produce steady results at $Re = 2000$, it conclusively proves that for this Reynolds number and configuration, there indeed is a unique steady solution.

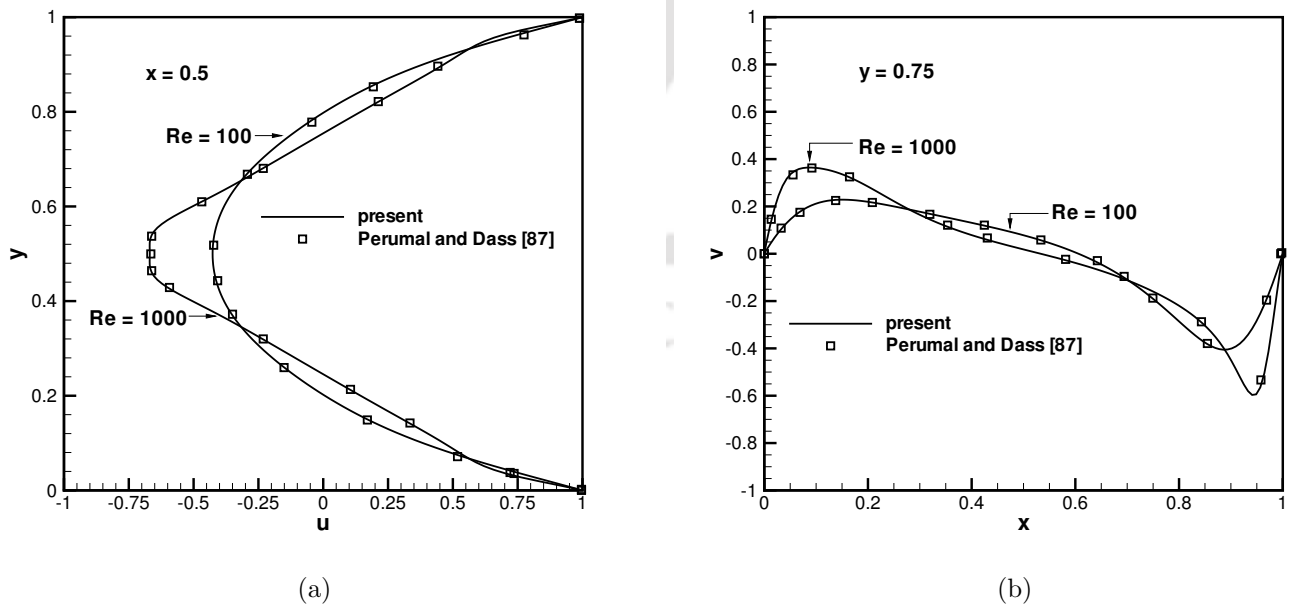


Fig. 3.18: Steady-state velocity profiles: (a) horizontal velocity along the vertical centreline and (b) vertical velocity along the horizontal line at $y=0.75$, on a 129×129 clustered grid.

For computing transient results, the initial divergence-free velocity field used in this work is obtained by taking both u and v as zero at all grid-nodes except those on the top and the bottom moving walls where u is taken as unity and v as zero. Fig. 3.19 gives the picture of time-wise evolution of the streamline pattern at $Re = 1000$ till the steady state is reached. Fig. 3.19(a) shows the streamline pattern at $t = 1.0$ when the primary vortex cores are seen close to the top right and bottom right corner symmetric to each other about the horizontal centreline. At this instant the secondary vortices have not developed as yet. At time $t = 3.0$ (Fig. 3.19(b)) the two primary vortex cores are seen to move away from the nearest corners maintaining their symmetry about the horizontal centreline. Also seen at this instant is the emergence of a pair of two counter-rotating secondary vortices symmetrically placed about the horizontal centreline near the centre of the right wall. At time $t = 5.0$ (Fig. 3.19(c)) the movement of the primary vortex cores away from their nearest corners is seen to continue whereas the size of the pair of secondary vortices enlarges. At time $t = 7.0$ (Fig. 3.19(d)) the movement of the primary vortex cores away from the respective corners is seen to continue but secondary vortex sizes do not change much compared with those at $t = 5.0$. Fig. 3.19(e) shows the streamline patterns at time $t = 10.0$, when the primary vortex cores are seen to move further away from their corners of origin and almost reach the centres of the top and bottom halves. For a fall of mass residual below 10^{-5} the steady state is reached at time $t = 82.2$. At this steady state the two counter-rotating primary vortex cores are stably positioned very close to the geometric centres of the top and bottom halves of the cavity. Also there are no changes in the positions and shapes of the counter-rotating pair of secondary vortices. Observing these figures in the chronological sequence also reveals that the flow gradient in the free shear layer about the horizontal centreline of the cavity becomes sharper with time.

The accuracy of the present steady results for the two-sided lid-driven cavity flow obtained on a relatively coarse clustered grid of 129×129 is already seen to be high as they compare very well with carefully produced results [87] on much finer uniform grids. This has been possible as a result of using discretizations of good spatial accuracy and adequately clustered

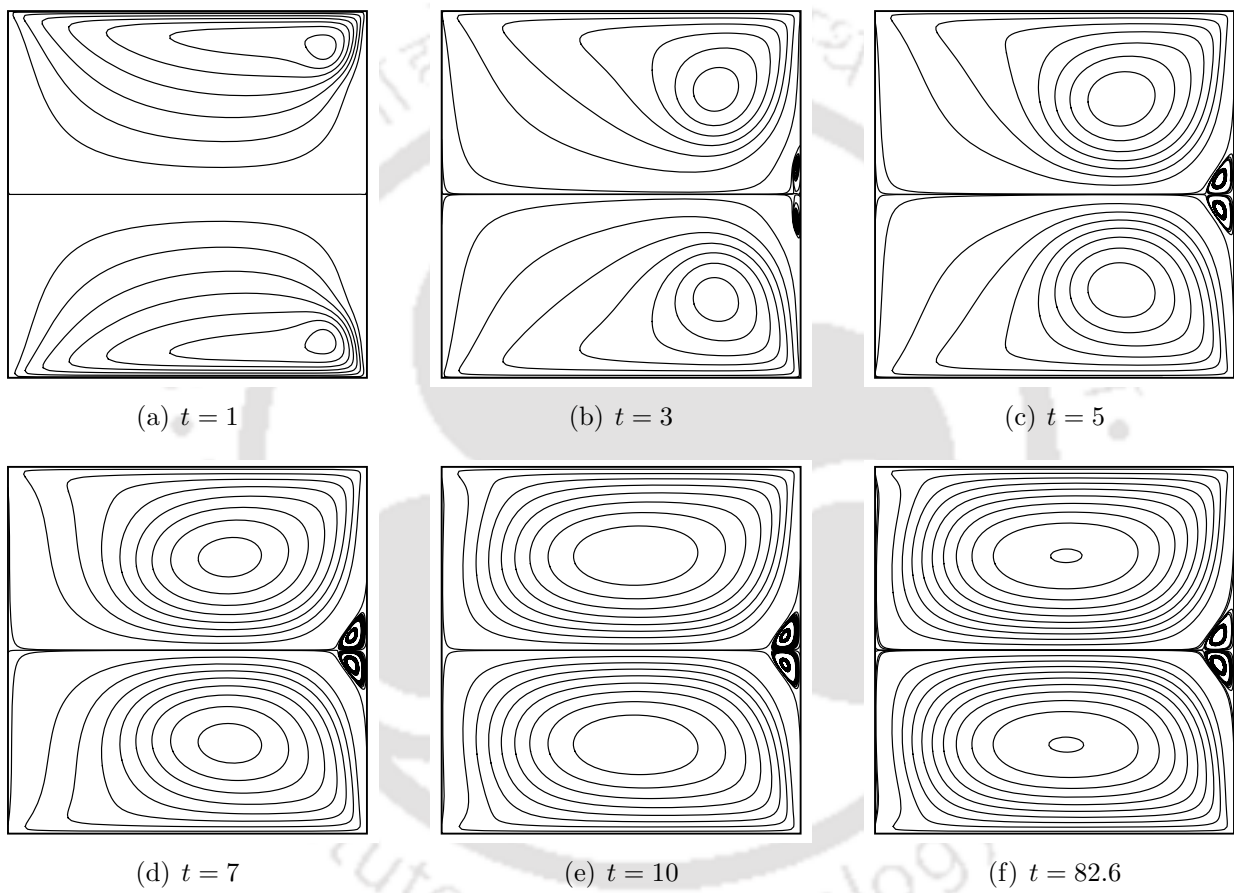


Fig. 3.19: Time evolution of streamlines for the two-sided lid-driven square cavity at $Re = 1000$.

grids in the regions of sharp flow gradients. The transient computations for this hitherto unexamined flow configuration are also highly reliable. This is because of the second order time accuracy of the scheme and the care taken to maintain time-step-independence of the results throughout. Use of high precision arithmetic to avoid possible accumulation of round-off errors lends further credibility to the results. The time-wise efficiency of the transients computations is also high as multigrid has been used to good effect. Overall these results are produced accurately and efficiently with grid economy.

3.5.3 Backward Facing Step Flow

The previous sections were devoted to demonstrate the ability of the present code to compute accurately the transient and the steady solutions of the single-sided and two-sided lid-driven cavity problems. The next problem that we have considered is the flow over the backward-facing step (BFS) in a channel (Fig. 3.20). This is one of the most important benchmark problems used in CFD. It has an outflow boundary condition, flow separation, reattachment and several recirculation zones. This flow configuration has been addressed by numerous authors using a wide variety of experimental and numerical methods [5, 40, 46, 59, 101]. This flow configuration also provides an excellent test case for the accuracy of the present code as various authors have listed a flow reattachment length as a function of Reynolds number. The main feature of this flow configuration is a recirculation region just downstream of the step. Numerical simulations are carried out for Reynolds numbers 100 – 1000 on a 512×128 nonuniform grid. Yee *et al.* [118] showed that for BFS flow when a coarse grid is used, one can obtain a spurious oscillatory numerical solution. They have clearly stated that adequate grids are necessary in order to avoid spurious solutions for the BFS flow.

The flow configuration is shown in Fig. 3.20. At the inlet (ED) a parabolic velocity profile is prescribed and is given by $u(y) = 1.5(1.0 - (\frac{y-0.75}{0.25})^2)$ for $0.5 \leq y \leq 1.0$. This produces a maximum inflow velocity of 1.5 and an average inflow velocity of 1.0 and at the step (AE) $u = v = 0$ for $0 \leq y < 0.5$. The downstream zero-gradient boundary conditions are prescribed at a distance of $30H$, where H is the height of the channel. This length allows

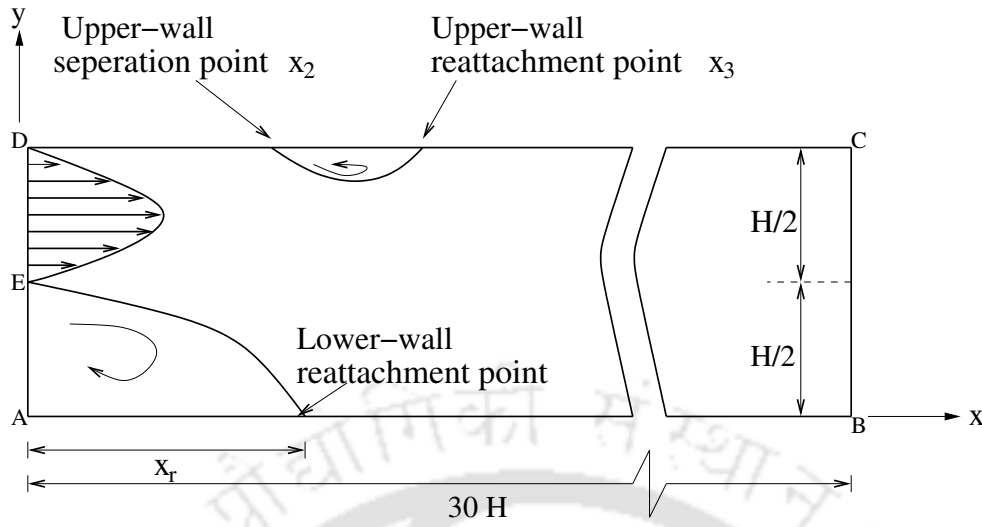


Fig. 3.20: Configuration of backward-facing step problem.

the flow to become fully developed. Thus at the outlet (BC) $\frac{\partial u}{\partial x} = 0, v = 0$ is prescribed. At the stationary walls (AB) and (CD) $u = v = 0$.

Fig. 3.21 shows the steady-state streamline patterns for $Re = 100$ to 800 . The streamlines illustrate the key features of the separated flow. At the enlargement, the flow velocity is suddenly reduced and as a consequence the pressure is increased. At point E where the step ends the flow separates causing the formation of a recirculation zone just downstream of the sudden enlargement. The reattachment length x_r increases with an increase in Re , which is clearly seen in Fig. 3.21. The secondary vortex at the upper wall can be clearly seen at $Re = 500$ and this behaviour is also reported in [58,101]. Fig. 3.22 depicts the velocity vector plots of the steady-state solution for $Re = 100$ to 800 . These velocity vector plots give a clear picture of the recirculation zones and the velocity profiles at different streamwise locations. After reattachment of the upper wall eddy, the flow slowly recovers towards a fully developed parabolic profile towards the exit.

Different researchers have debated whether it is possible to obtain a steady solution or not at $Re = 800$. This issue is discussed in detail by Gresho *et al.* [42] and they concluded that the flow is stable and computable at $Re = 800$. The same is the experience with us. In Fig. 3.23, we show the reattachment length as a function of the Reynolds number, and compare it with the experimental and predicted results of Armaly *et al.* [5] and the

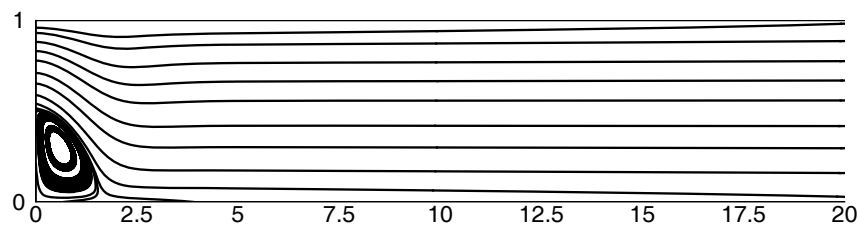
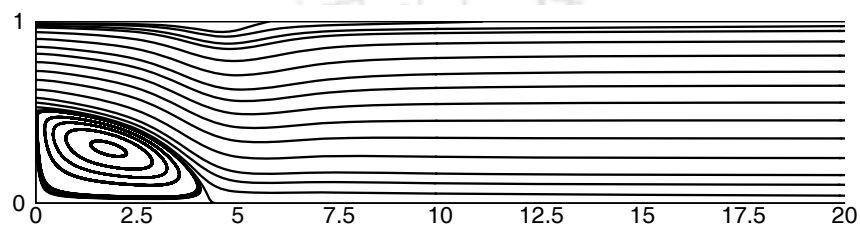
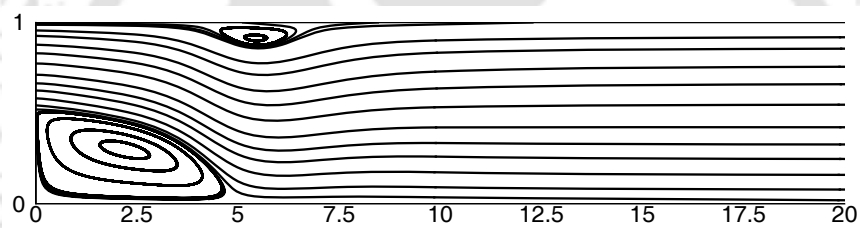
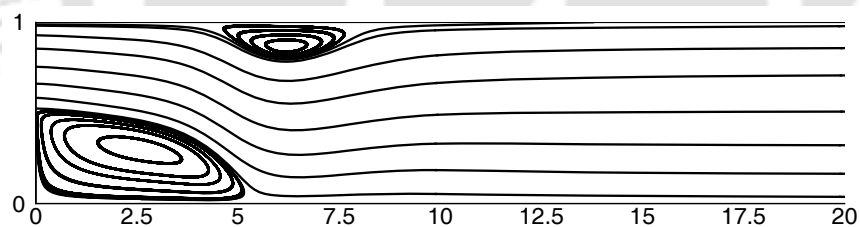
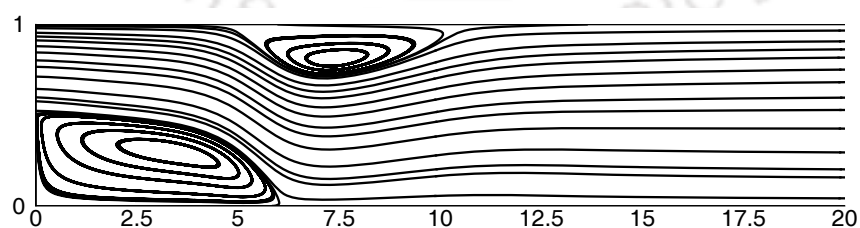
(a) $Re = 100$ (b) $Re = 400$ (c) $Re = 500$ (d) $Re = 600$ (e) $Re = 800$

Fig. 3.21: Steady-state streamlines for the backward-facing step flow at various Reynolds numbers.

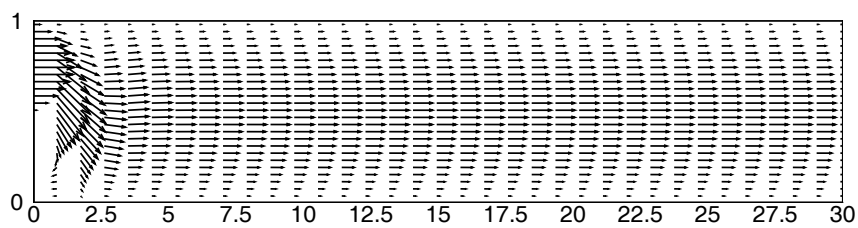
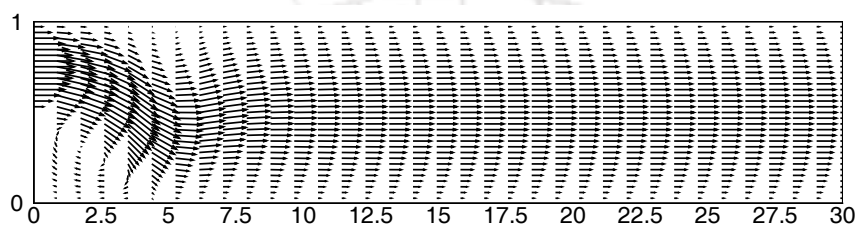
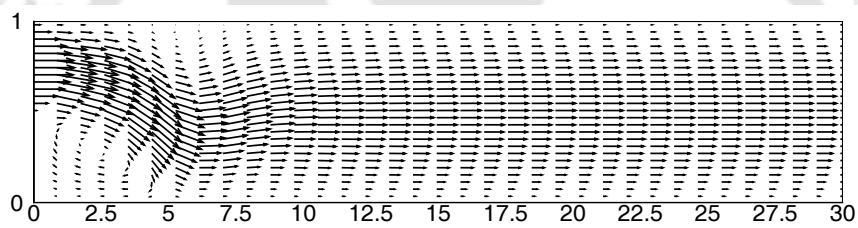
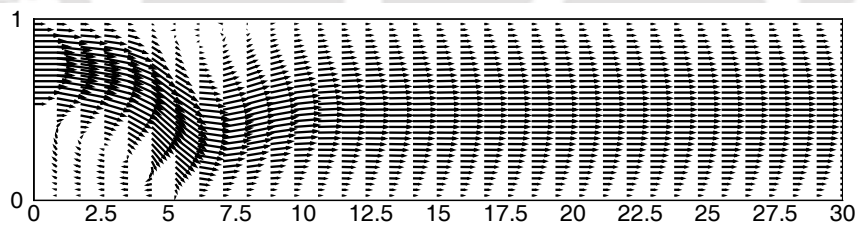
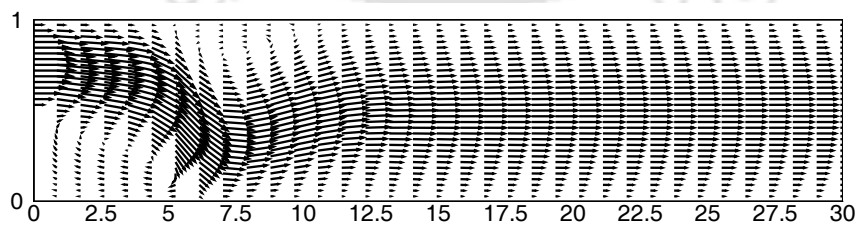
(a) $Re = 100$ (b) $Re = 400$ (c) $Re = 500$ (d) $Re = 600$ (e) $Re = 800$

Fig. 3.22: Steady-state velocity-vector plots for the backward-facing step flow at various Reynolds numbers.

numerical results of Kalita and Gupta [58]. As seen in Fig. 3.23, the present results are in excellent agreement with the results of [5, 58] for $100 \leq Re \leq 500$. The discrepancy between the numerical and experimental results of [5] for $Re > 600$ may arise owing to the three-dimensional effects [40, 58, 101, 120]. The streamwise velocity profiles for $Re = 800$ at $x = 7$ and 15 are plotted in Fig. 3.24 and compared with those reported in [40, 58]. It is clear that the present computations show an excellent match with these results. Fig. 3.25 depicts the pressure contours for various Reynolds numbers. Fig. 3.26 shows the pressure distribution on the upper and lower walls of the channel. As the axial distance increases beyond $x = 10$, the upper- and lower-wall pressure distributions approach each other and finally they collapse into a single line as the flow becomes fully developed. This behaviour (Fig. 3.26) is consistent with that obtained by Gartling [40]. We have seen that the present unsteady code provides steady results even at $Re = 1000$ and produces steady-state solutions, which were also obtained by [25, 92]. Fig. 3.27 shows the streamlines, velocity vectors and pressure plots at $Re = 1000$.

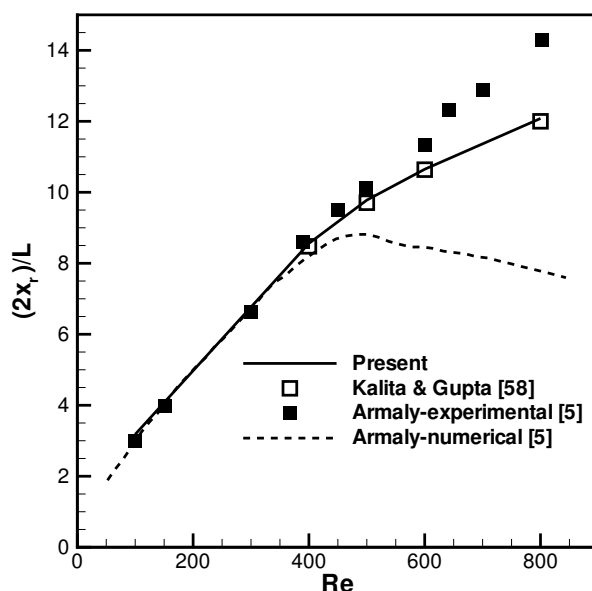


Fig. 3.23: The reattachment length as a function of Reynolds number.

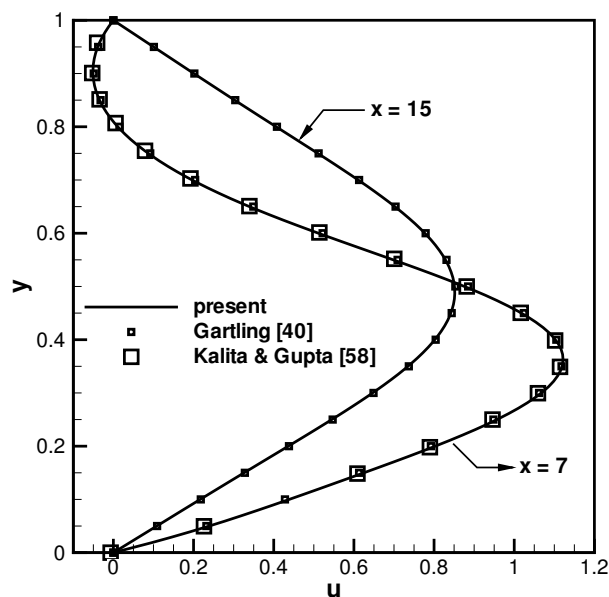


Fig. 3.24: Comparison of u profile for $Re = 800$ at $x = 7$ and $x = 15$ with the result of Gartling [40] and Kalita and Gupta [58].

3.5.4 Flow Past a Square Prism

The viscous flow past a bluff body and the resulting separated region behind it, has been the focus of numerous experimental and numerical investigations. The presence of wall confinement is one of the most common examples in practical flow situations (e.g., the flow situation in wind tunnels, flow around tall buildings and bridges, etc.) and therefore the study of effect of wall confinement on flow field characteristics is of crucial importance. This section of the present chapter is devoted to the computation of flow past a square prism in a channel. Studies on problems of wake development and vortex shedding behind a two-dimensional rectangular prism in confined and un-confined flows have been investigated numerically and experimentally by Davis *et al.* [27, 28]. Davis and Moore [27] reported numerical results generated by their SIMPLEC code. Davis *et al.* [28] examined confined flow past a rectangular cylinder both numerically and experimentally. Franke *et al.* [38] examined numerical results for Reynolds number less than or equal to 300 using a SIMPLEC code. Sohankar *et al.* [100] examined quite different numerical methods for two-dimensional

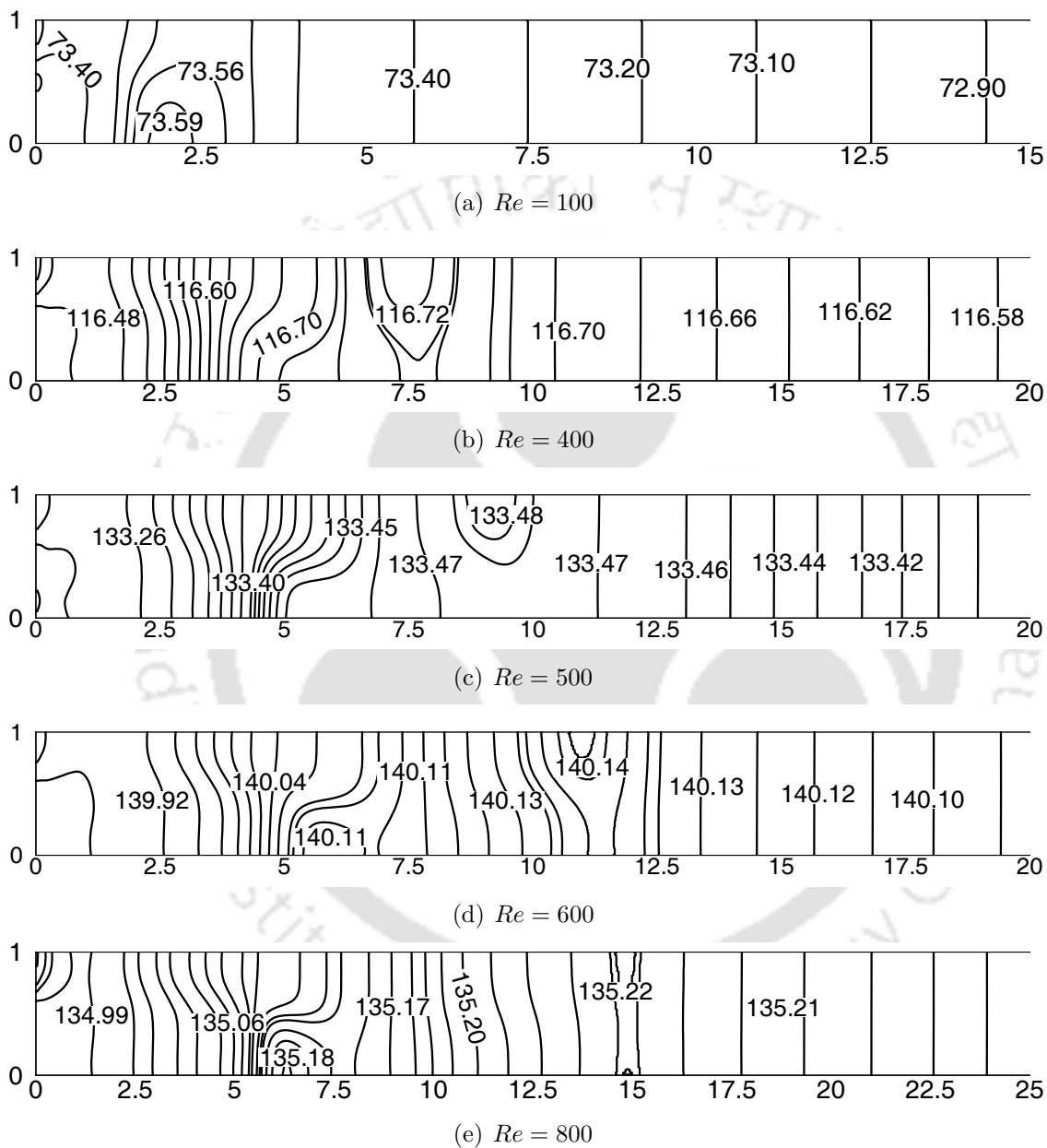


Fig. 3.25: Steady-state pressure contours for the backward-facing step flow at various Reynolds number.

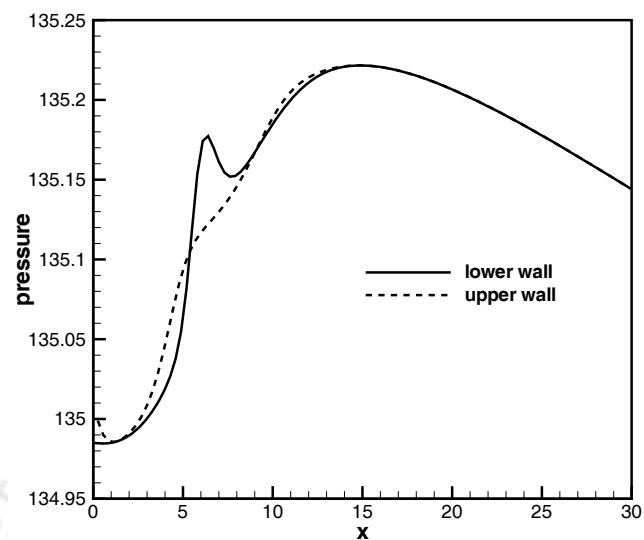
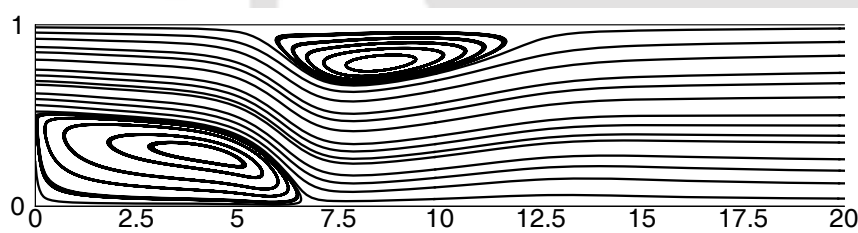
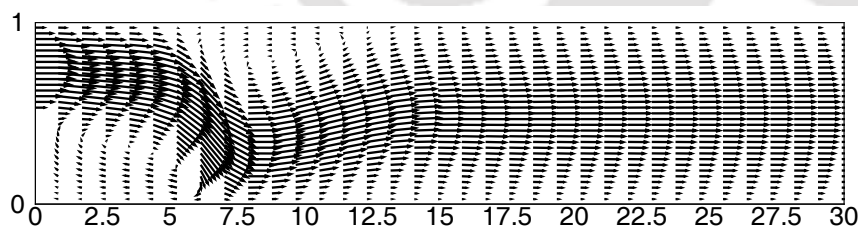


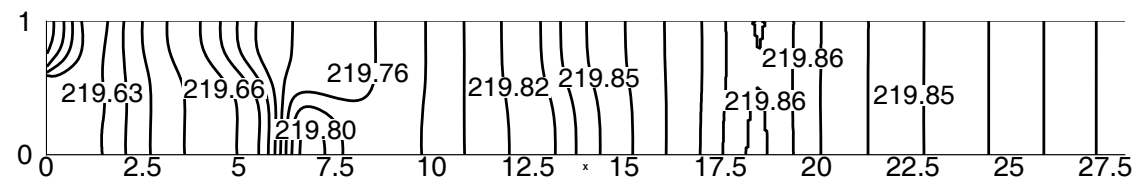
Fig. 3.26: Pressure distribution along upper and lower channel walls at $Re = 800$.



(a) $Re = 1000$



(b) $Re = 1000$



(c) $Re = 1000$

Fig. 3.27: At $Re = 1000$, steady-state: (a) streamline pattern (b) velocity-vector plot (c) pressure contours.

and three-dimensional unsteady flows around a square prism and observed that a transition from 2D to 3D shedding flow may occur for Reynolds number between 150 and 200.

Flow past a rectangular cylinder in the presence of a fixed wall has numerically been studied by Hwang and Yao [50] and Roy and Bandyopadhyay [94]. Other numerical studies examined the vortex shedding on the square prism [13, 61, 80, 96, 108]. From the literature, it is observed that when bounding walls are present in the flow, the behaviour of vortex shedding behind the body is affected significantly. It should be stated clearly that the objective of the present work is to demonstrate the reliability of the present code to capture the vortex shedding. The domain of the flow is multiply connected and the nature of the flow itself is very complex. At sufficiently high Reynolds numbers owing to the wake instability mechanisms the vortical region downstream of the body breaks into the phenomenon of vortex shedding characterized by an unsteady periodic flow situation in which the separated vortices are shed alternately from the upper and lower sides of the body.

The unsteady behaviour of the flow that evolves with time for Reynolds numbers beyond a critical value makes this problem quite challenging and interesting. Thus, simulation of this flow configuration is a good test case to check the robustness and reliability of the present code. The flow configuration is similar to those of [13, 58] as shown in Fig. 3.28(a). It has a prism with square cross-section with width D that is mounted centrally inside a plane channel of height H with blockage ratio $B = \frac{D}{H} = \frac{1}{8}$. The channel length L is fixed at $\frac{L}{D} = 50$ to reduce the influence of inflow and outflow boundary conditions. An inflow length of $l = \frac{L}{4}$ has been chosen. At the inlet, a parabolic velocity profile is introduced while at the outlet, convective boundary conditions given by

$$\frac{\partial \phi}{\partial t} + u_{conv} \frac{\partial \phi}{\partial x} = 0 \quad (3.24)$$

has been used with ϕ standing for u and v . This condition ensures that vortices can approach and pass the outflow boundary without significant disturbances or reflections into the inner domain. Here u_{conv} is set equal to the maximum u -velocity at the inlet. The no-slip boundary

conditions are applied on the surface of the square prism and on both the upper and lower walls.

Due to the specific geometry in the present study, nonuniform Cartesian grids are used. This has the advantage that grid points can be clustered in regions of large gradients, i.e., in the vicinity of the square prism and coarser grids can be used in regions with small gradients (see Fig. 3.28(b)). A 1024×128 grid is used in the present study. This flow configuration

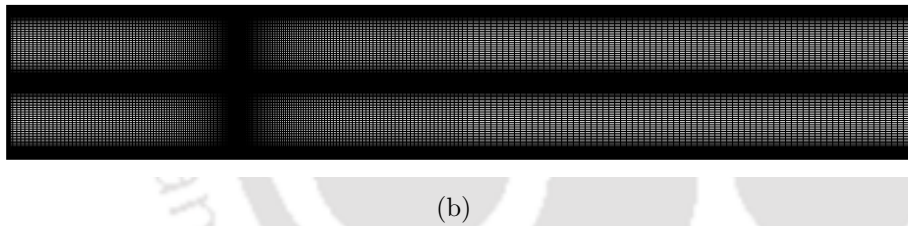
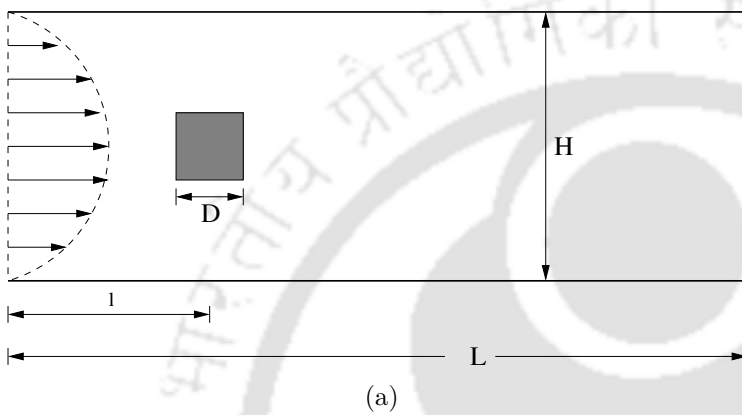


Fig. 3.28: (a) Configuration of the flow past a square prism (b) Grid arrangement for flow past a square prism.

has been solved for the Reynolds numbers 1, 30 and 100 where Re is based on the prism diameter and the maximum flow velocity u_{max} of the parabolic inflow profile. Fig. 3.29 shows the computational streamlines at three different Reynolds numbers $Re = 1, 30$ and 100 each characterising a different flow regime. At low $Re = 1$, the creeping steady flow past the square prism persists without separation (Fig. 3.29(a)). Relative magnitude of viscous forces decreases with an increasing Re until at a certain value a separation of the laminar boundary layer occurs. As Re increases separation at the trailing edges of the sharp-edged body can be observed. At $Re = 30$, the wake comprises a steady recirculation region of two symmetrically placed vortices on each side of the channel centreline as shown in Fig. 3.29(b). Due to the

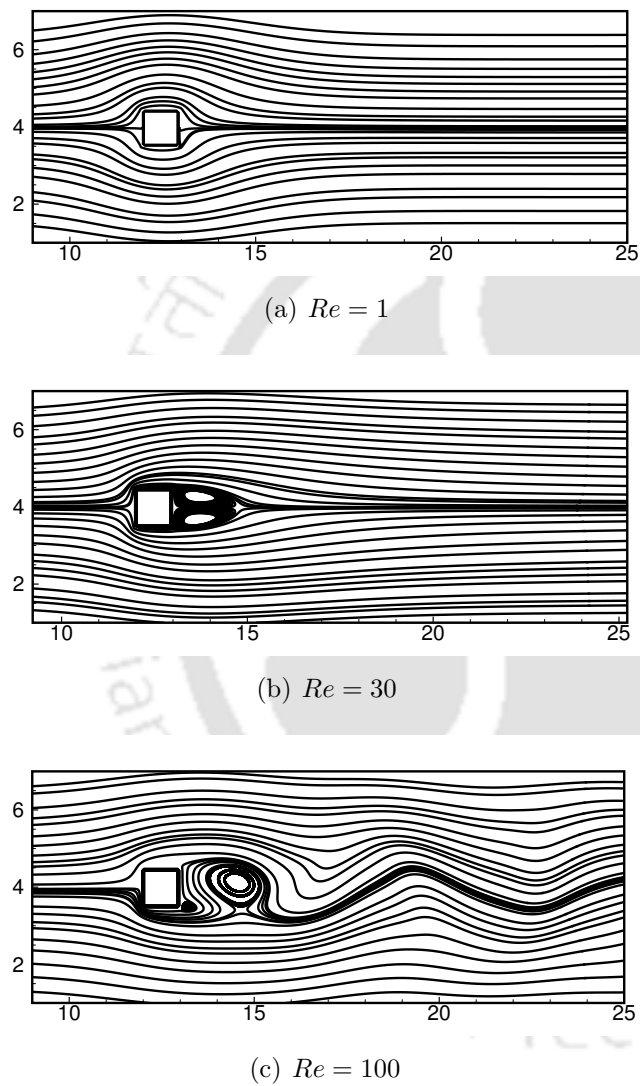


Fig. 3.29: Streamline patterns for flows past a square prism at different Reynolds numbers.

sharp corners, the separation point is fixed at the trailing edge and the flow is attached at the side walls. The corresponding vorticity contours for $Re = 1$ and 30 are depicted in Fig. 3.30(a-b). Literature states that initially the size of the recirculation region increases with an increase in Re and when a critical Reynolds number is exceeded, the well known von Kármán vortex street with periodic vortex shedding from the prism can be detected in the wake. Based on experimental investigations, Okajima [84] found periodic vortex motion at $Re \approx 70$ leading to an upper limit of $Re_{crit} \leq 70$. A smaller value ($Re_{crit} = 54$) was determined by Kelkar and Patankar [61] based on a stability analysis of the edges of the prism. Present computations at $Re = 100$ (Fig. 3.30(c)) show that the free shear layers roll up and form eddies as shown in Fig. 3.30(c). This phenomenon is popularly known

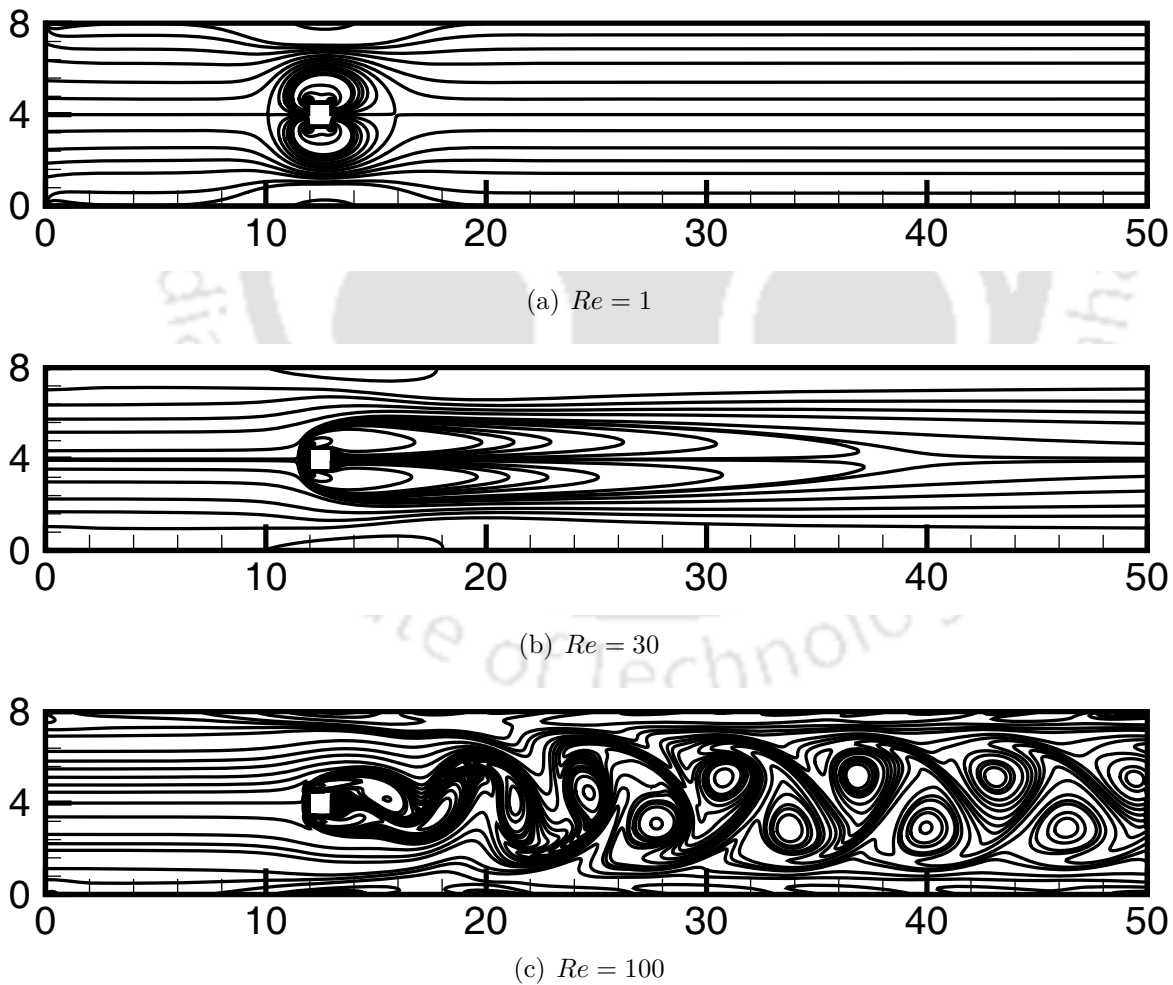


Fig. 3.30: Vorticity contours for flows past a square prism at different Reynolds numbers.

as the von Kármán vortex shedding and the corresponding streamline pattern is shown in Fig. 3.29(c). The temporal evolution of the streamline patterns over one complete period is shown in Fig. 3.31. The streamlines are wavy and sinuous on the leeward side of the square prism. However, the upstream depicts a potential-flow-like pattern. Two eddies are shed within each period from the aft of the square prism. These eddies are formed behind the prism and are washed away into the wake region. The temporal evolution of the vorticity contours over one complete vortex shedding cycle is depicted in Fig. 3.32. The vorticity contours reveal several additional features which could not be directly perceived from the streamlines. The staggered nature of the Kármán shedding is clearly seen from these plots. The eddies are alternately of positive and negative vorticity. This is indeed reflected in the form of crests and troughs in the sinuous wake of the streamlines.

The two most important characteristic quantities of flow around a prism are the drag coefficient (C_D) and the lift coefficient (C_L). The coefficients C_D and C_L are defined as $F_D/\frac{1}{2}\rho u_{max}^2 D$ and $F_L/\frac{1}{2}\rho u_{max}^2 D$, where F_D and F_L denote the drag and lift force on the square prism, respectively. The drag force (F_D) is calculated by the expression $\sum_1 p\delta y - \sum_2 p\delta y$, where the suffixes 1 and 2 denotes the forward and rear sides of the prism. The lift force (F_L) is calculated by the expression $\sum_3 p\delta x - \sum_4 p\delta x$, where the suffixes 3 and 4 denotes the top and bottom sides of the prism. Fig. 3.33(a) presents the time history of the drag coefficient (C_D) and lift coefficient (C_L) after the flow has attained periodicity. The periodic eddy shedding is reflected in the fluctuating drag and lift coefficient history. It is reported that the C_D of the square prism has a higher value than the C_D of a circular cylinder [114]. Fig. 3.33(a) shows that for each time period C_D has two crests and two troughs of unequal amplitudes, which are a consequence of the periodic vortex shedding from the top and bottom surfaces. The same figure shows that C_L for each time period has just one trough and one crest; this is because C_L is not influenced by the pressure distribution on the right face, which is strongly influenced by the vortex shedding at the top and bottom surfaces. The same trend has been observed in flow past a circular cylinder [18]. The value of the lift force fluctuation is directly connected to the formation and shedding of the eddy and, therefore,

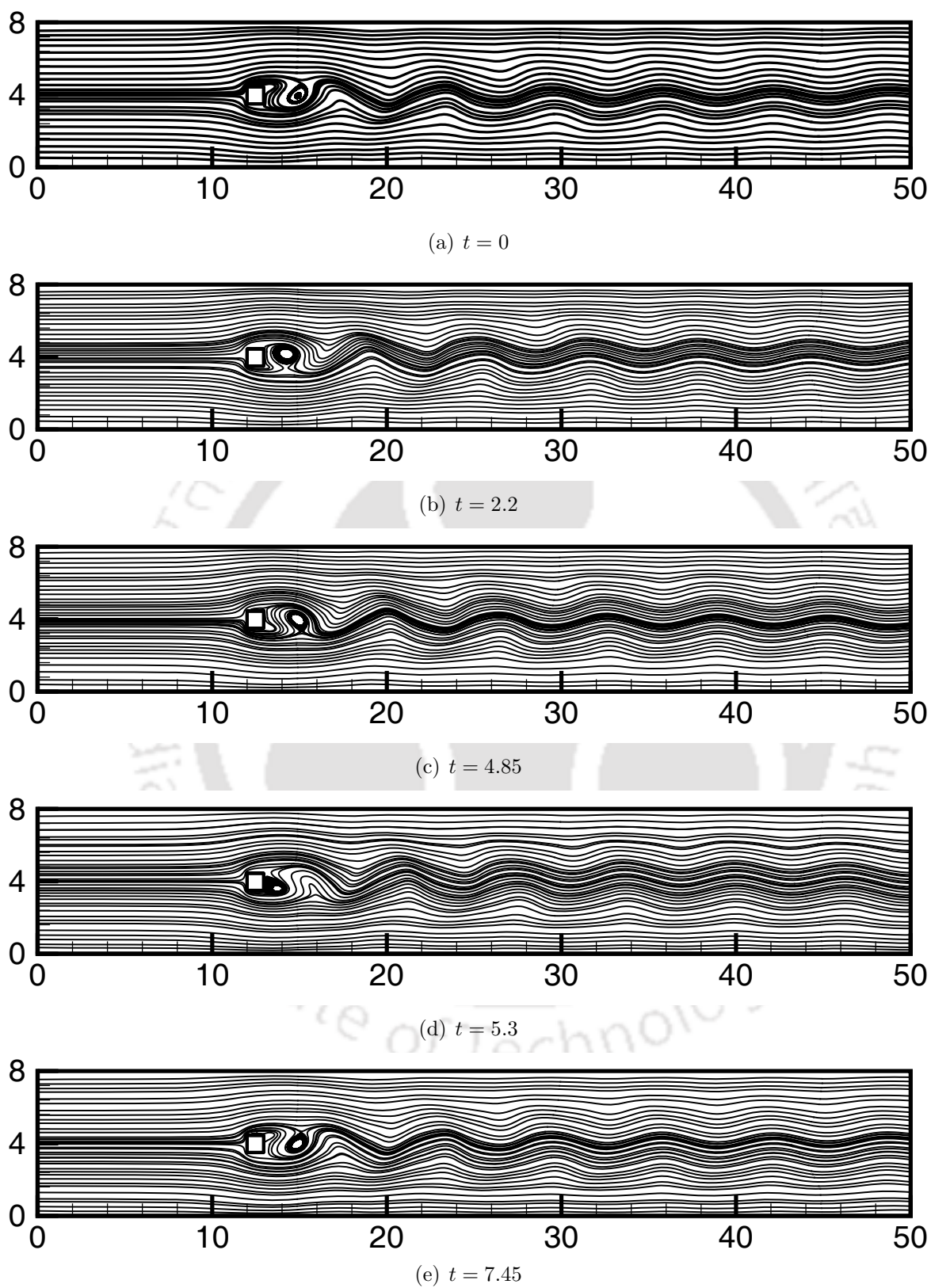


Fig. 3.31: Temporal evolution of streamline patterns for the flow over a square prism at $Re = 100$ over one period.

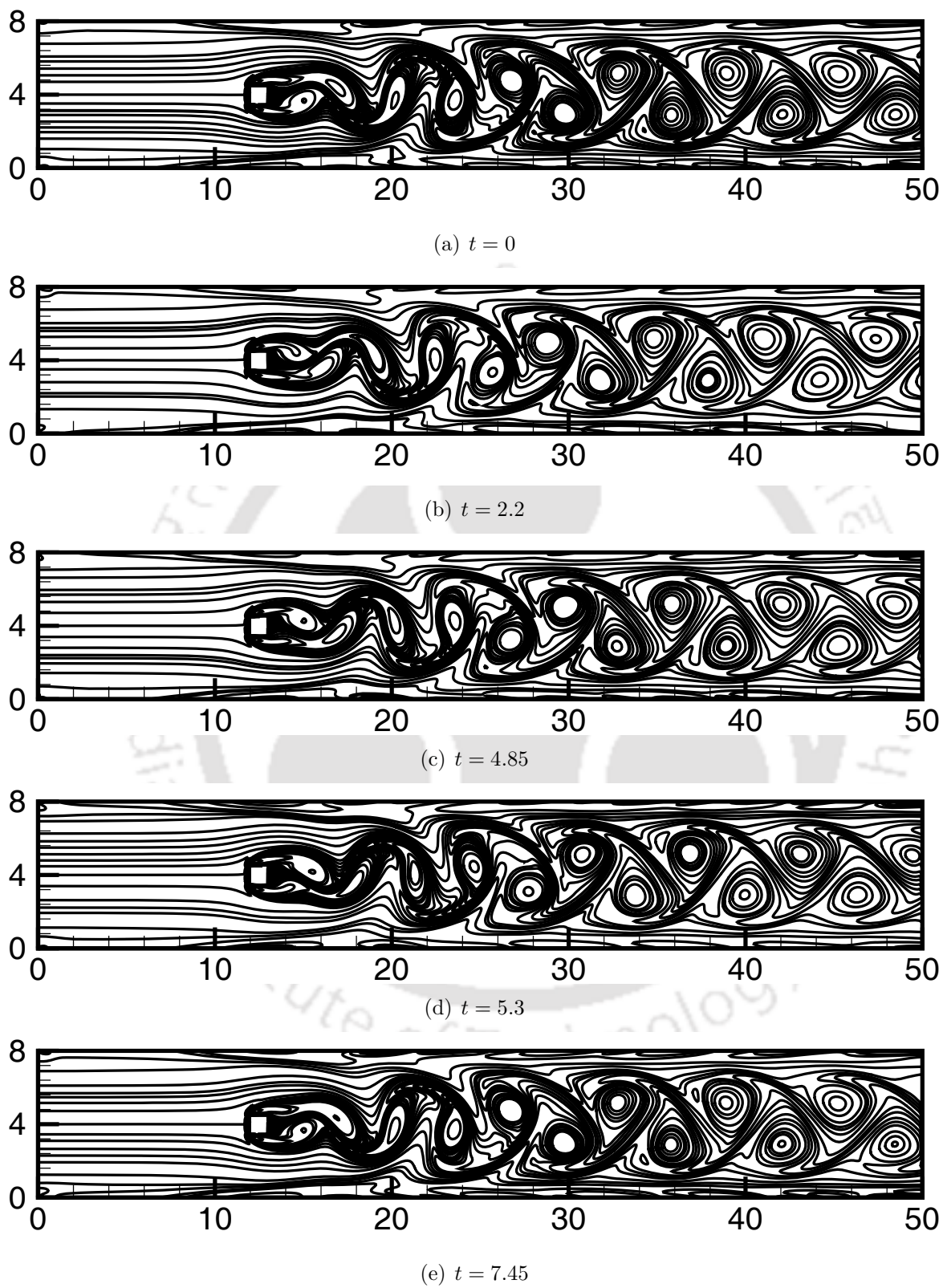


Fig. 3.32: Temporal evolution of vorticity contours for the flow over a square prism at $Re = 100$ over one complete vortex shedding cycle.

its value varies between a positive maximum and a negative maximum of equal magnitude.

A power spectrum plotted in Fig. 3.33(b) for C_L using the Fourier analysis confirms that the solution is periodic with one dominant harmonic and its corresponding frequency is 0.137. After the flow reaches a stable periodic state, a phase diagram (Fig. 3.33(c)) is drawn between u and v at the monitoring point $(53/4, 7/2)$. These plots indicate a perfect periodic pattern for the solutions obtained. Another important quantity considered in the present analysis is the Strouhal number (St), computed from the prism width D , the measured frequency of the vortex shedding f and the maximum velocity u_{max} at the inflow plane through the relation

$$St = \frac{fD}{u_{max}} \quad (3.25)$$

where f , the frequency of vortex shedding, is determined by a spectral analysis of the time series of the lift coefficient C_L (Fig. 3.33(b)). The mean drag coefficient (C_{Dmean}) is calculated by time-averaging C_D for integral number of cycles using the Simpson's one-third rule. The computed C_{Dmean} and Strouhal number are compared with earlier results reported in the literature and are shown in Table 3.4. The present results are in good agreement with earlier investigations.

Table 3.4: Comparison of C_{Dmean} and Strouhal number

	Breuer [13]	De and Dalal [29]	present
C_{Dmean}	1.37	1.31	1.318
Strouhal number	0.139	0.142	0.137

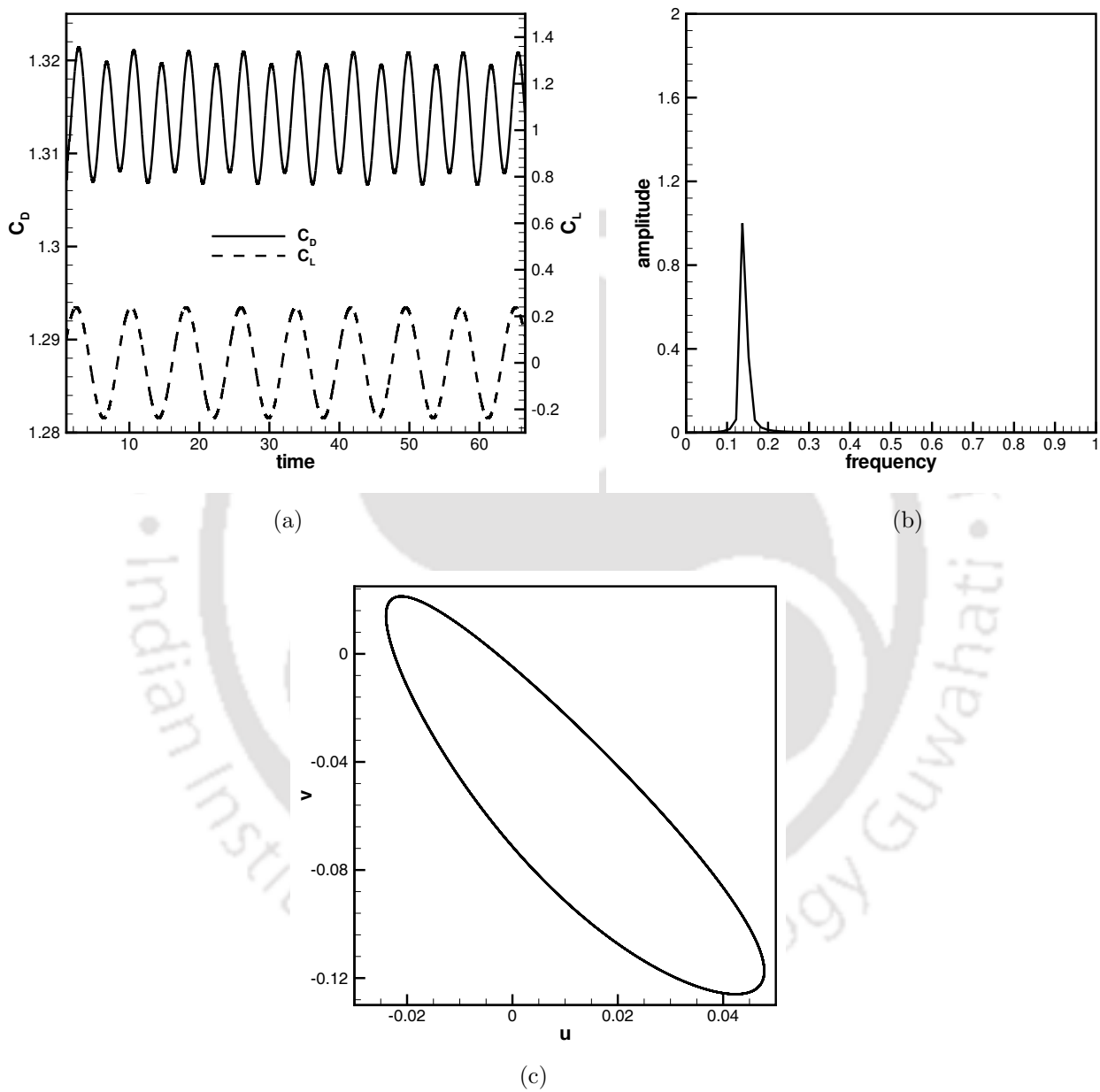


Fig. 3.33: (a) Variation of drag and lift coefficients, (b) power spectrum and (c) phase diagram at monitoring point $(53/4, 14/4)$ for $Re = 100$.

3.6 Conclusions

The main aim of the present chapter is the development of a 2D transient Navier-Stokes solver and test its capability and efficiency in known and unknown situations. In the efficiency front, to achieve grid economy graded Cartesian meshes coupled with grid-transformation are used and to enhance the time-wise efficiency, multigrid method is used to solve the cumbersome pressure-Poisson equation that needs to be solved at every time step in a transient solver. To test the capability of the code it is first applied to capture the transient and steady-state flow features in the configuration of a standard 2D single-sided lid-driven cavity. Care has been taken to produce results independent of the space-grid and time step. As already mentioned the discretizations used in the present code achieve a second order temporal and at least a third order spatial accuracy. Consequently the results that are produced on relatively coarser grids (because of graded mesh) with good time-wise efficiency (because of multigrid) agree quite well with highly reliable existing results. The capability of the code is further tested by capturing periodic flows in the cavity for $Re = 8200$ and 10000 at which some of the earlier works showed time-stationary results.

After having thus gained an insight into various aspects of the code including its accuracy and capability, it is then used to capture the transient flow features that evolved with time in a 2D two-sided lid-driven cavity. This flow configuration is fraught with many interesting issues in that it involves development of a pair of off-corner vortices and a ‘free shear’ layer. To obtain better flow resolution the grids are clustered adequately at appropriate locations. Given the accuracy and capability of the code there is no doubt that the transient results for this untested flow configuration are highly reliable and accurate. It may be mentioned that steady results for some Reynolds numbers for this geometry exist but as they are produced by solving steady equations it was not clear whether the flow was really steady at, for example, $Re = 2000$. This issue was resolved by the present code by showing that the flow at $Re = 2000$ for the two-sided lid-driven cavity is indeed steady.

The third problem, the flow over a backward facing step, is solved in the Reynolds number

range of 100 to 1000. The present results are in excellent agreement with earlier published results. It has been seen from the present unsteady code that there exists a steady solution at $Re = 1000$. Finally the robustness of the present code is highlighted by capturing the van Kármán street for flow over a square prism confined in a channel. We examine some physical quantities such as drag coefficient, lift coefficient and the Strouhal number and also the vortex shedding formation which provides an excellent means of visualizing the von Kármán vortex street. The results obtained are in good agreement with those reported in the literature.

Another issue addressed in the present chapter is the comparative performance of the multigrid algorithm on uniform and nonuniform grids of equal sizes. Though acceleration produced by the multigrid on a uniform grid is seen to be somewhat better than that for a nonuniform grid, this fact is not significant as in many situations it is possible to use a nonuniform grid involving much smaller number of grid nodes to produce the same quality of solution as on a uniform grid. The results and the subsequent discussions suggest that the discretizations and solution algorithm used and grids chosen render the present computations accurate and efficient and the developed code can be considered a good numerical tool.

Chapter 4

A Multigrid-Accelerated Three-Dimensional Transient-Flow Code and its Application to a New Test Problem

4.1 Introduction

A multigrid-assisted solver for the three-dimensional time-dependent incompressible Navier-Stokes equations on graded Cartesian meshes is developed in the present chapter. Three-dimensional cavity flows, which are more complex, realistic and global have been an interesting research topic during the last three decades. Compared with 2D lid-driven cavity, flows in a cubic cavity are richer in physical phenomena. Numerical results for 3D driven cavity flows are relatively scarce. A detailed review of the flow physics in the driven cavity was provided by Shankar and Deshpande [97]. Koseff and Street [65] showed the discrepancy between the two-dimensional solution of the lid-driven cavity flow and the experimentally obtained flow. Koseff *et al.* [66] showed that the flow becomes three-dimensional at sufficiently high Reynolds number due to the Taylor-Gortler-like (TGL) vortices. Ku *et al.* [67] employed a

Chebyshev pseudospectral method for unsteady flows based on primitive variables by using a time-splitting technique with a grid of $31 \times 31 \times 16$ points for $100 \leq Re \leq 1000$. Mei *et al.* [78] employed Lattice Boltzmann method for 3D cubical cavity flow with curved boundary treatment in the range of Reynolds number from $Re = 100$ to 2000. Shu *et al.* [31] simulated 3D cubical cavity flow by Taylor series expansion and least squares-based Lattice Boltzmann method using the fifteen-velocity model ($D3Q15$) at three different Reynolds numbers $Re = 100, 400$ and 1000 and they observed a weak pair of TGL at $Re = 1000$ at $x = 0.5$, x being the direction of lid movement. Most earlier 3D simulations in a cubic cavity emphasised on steady results. There are only a few works that deal with transient solutions of the 3D lid-driven cavity flows. Tang *et al.* [105] and Guermond *et al.* [43] produced transient results for the cubic lid-driven cavity at various Reynolds numbers. A discussion on the formation of a pair of TGL vortices at $Re = 1000$ may be found in these works. Albensoeder and Kuhlmann [2] provided a benchmark data for three-dimensional lid-driven cavity at $Re = 1000$ using Chebyshev collocation method. Recently Rahmati and Ashrafizaadeh [90] proposed a generalised lattice Boltzmann method and solved 3D lid driven cavity for various Reynolds number and observed the formation of a couple of TGL vortices at $Re = 1000$. Three-dimensional cavities with more than one moving walls have not been extensively studied in the literature. Recently, Beya and Lili [10] numerically solved the three-dimensional two-sided non-facing lid-driven cavity flows for various Reynolds numbers using the projection method. They employed full multigrid method for solving the Poisson equation on a staggered grid system using the finite volume method. However, no work on the study of three-dimensional two-sided (parallel wall motion) lid-driven cavity flow has been reported in the literature.

In the present chapter a finite-difference discretization of the governing equations is carried out on a staggered grid [47,85,86]. The convective terms in the Navier-Stokes equations are discretized here using a third-order accurate upwind scheme [60] and the viscous terms are discretized centrally to fourth-order accuracy. As the time accuracy of the fractional-step method used in the present work is also of second-order, the transient results produced here

are reliable. This aspect of the code is examined by first applying it to compute the transient and steady (obtained by marching sufficiently in time) solutions for a single-sided lid-driven cubic cavity. Close agreement with existing results [2,67] proves the accuracy of the present computations. The validated code is then applied to a new test case, namely, the three-dimensional two-sided (parallel wall motion) lid-driven cavity problem that involves gradual development of a free shear layer with associated vortices. Although there are no existing data with which these results could be compared, the computations are lent prestige by the fact that the accuracy of the code has already been substantiated and that it has a good spatio-temporal accuracy. Also care have been taken to produce time-step-independent and grid-independent results. Use of high-precision arithmetic to offset accumulation of round-off errors in the transient computations lends added credibility to the results. Thus it can be summarised that the main purpose of the present work is to develop a useful and efficient numerical tool for computing incompressible transient-viscous flows using multigrid method on graded Cartesian meshes and also to record our experiences on the application of multigrid to nonuniform meshes through the two problems studied, that include the one-sided and two-sided three-dimensional lid-driven cavity problems. Other aims of this work are to carry out a comparison of transient and steady-state results between 2D and 3D geometries and apply the present code to produce accurate transient and steady results for a new test case in the shape of the two-sided 3D lid-driven cavity problem.

4.2 Governing Equations

The 3D unsteady incompressible N-S equations in the primitive variable formulation and non-dimensional form can be written as

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \quad (4.1)$$

$$\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} = -\frac{\partial p}{\partial x} + \frac{1}{Re} \left[\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right] \quad (4.2)$$

$$\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + w \frac{\partial v}{\partial z} = -\frac{\partial p}{\partial y} + \frac{1}{Re} \left[\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right] \quad (4.3)$$

$$\frac{\partial w}{\partial t} + u \frac{\partial w}{\partial x} + v \frac{\partial w}{\partial y} + w \frac{\partial w}{\partial z} = -\frac{\partial p}{\partial z} + \frac{1}{Re} \left[\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2} \right] \quad (4.4)$$

Here u , v and w denote the velocities along x , y and z directions, t the time and p the pressure, respectively. The pressure is treated explicitly by solving a pressure-Poisson equation, since the solution of Eqs. (4.1) - (4.4) does not possess an independent equation for pressure. The solution of governing Eqs. (4.1) - (4.4) on graded Cartesian meshes using finite difference method makes it necessary to transform the equations from the physical to the computational plane.

4.2.1 Governing Equations in Transformed Plane

The governing equations (4.1) - (4.4) from the physical plane are transformed to the computational plane without changing the mathematical characteristics of the equations. The details of the transformation for 2D equations is given in Appendix A. The same transformation procedure can be expanded for 3D equations. Using these transformations and observing that for a graded Cartesian mesh $\xi = \xi(x)$, $\eta = \eta(y)$ and $\zeta = \zeta(z)$, the governing equations in the computational plane can now be written

$$\xi_x \frac{\partial u}{\partial \xi} + \eta_y \frac{\partial v}{\partial \eta} + \zeta_z \frac{\partial w}{\partial \zeta} = 0 \quad (4.5)$$

$$\begin{aligned} \frac{\partial u}{\partial t} + (u\xi_x) \frac{\partial u}{\partial \xi} + (v\eta_y) \frac{\partial u}{\partial \eta} + (w\zeta_z) \frac{\partial u}{\partial \zeta} = & -\xi_x \frac{\partial p}{\partial \xi} + \frac{1}{Re} \left[\xi_{xx} \frac{\partial u}{\partial \xi} + \xi_x^2 \frac{\partial^2 u}{\partial \xi^2} \right] + \\ & \frac{1}{Re} \left[\eta_{yy} \frac{\partial u}{\partial \eta} + \eta_y^2 \frac{\partial^2 u}{\partial \eta^2} \right] + \frac{1}{Re} \left[\zeta_{zz} \frac{\partial u}{\partial \zeta} + \zeta_z^2 \frac{\partial^2 u}{\partial \zeta^2} \right] \end{aligned} \quad (4.6)$$

$$\begin{aligned} \frac{\partial v}{\partial t} + (u\xi_x) \frac{\partial v}{\partial \xi} + (v\eta_y) \frac{\partial v}{\partial \eta} + (w\zeta_z) \frac{\partial v}{\partial \zeta} = -\eta_y \frac{\partial p}{\partial \eta} + \frac{1}{Re} \left[\xi_{xx} \frac{\partial v}{\partial \xi} + \xi_x^2 \frac{\partial^2 v}{\partial \xi^2} \right] + \\ \frac{1}{Re} \left[\eta_{yy} \frac{\partial v}{\partial \eta} + \eta_y^2 \frac{\partial^2 v}{\partial \eta^2} \right] + \frac{1}{Re} \left[\zeta_{zz} \frac{\partial v}{\partial \zeta} + \zeta_z^2 \frac{\partial^2 v}{\partial \zeta^2} \right] \end{aligned} \quad (4.7)$$

$$\begin{aligned} \frac{\partial w}{\partial t} + (u\xi_x) \frac{\partial w}{\partial \xi} + (v\eta_y) \frac{\partial w}{\partial \eta} + (w\zeta_z) \frac{\partial w}{\partial \zeta} = -\zeta_z \frac{\partial p}{\partial \zeta} + \frac{1}{Re} \left[\xi_{xx} \frac{\partial w}{\partial \xi} + \xi_x^2 \frac{\partial^2 w}{\partial \xi^2} \right] + \\ \frac{1}{Re} \left[\eta_{yy} \frac{\partial w}{\partial \eta} + \eta_y^2 \frac{\partial^2 w}{\partial \eta^2} \right] + \frac{1}{Re} \left[\zeta_{zz} \frac{\partial w}{\partial \zeta} + \zeta_z^2 \frac{\partial^2 w}{\partial \zeta^2} \right] \end{aligned} \quad (4.8)$$

The pressure-Poisson equation in the transformed plane, which is solved using the multigrid method at each time step to satisfy the divergence-free condition is given by

$$\begin{aligned} \xi_{xx} \frac{\partial p}{\partial \xi} + \xi_x^2 \frac{\partial^2 p}{\partial \xi^2} + \eta_{yy} \frac{\partial p}{\partial \eta} + \eta_y^2 \frac{\partial^2 p}{\partial \eta^2} + \zeta_{xx} \frac{\partial p}{\partial \zeta} + \zeta_x^2 \frac{\partial^2 p}{\partial \zeta^2} = \\ \frac{1}{\Delta t} \left(\xi_x \frac{\partial u^*}{\partial \xi} + \eta_y \frac{\partial v^*}{\partial \eta} + \zeta_z \frac{\partial w^*}{\partial \zeta} \right) \end{aligned} \quad (4.9)$$

4.3 Computations, Results and Discussions

The code based on the fractional-step method (explained in Section 3.3) for the time integration in 3D N-S equations. The numerical schemes used in the present work is discussed in Section 3.4. The performance of a 2D code is already evaluated in regard to time-wise efficiency and accuracy in Chapter 3. The results show that the 2D code is capable of capturing transient results accurately and the methods used to develop the code seems to be sound. The present code for 3D is based on the same strategy and therefore expected to accurately compute various 3D transient flows.

4.3.1 Transient Flow in a Single-Sided Lid-Driven Cubic Cavity

Initially the transient 3D incompressible laminar lid-driven flow in a cubic cavity is computed through the developed code. The flow pattern inside the cubic cavity includes several vortices whose characteristics depend on the Reynolds number. The geometry and the nonuniform grid arrangement for the 3D lid-driven cavity is shown in Fig. 4.1 where the top wall is moving

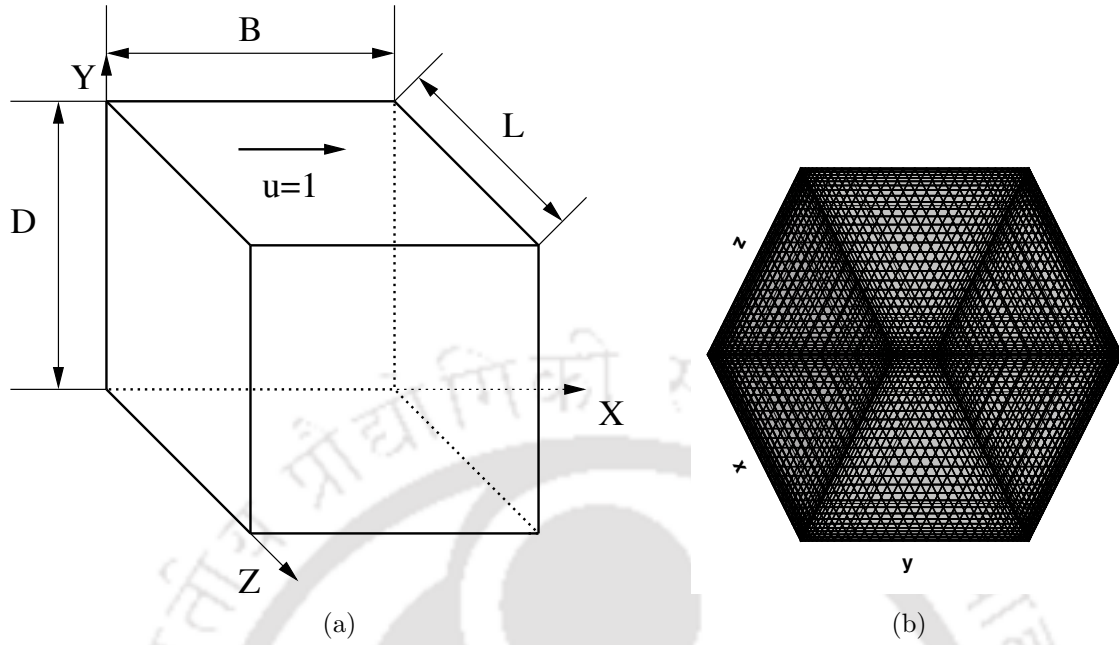


Fig. 4.1: (a) Schematic of lid-driven cubic cavity (b) nonuniform grid arrangement for lid-driven cubic cavity.

in the x -direction and the remaining walls are stationary. Here B , D and L denote the width, depth and length of the cavity respectively. The span-wise aspect ratio ($SAR=L/B$) of 1:1 is considered. The physical boundary conditions of the geometry are specified for $t > 0$, as

$$\text{At } x = 0 \text{ and } 1, u = v = w = 0$$

$$\text{At } z = 0 \text{ and } 1, u = v = w = 0$$

$$\text{At } y = 0, u = v = w = 0$$

$$\text{At } y = 1, u = 1, v = w = 0$$

The moving wall generates momentum which diffuses inside the cavity and this diffusion is the driving mechanism of the flow. At high Reynolds numbers, several secondary and tertiary vortices begin to appear, whose characteristics depend on Re . Because of the presence of large gradients near the walls, the grids are clustered in the vicinity of walls to capture the smaller spatial scales in these regions. The stretching function [57] given below is used to

Table 4.1: Performance of multigrid on a $65 \times 65 \times 65$ grid at $Re = 1000$ for a single-sided lid-driven cubic cavity

time-steps ($\Delta t = 0.001$)	CPU time (minutes)		Speed-up
	Single grid	MG-4 level	
100	31.05	2.36	11.96
500	140.42	8	17.55

cluster grid points near the wall.

$$\begin{aligned} x &= \xi - \frac{\lambda}{2\pi} \sin(2\pi\xi) \\ y &= \eta - \frac{\lambda}{2\pi} \sin(2\pi\eta) \\ z &= \zeta - \frac{\lambda}{2\pi} \sin(2\pi\zeta) \end{aligned} \quad (4.10)$$

where λ denotes the stretching parameter with $0 < \lambda < 1$. It may be noted that $\lambda = 0$ results in a uniform grid and $\lambda = 0.9$ produces dense clustering. In the present case λ is taken as 0.6.

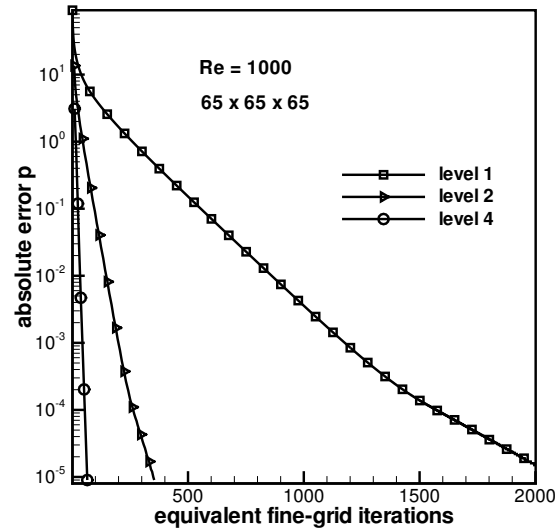


Fig. 4.2: Pressure error history at $t = 0.1$ for the Poisson equation at $Re = 1000$.

In the present study a 4-level V-cycle multigrid shown in Fig. 3.3 has been used in all the computations to solve the pressure-Poisson equation and the number of sweeps used in various grid-levels is given inside the corresponding circles. At $Re = 1000$ on a $65 \times 65 \times 65$ grid, it is observed that using four level is good enough and a further increase of level produces no time-wise gain. To gain an insight into the convergence behaviour after 100 time steps, at $t = 0.1$ a plot between the pressure error and number of iterations required to solve the pressure-Poisson equation is shown in Fig. 4.2. The convergence histories obtained with the single-grid and with sequences of 2- and 4- levels are shown in Fig. 4.2. Expectedly the convergence curves relating to multigrid are considerably steeper than those relating to the single grid. The speed-up achieved by the 4-level multigrid is represented in Table 4.1. The CPU times of multigrid and single-grid computations to reach $t = 0.1$ and $t = 0.5$ for $Re = 1000$ are shown in the table. For the same fall of residual, time gain by multigrid is impressive.

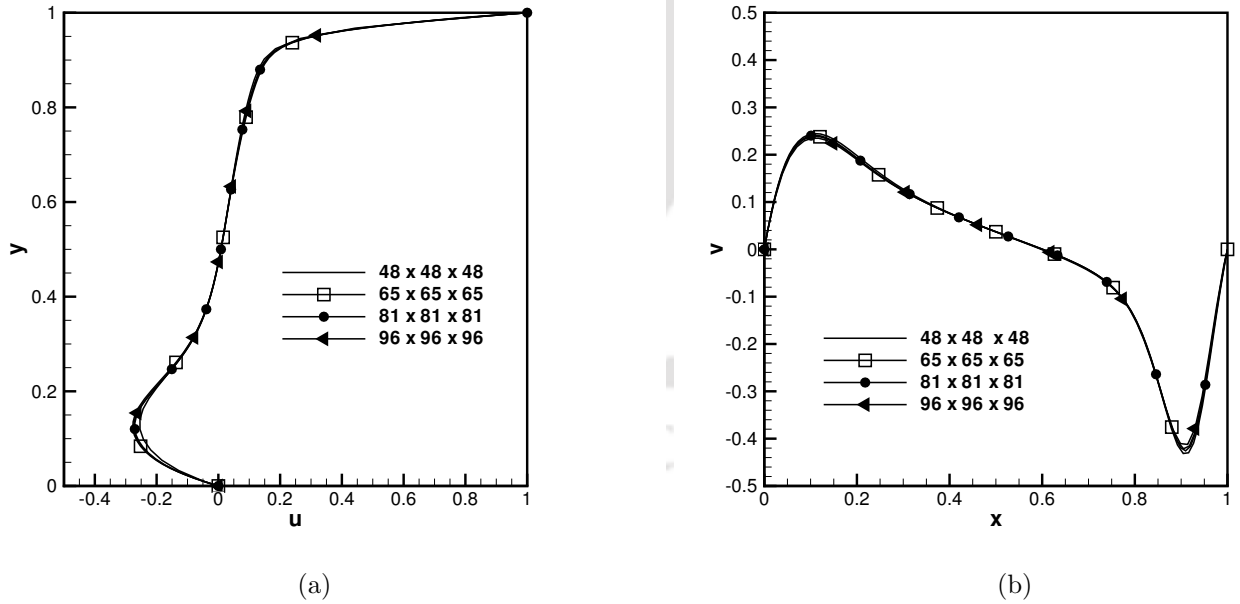


Fig. 4.3: Grid-independence study: steady-state (a) horizontal velocity along vertical centreline (b) vertical velocity along horizontal centreline, at $z = 0.5$ and $Re = 1000$ for the single-sided cubic cavity.

To test the grid independence of the solution numerical experiments were performed for

various grids for $Re = 1000$ and the corresponding results are shown in Fig. 4.3. It is clear that the graded Cartesian meshes of $65 \times 65 \times 65$ is adequate and refining the grid further has no effect on the result (Fig. 4.3(a-b)). Fig. 4.4(a-b) shows the horizontal velocity along

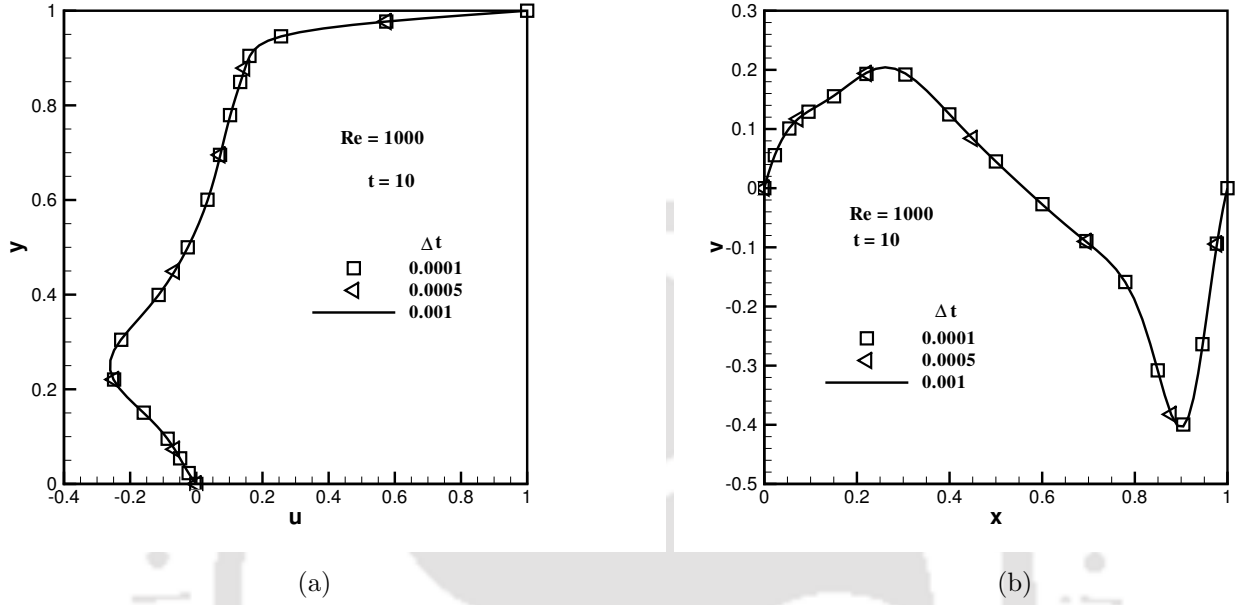
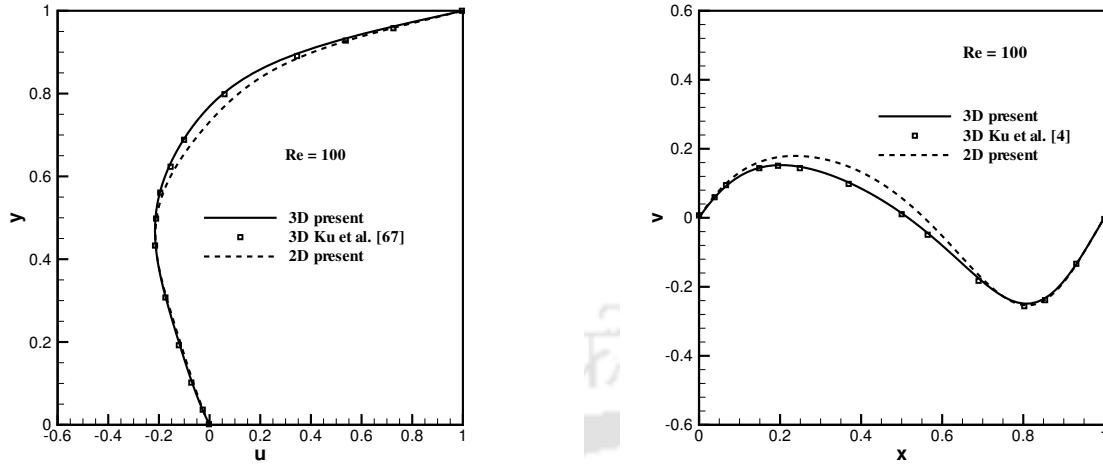
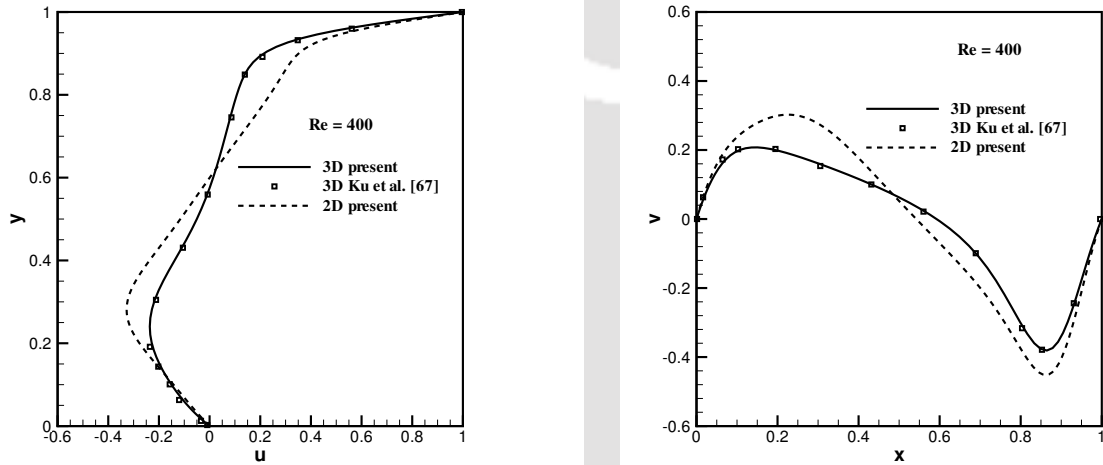
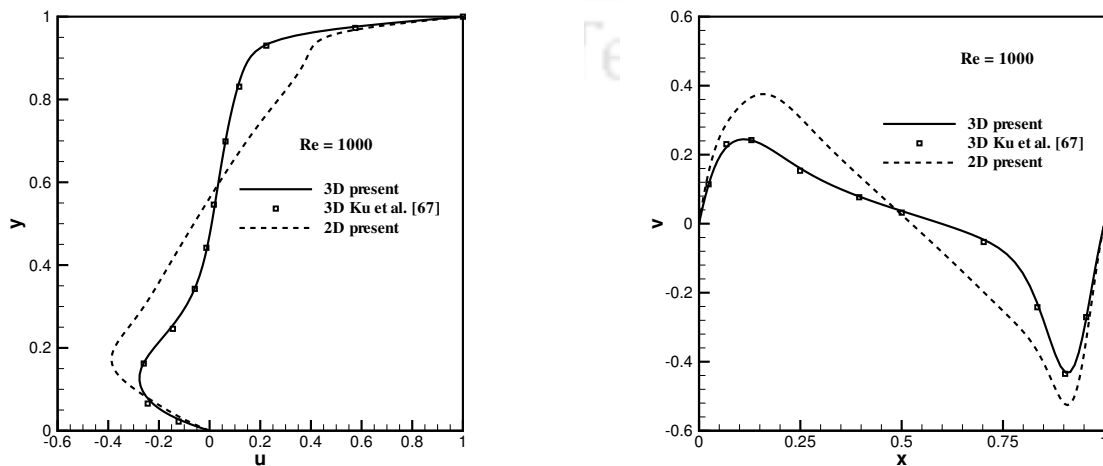


Fig. 4.4: Time-step independence study: (a) horizontal velocity along vertical centreline (b) vertical velocity along horizontal centreline at $z = 0.5$ and $Re = 1000$ for the single-sided cubic cavity.

the vertical centreline and vertical velocity along the horizontal centre line of the cavity respectively at instant $t = 10.0$ captured with three different time steps $\Delta t = 0.001$, 0.0005 and 0.0001 for $Re = 1000$. These profiles plotted for a $65 \times 65 \times 65$ grid show no discernable differences, proving the time-accuracy of the present computations, which is a necessary condition for the transient results to be physically meaningful. Thus, a time step of 0.001 has been used for all subsequent computations. By way of verification the present results for various Reynolds numbers are compared with those available in the literature. Figs. 4.5(a) 4.5(b) show the steady-state velocity profiles of the u -component on the vertical centreline and the v -component on the horizontal centreline at the plane $z = 0.5$. The results show an excellent match with those of Ku *et al.* [67]. The 2D results are also shown for comparison. At $Re = 100$ (Fig. 4.5(i)) there is a small deviation between 2D and 3D results indicating the

(i) $Re = 100$ (ii) $Re = 400$ 

(a) u-velocity

(b) v-velocity

(iii) $Re = 1000$

TH-955_05610304

Fig. 4.5: Comparison of steady-state (a) horizontal velocity along the vertical centreline (b) vertical velocity along the horizontal centreline for $Re = 100, 400$ and 1000 at $z = 0.5$ for

presence of side-wall 3D effects even at relatively low Reynolds numbers. As Re increases to 400 and 1000 the deviations also increase (Fig. 4.5(ii) and 4.5(iii)) indicating that the 3D effect produced by the side walls become stronger with the increase in Reynolds number. Another notable point is that the magnitude of the velocity components for the 3D flow are lower than those for the 2D flow. This is because the momentum imparted to the viscous fluid by the movement of the top wall, in a 2D flow, is confined only to the z -plane where as in a 3D flow it has component in the z -direction as well. The dissipative effect of the side walls may also has a role to play. This reduction in magnitude increases with an increase in Re . To lend further credibility to the present computations we also compare the present

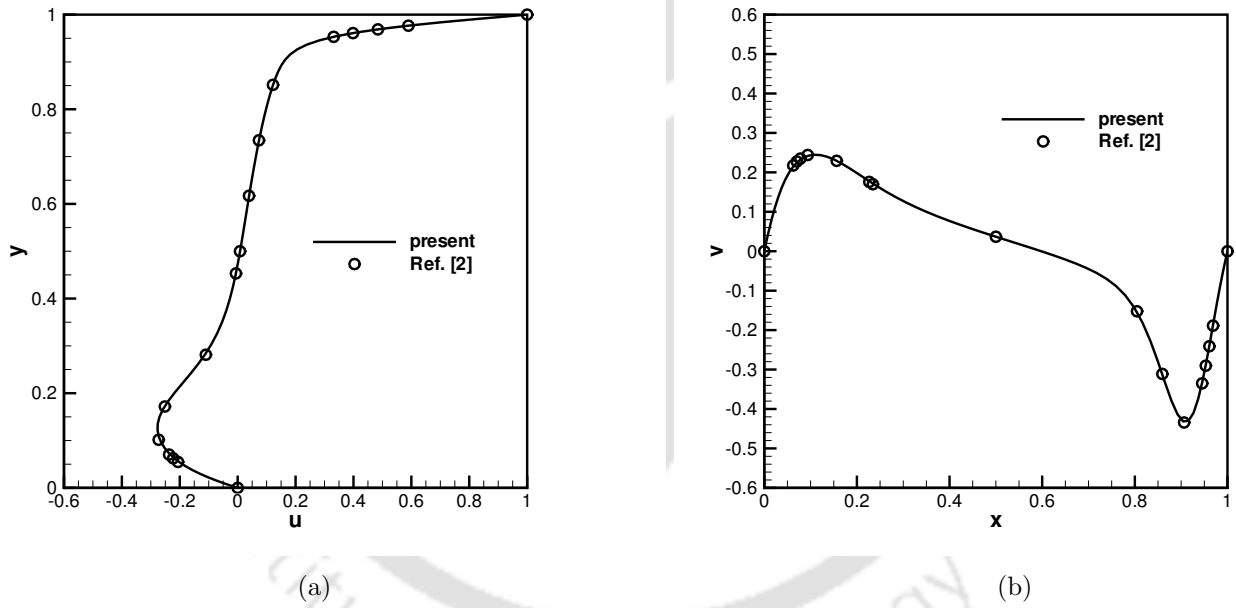


Fig. 4.6: Comparison of steady-state centreline u and v profiles with Albensoeder and Kuhlmann [2] for the single-sided cubic cavity at $Re = 1000$.

mid-span u and v velocity profiles at $Re = 1000$ with the fairly recent and highly accurate results of Albensoeder and Kuhlmann [2] in Fig. 4.6(a-b). The high accuracy of the present computations are brought out by the close matching of the results. As the present results are produced on a $65 \times 65 \times 65$ graded Cartesian mesh and Albensoeder and Kuhlmann uses a grid of size $96 \times 96 \times 64$, the grid economy of the present computations also stand highlighted.

Fig. 4.7 shows the time evolution of streamline patterns for $Re = 400$ at $z = 0.5$ till the steady state is reached. For transient computations, in order to give a divergence-free initial velocity field, the initial values of u , v and w are assumed to be zero at all grid nodes except on the moving top wall where the initial values are $u = 1$ and $v = w = 0$. When the lid is set into motion shear stress develops along the cavity lid, drags the fluid to the top right corner and this fluid then gets deflected by the right wall. The deflected fluid gets mixed with the quiescent fluid region to generate a clockwise-rotating primary vortex close to the top right corner, which in the central z -plane at time $t = 1.25$ is shown in Fig. 4.7(a). With time this vortex grows larger and moves towards the cubic cavity centre as depicted by Fig. 4.7(b) at $t = 2.5$. At $t = 5.0$ an additional counter-clockwise-rotating secondary eddy is also seen at the right wall (Fig. 4.7(c)). Subsequently, this eddy grows bigger and moves towards the bottom right corner, which at $t = 10.0$ is shown in Fig. 4.7(d). At $t = 20.0$ (Fig. 4.7(e)) it is seen that the aforesaid trend of the two vortices continues. Steady state, for a mass residual of 10^{-5} , is obtained at $t = 44.5$ (Fig. 4.7(f)) when compared with the flow pattern at $t = 20.0$ (Fig. 4.7(e)), some minor readjustment is seen to take place especially at the top left corner. Unlike a 2D square cavity flow, no secondary vortex is seen to appear at the bottom left corner. As Fig. 4.7 depicts the evolution of streamline pattern at $Re = 400$, Fig. 4.8 shows the evolution of streamlines for $Re = 1000$ at the symmetry plane $z = 0.5$ till the steady state is reached. The streamlines have a spiral shape for both the Reynolds numbers. Similar to 2D flows, at $Re = 1000$, secondary vortices develop at the right- and left-bottom corners of the cavity. Increase in Re increases the time taken to reach the steady state (see Fig. 4.7(f) and Fig. 4.8(f)). This is because as Re increases so does the scale multiplicity and complexity.

Figs. 4.9 and 4.10 illustrate the change of steady flow streamline patterns and velocity fields with increase in Re at $z = 0.5$. It is observed that similar to 2D cavity flows the primary vortex centre is situated somewhere near the upper-right corner for low Re ; the location of the primary vortex centre cavity then gradually moves towards the centre with an increase in Re . The flow also demonstrates strong 3D characteristics. The steady-state streamline

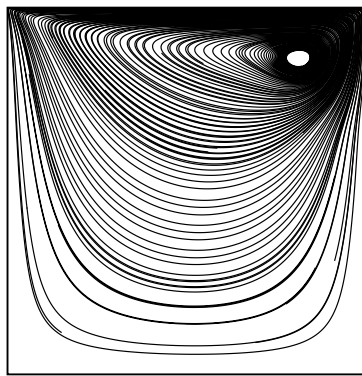
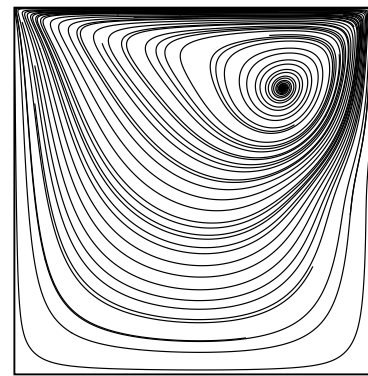
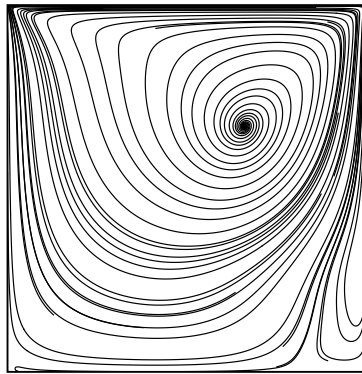
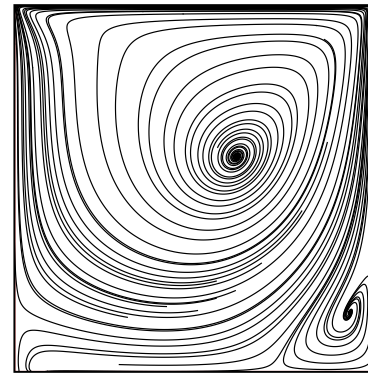
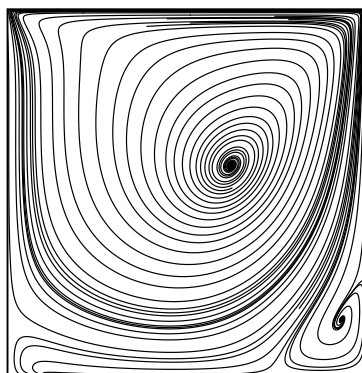
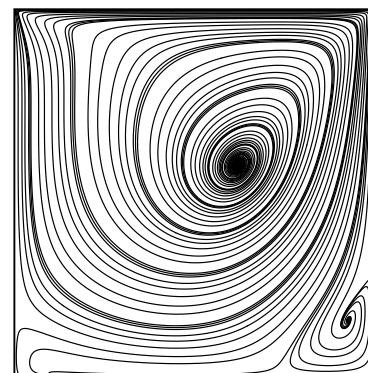
(a) $t = 1.25$ (b) $t = 2.5$ (c) $t = 5.0$ (d) $t = 10.0$ (e) $t = 20.0$ (f) $t = 44.5$ (steady state)

Fig. 4.7: Evolution of projected streamlines for the single-sided cubic cavity at different instants of time at $Re = 400$ and $z = 0.5$.

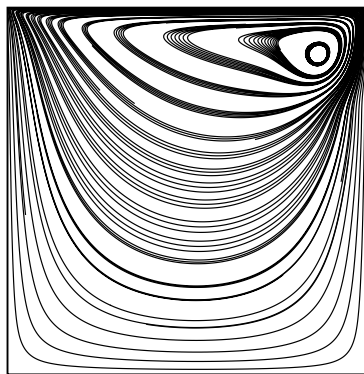
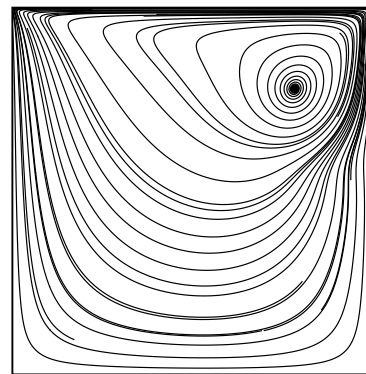
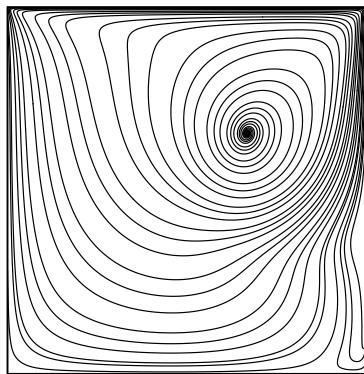
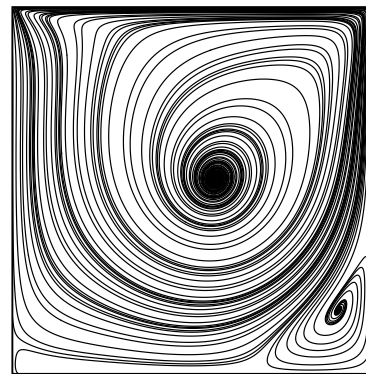
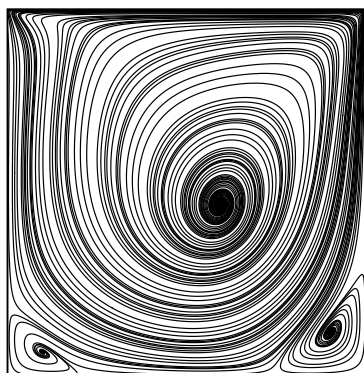
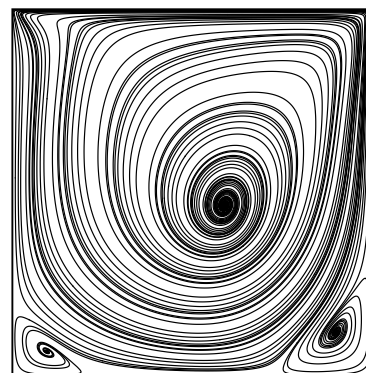
(a) $t = 1.25$ (b) $t = 2.5$ (c) $t = 5.0$ (d) $t = 10.0$ (e) $t = 40.0$ (f) $t = 91.5$ (steady state)

Fig. 4.8: Evolution of projected streamlines for the single-sided cubic cavity at different instants of time at $Re = 1000$ and $z = 0.5$.

patterns and velocity fields obtained by time marching with the present code for $Re = 100$, 400 and 1000 (Figs. 4.9 and 4.10) are consistent with the plots of Ku *et al.* [67]. The

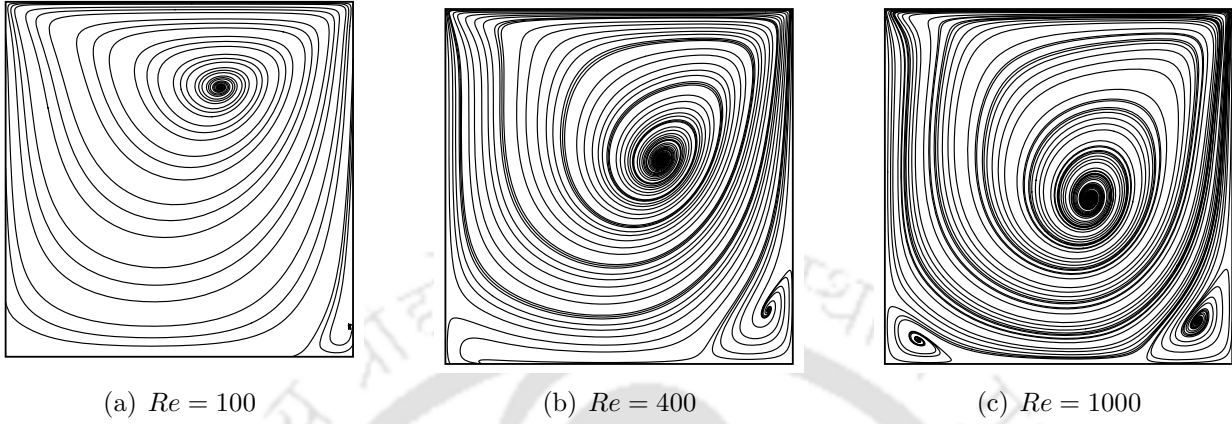


Fig. 4.9: Steady state projected streamline patterns for the single-sided cubic cavity for different Reynolds numbers at symmetry plane $z = 0.5$.

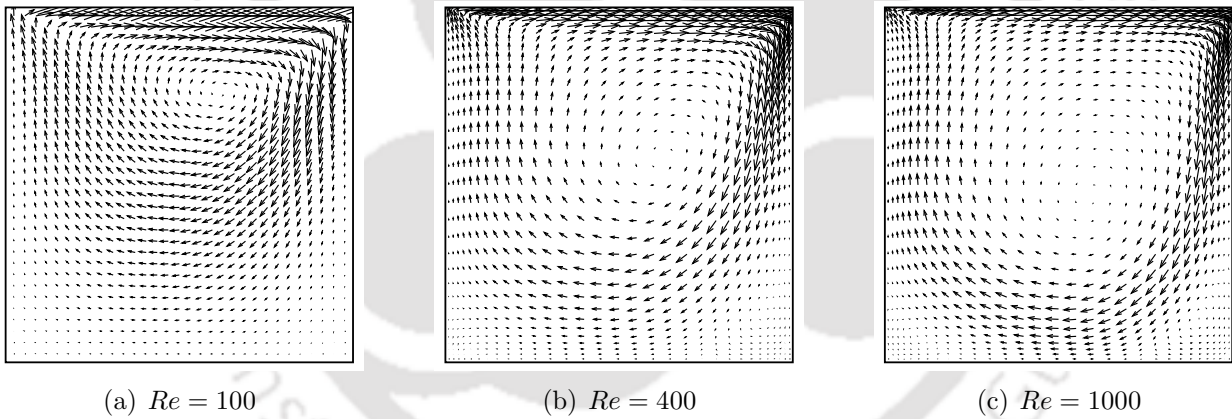


Fig. 4.10: Steady state velocity vectors for the single-sided cubic cavity for different Reynolds numbers at symmetry plane $z = 0.5$.

pressure contours at $z = 0.5$ have been displayed in Fig. 4.11. It is noted that the pressure contours are very close near the top right corner. The close spacing of the contourlines indicate that pressure changes rapidly in the space near this corner. The maximum pressure is also obtained at this corner point. Since pressure is a harmonic function its contours are always open curves and end on the boundaries which is seen in Fig. 4.11. Figs. 4.12 and 4.13 show a comparison of the velocity profiles at different instants of time between 2D and 3D computations at $Re = 400$ and 1000, in which u is plotted along the vertical centreline

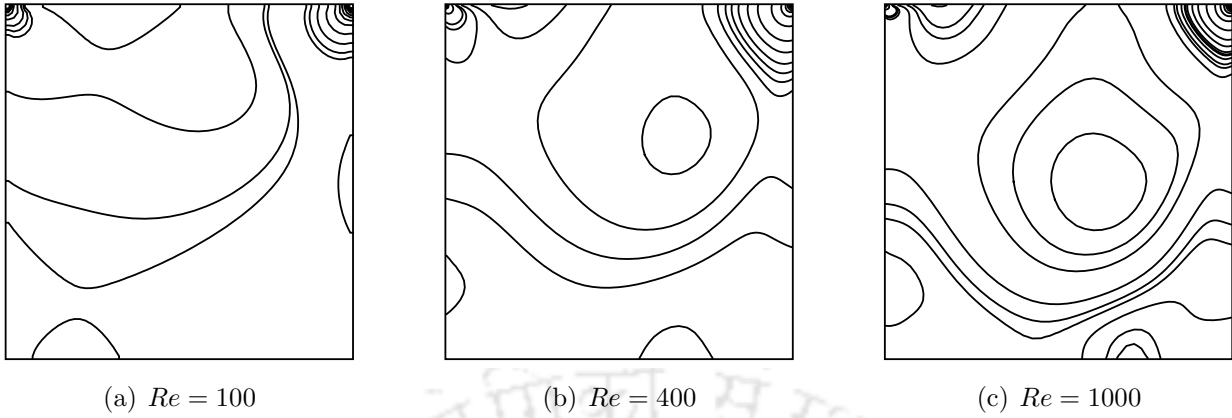


Fig. 4.11: Steady state pressure contours for the single-sided cubic cavity for different Reynolds numbers at symmetry plane $z = 0.5$.

and v is plotted along the horizontal centreline. These plots indicate that some time must elapse before the 3D effects produced by the side walls reach the central ($z = 0.5$) plane as seen at $t = 1.25$ and $t = 2.5$ (Figs. 4.12(a-b) and 4.13(a-b)). With time the flow gradually loses its two-dimensional characteristics as seen from Figs. 4.12(c-d) and 4.13(c-d). This can be confirmed also from Figs. 4.7 and 4.8 which show that unlike at $t = 1.25$ at other instants at mid-span the streamlines show a spiral-like shape. From about $t = 20$ (not shown in the figures) till the time to reach the steady state it is observed that there is no major change in the mid-span velocity pattern. As the presence of the side walls render the flow three-dimensional and the flow velocity also has a z -component, at mid-span u and v are smaller compared with those for a purely 2D flow. This fact and the effect of the spanwise aspect ratio (SAR) is depicted in Fig. 4.14. Expectedly as the SAR increases the flow comes closer to 2D. It is also along expected lines that for the lower $Re = 400$ (Fig. 4.14(a-b)) the discrepancy with 2D flow is less compared with that for the higher $Re = 1000$ (Fig. 4.14(c-d)). Fig. 4.15 show a comparison of steady-state streamline patterns between SAR of 1:1 and 2:1 for $Re = 400$ and 1000 indicating that at SAR = 2:1 the pattern resembles those of a 2D flow.

Fig. 4.16 depicts, at $Re = 1000$, the flow patterns on planes $x = 0.2, 0.5$ and 0.8 . Point 2 in Fig. 4.16(a) is such a point from which the flow apparently immerses in various directions. At $x = 0.5$ (Fig. 4.16(b)) is seen two pairs of Taylor-Gortler-like (TGL) vortices near the

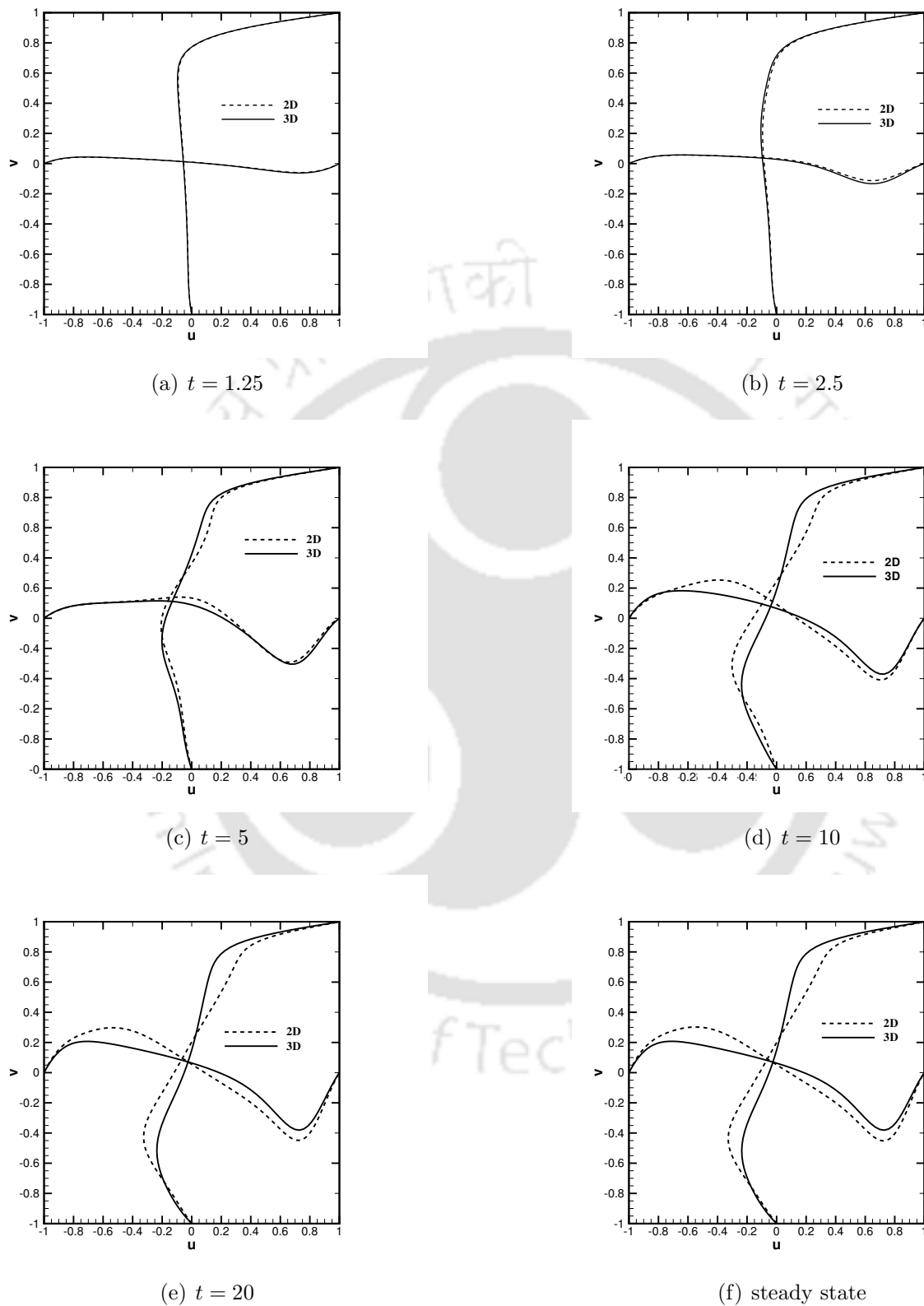


Fig. 4.12: Comparison of 2D and 3D centreline velocity profiles for the single-sided cubic cavity for u and v at different instants of time at $Re = 400$ and $z = 0.5$.

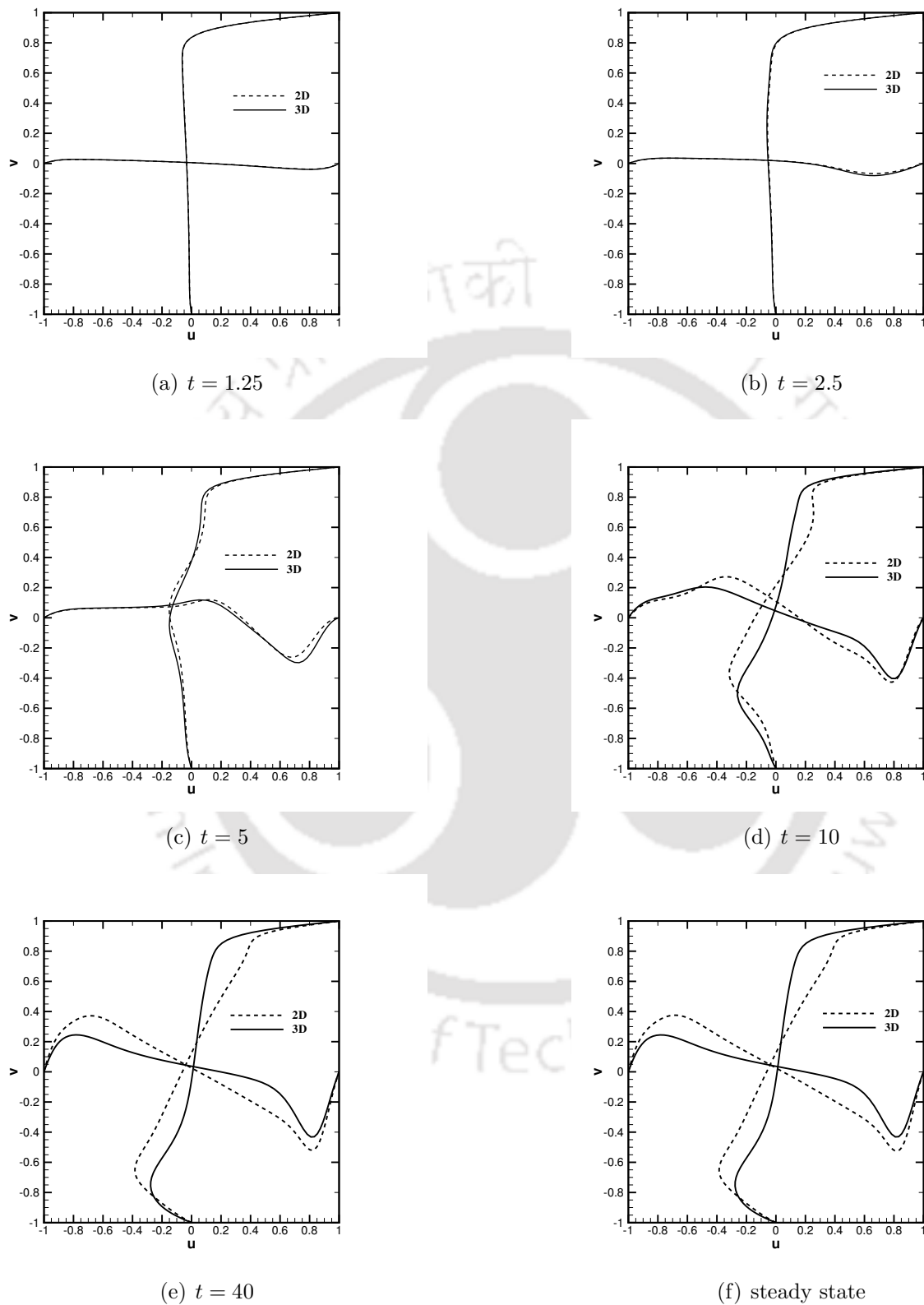


Fig. 4.13: Comparison of 2D and 3D centreline velocity profiles for the single-sided cubic cavity for u and v at different instants of time at $Re = 1000$ and $z = 0.5$.

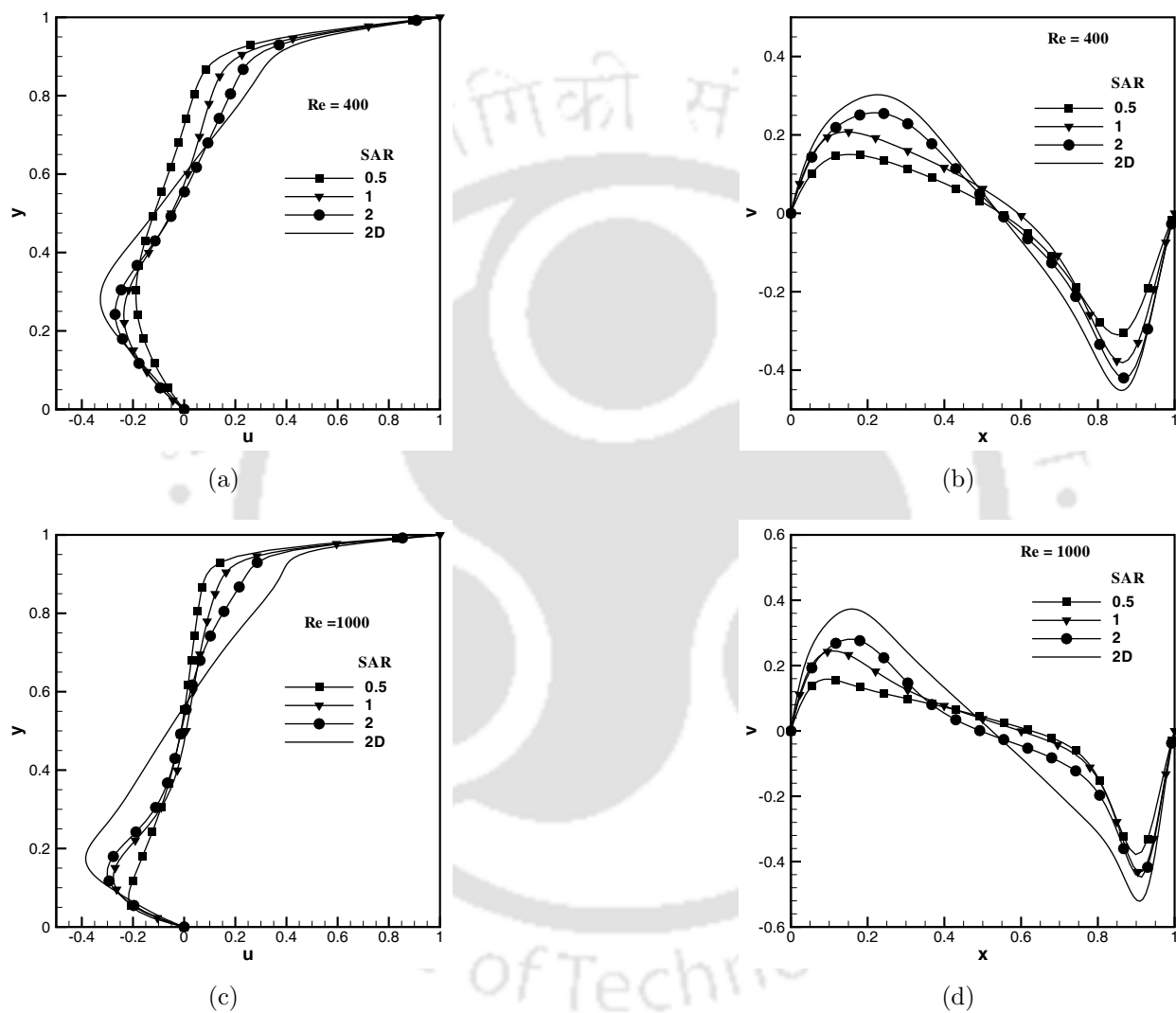


Fig. 4.14: Comparison of steady state velocity profiles for the single-sided cubic cavity at different span-wise aspect ratios for $Re = 400$ and $Re = 1000$.

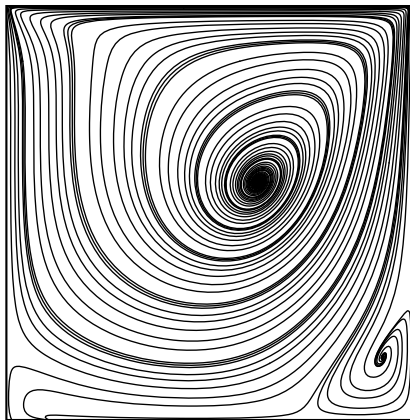
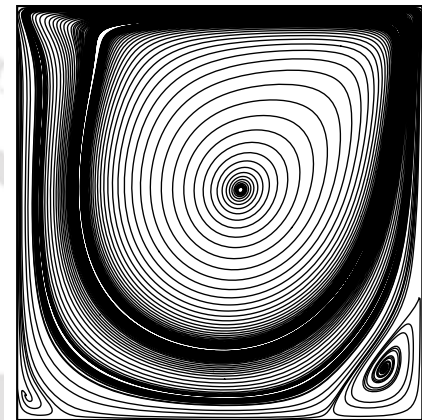
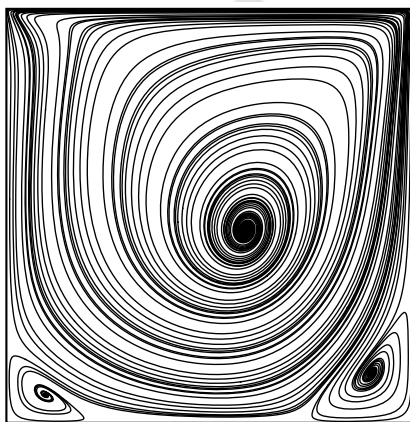
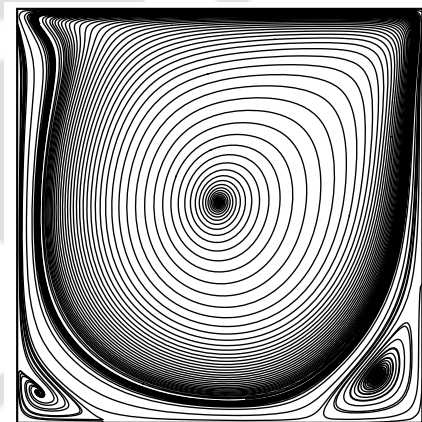
(a) $Re = 400$, $SAR = 1 : 1$ (b) $Re = 400$, $SAR = 2 : 1$ (c) $Re = 1000$, $SAR = 1 : 1$ (d) $Re = 1000$, $SAR = 2 : 1$

Fig. 4.15: Comparison of mid-span ($z = 0.5$) steady-state streamline pattern for the single-sided cubic cavity for span-wise aspect ratio 1:1 and 2:1.

four corners symmetric about the vertical centreline. In Fig. 4.16(c) is seen two saddle points 1 with sink-like appearance, to which the streamlines appear to converge. Fig. 4.17 shows, at $Re = 1000$, the streamline patterns on planes $y = 0.2, 0.5$ and 0.8 , indicating that there is a line of symmetry at $z = 0.5$. At $y = 0.2$ (Fig. 4.17(a)) point 1 represents a saddle point and 2 a point of reattachment. At $y = 0.5$ (Fig. 4.17(b)), i.e., midway between the top and the bottom walls are seen a pair of vortices near the upstream corners. This pair of vortices, symmetric about $z = 0.5$, covers the entire span. Fig. 4.17(c) depicts the flow pattern near the top wall at $y = 0.8$. Point 2 in the figure is a point of reattachment. Fig. 4.18 gives, at $Re = 1000$, the flow patterns on planes $z = 0.2, 0.5$ and 0.8 . Planes $z = 0.2$ and 0.8 located near the side walls and as they are symmetrically placed about $z = 0.5$, the flow patterns in these planes are identical (Fig. 4.18(a)) and 4.18(c)). There are no secondary vortices near the bottom corners in these planes because of the side-wall effect. At $z = 0.5$, i.e., midspan two secondary vortices are observed (Fig. 4.18(b)) near the bottom corners as in the case of the 2D cavity flow. Fig. 4.19 shows, at $Re = 1000$, the time evolution of the streamline patterns at $x = 0.5$, a plane normal to the direction of wall movement. Evidently it takes some time for the endwall effects to reach all parts of the cavity. At $t = 1.25$ (Fig. 4.19(a)) the projection of the streamlines on this plane appear more or less vertical along the entire span, showing that flow is close to a 2D flow without end walls. At $t = 2.5$ (Fig. 4.19(b)) there is a manifestation of the 3D endwall effect in that the streamlines, though more or less vertical, are no longer parallel. Fig. 4.19(c) shows that at $t = 5$ there is marked presence of the end walls and their 3D effect with the appearance of the point of reattachment 2. At this instant, however, the mid-span flow still closely resembles a 2D flow (Fig. 4.13(c)). At $t = 10$ (Fig. 4.19(d)) is seen two pairs of secondary TGL vortices symmetrically placed about midspan ($z = 0.5$), one near the top and the other near the bottom corners. Sidewall 3D effects are marked even at midspan now (Fig. 4.13(d)). At $t = 40$ (Fig. 4.19(e)) all the TGL vortices are seen to move closer to the nearest cavity corners. The steady-state flow pattern shown at $t = 91.5$ (Fig. 4.19(d)) is not much different from that at $t = 40$ (Fig. 4.19(e)). At all the instants the flow is seen to maintain its symmetry about $z = 0.5$.

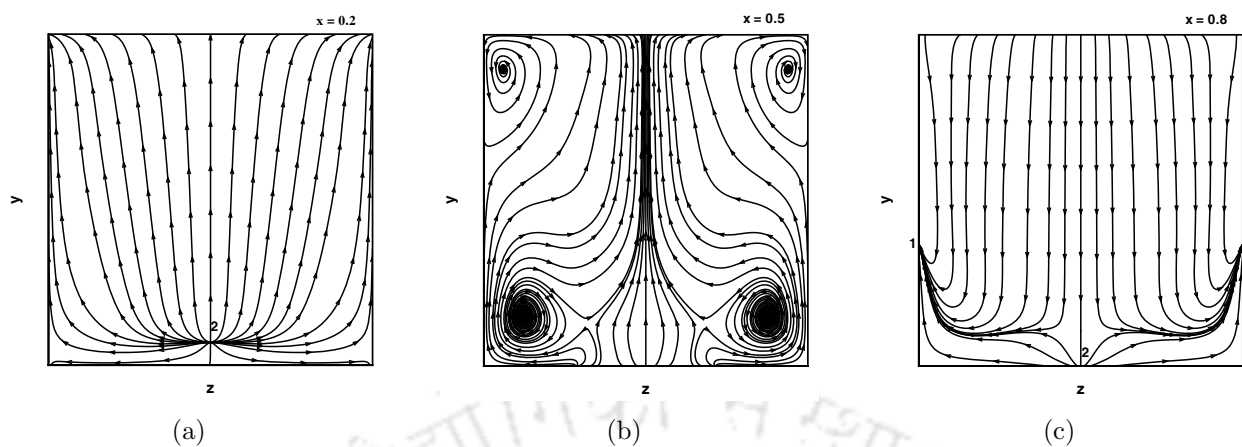


Fig. 4.16: Projected streamline patterns for the single-sided cubic cavity in different x -planes at $Re = 1000$.

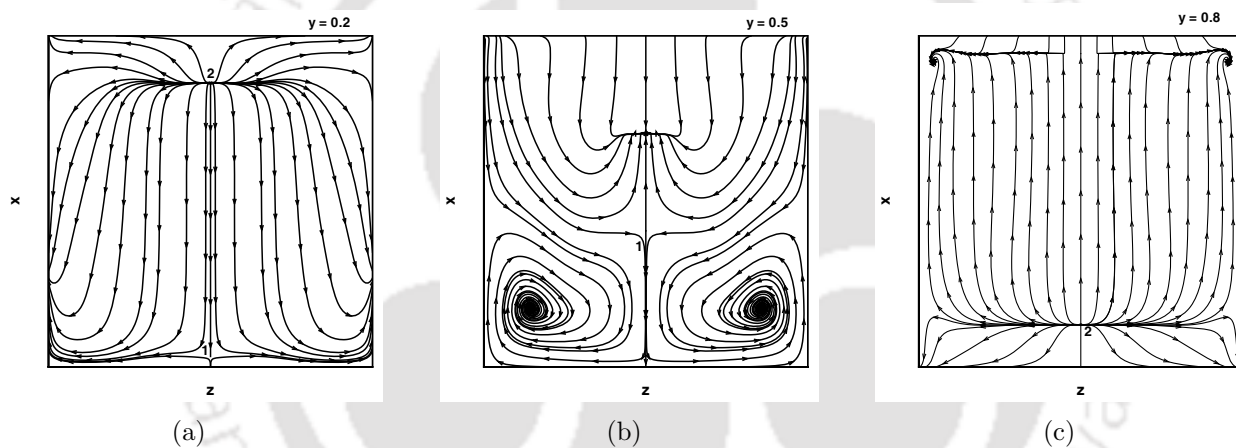


Fig. 4.17: Projected streamline patterns for the single-sided cubic cavity in different y -planes at $Re = 1000$.

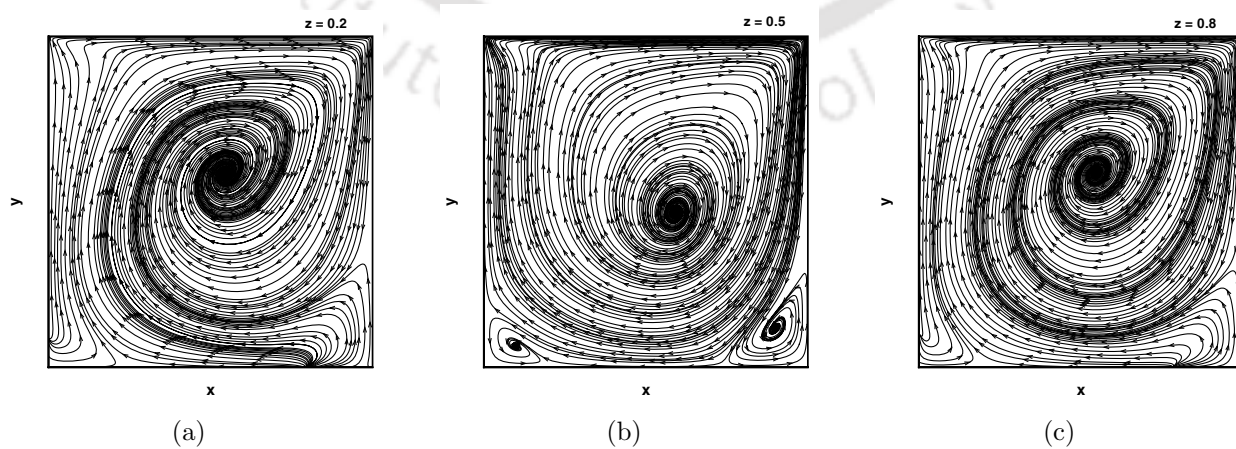


Fig. 4.18: Projected streamline patterns for the single-sided cubic cavity in different z -planes at $Re = 1000$.

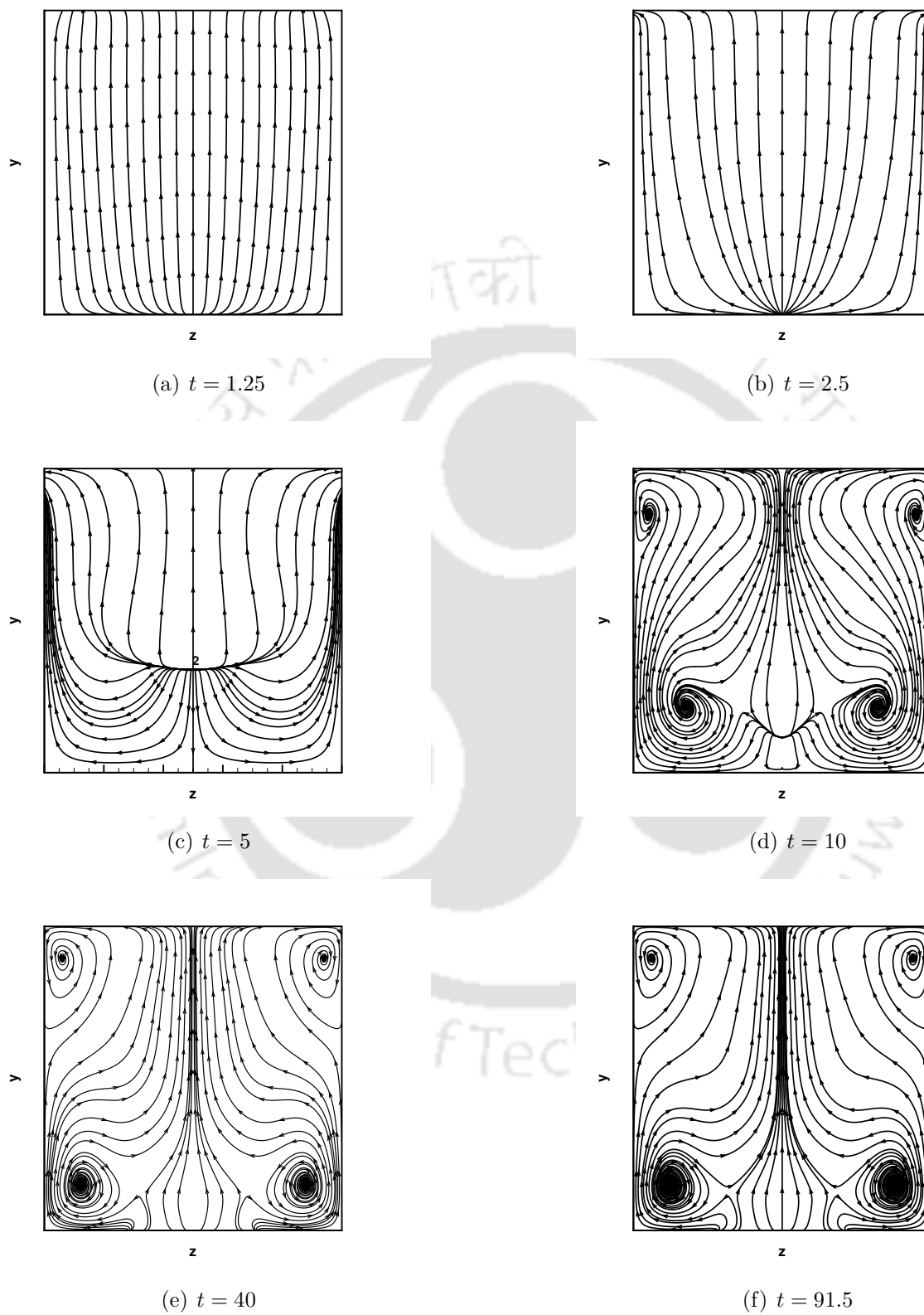


Fig. 4.19: Time evolution of projected streamline patterns for the single-sided cubic cavity at $x = 0.5$ and $Re = 1000$.

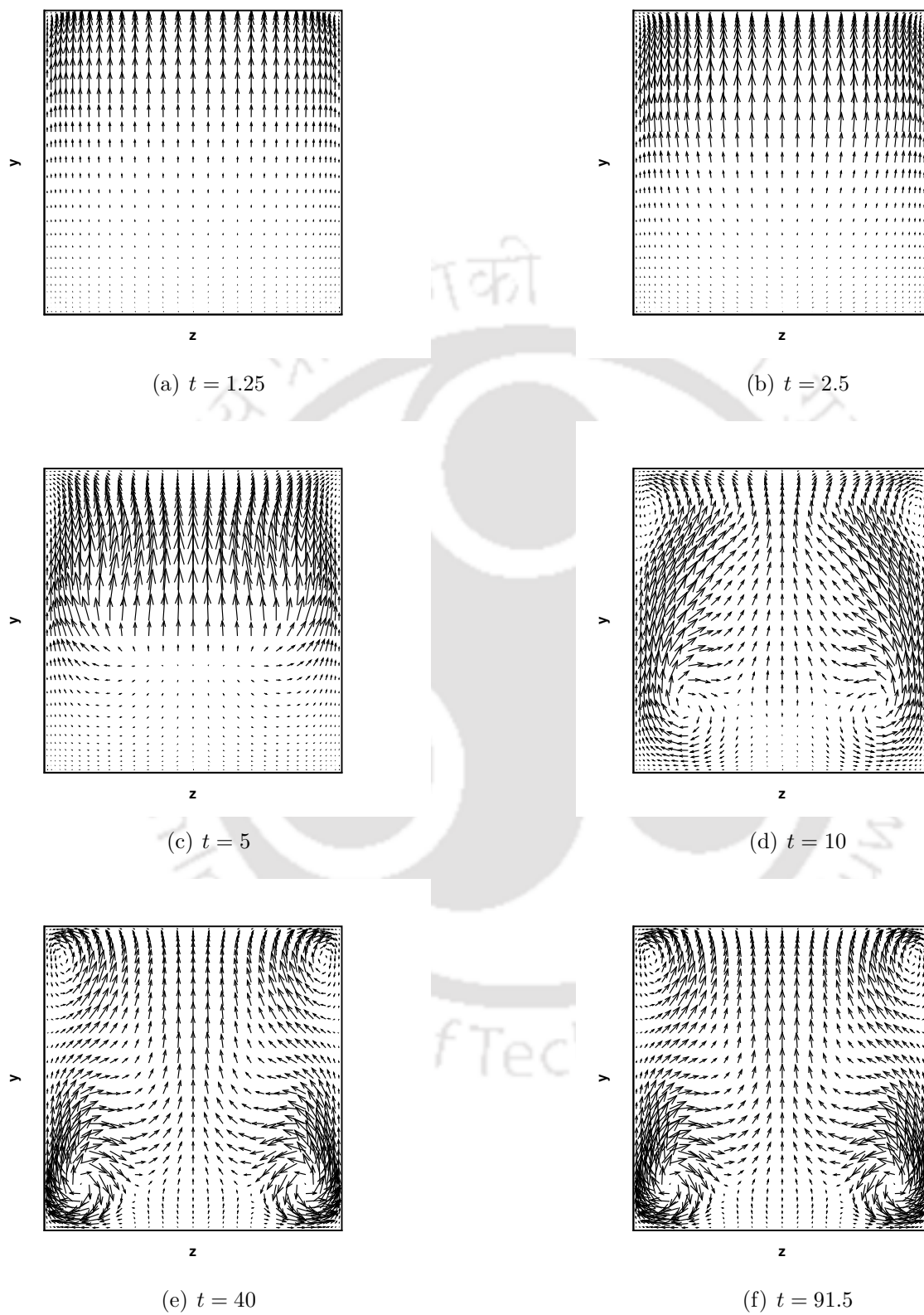


Fig. 4.20: Time evolution of velocity vectors for the single-sided cubic cavity at $x = 0.5$ and $Re = 1000$.

4.3.2 Two-Sided Lid-Driven Cubic Cavity: A New Test Problem

The results presented in the preceding section establish the accuracy and the capability of the present code to accurately compute transient viscous flows. After having thus gained confidence in the code, it is then applied to compute an hitherto unexplored flow. The problem consists of a cubic cavity filled with a stationary fluid that is set into motion by the sudden simultaneous movement of the top and the bottom walls from the left to right. The physical boundary conditions of the geometry are specified for $t \geq 0$ as

At $x = 0$ and 1 , $u = v = w = 0$

At $z = 0$ and 1 , $u = v = w = 0$

At $y = 0$, $u = 1, v = 0, w = 0$

At $y = 1$, $u = 1, v = 0, w = 0$

The reason for selecting this flow configuration for numerical study is that this geometry involves interesting flow features including gradual development of a free shear layer and associated off-corner secondary vortices. The use of a grid with appropriate clustering and the use of multigrid suggests that the present computations combine accuracy, space-grid economy and time-wise efficiency. We first show (Fig. 4.21) the sufficiency of the space grid of size $65 \times 65 \times 65$ clustered near the walls, by reproducing the steady-state results of Beya and Lili [10] for the flow in a cubic cavity with two-sided nonfacing wall movement. As has already been mentioned the flow in a cubic cavity with two-sided parallel wall movement has not been examined so far. We compute the flow in this geometry in a grid of similar size ($65 \times 65 \times 65$) but clustering points not only in the wall shear layers but also in the free shear layer mid-way between the moving walls (Fig. 4.22(a)). Computations are carried out for $Re = 100, 400$ and 600 . Fig. 4.22(b-d) shows the steady-state streamline patterns asymptotically obtained from the present transient computations on the $z = 0.5$ plane (x being the direction of wall movement as in the case of single-sided cavity). Fig. 4.22(b) shows that at $Re = 100$ two counter-rotating primary vortices appear and there is no trace of any near wall secondary vortices. At $Re = 400$ (Fig. 4.22(c)), apart from the primary vortices a counter-rotating pair of small secondary vortices symmetrically placed about the horizontal

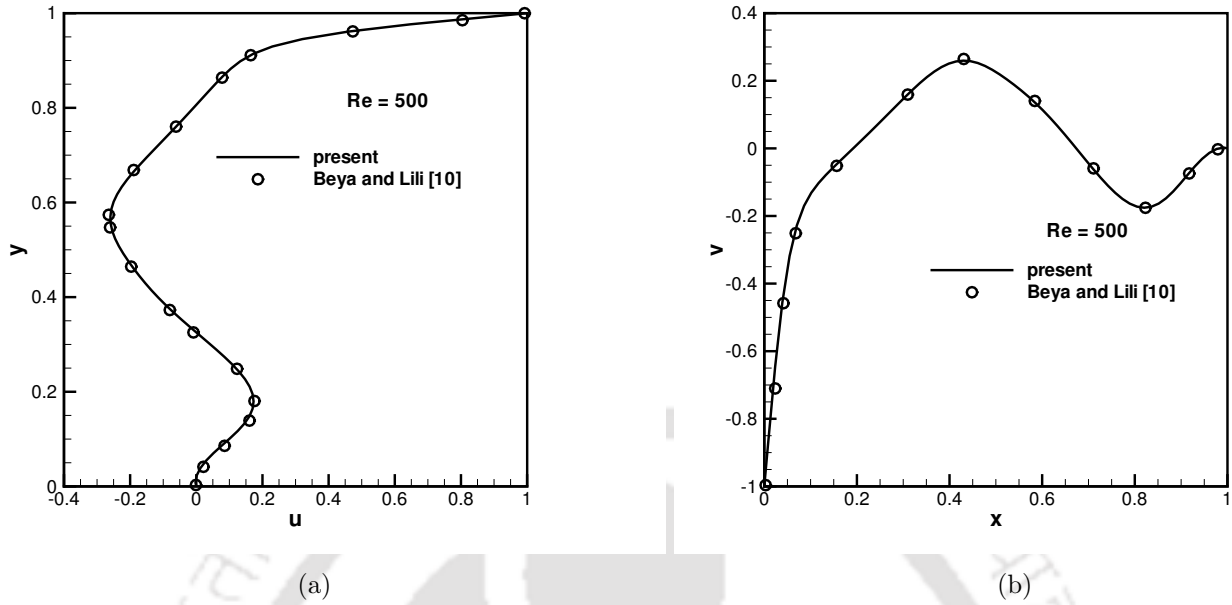


Fig. 4.21: Comparison of steady-state centreline u and v profiles with Beya and Lili [10] for a two-sided non-facing lid-driven cubic cavity on a $65 \times 65 \times 65$ grid at $Re = 500$.

cavity centreline is seen to appear near the centre of the right wall. Also a free shear layer about this centreline is now more prominent. Fig. 4.22(d) shows the streamline pattern for $Re = 600$. It is seen that the size of the secondary vortices increases significantly with an increase in Reynolds number. It is also seen that with an increase in Re the distance of the top and bottom primary vortex cores from the top- and bottom-right corners increases and at $Re = 600$ they take position near the centres of the top and bottom halves of the cavity. The counter-rotating pairs of primary and secondary vortices maintain their symmetry about the horizontal centreline for all the Reynolds numbers. Beya and Lili [10] points out that for a 3D two-sided non-facing lid-driven cavity at $Re = 600$ the flow is not steady and periodic solution exists, whereas for the two-sided lid-driven cubic cavity with parallel wall motion we find that there exists a unique, steady and stable solution. In order to distinguish between the 2D and 3D steady-flow patterns, 3D midspan ($z = 0.5$) velocity profiles are compared with 2D in Fig. 4.23(a-b). Fig. 4.23(i) shows the variation of u along the vertical centreline ($x = 0.5$) and variation of v along the horizontal line ($y = 0.75$) respectively. At $Re = 100$ the deviation between 2D and 3D results (Fig. 4.23(i)), though small, is good enough to

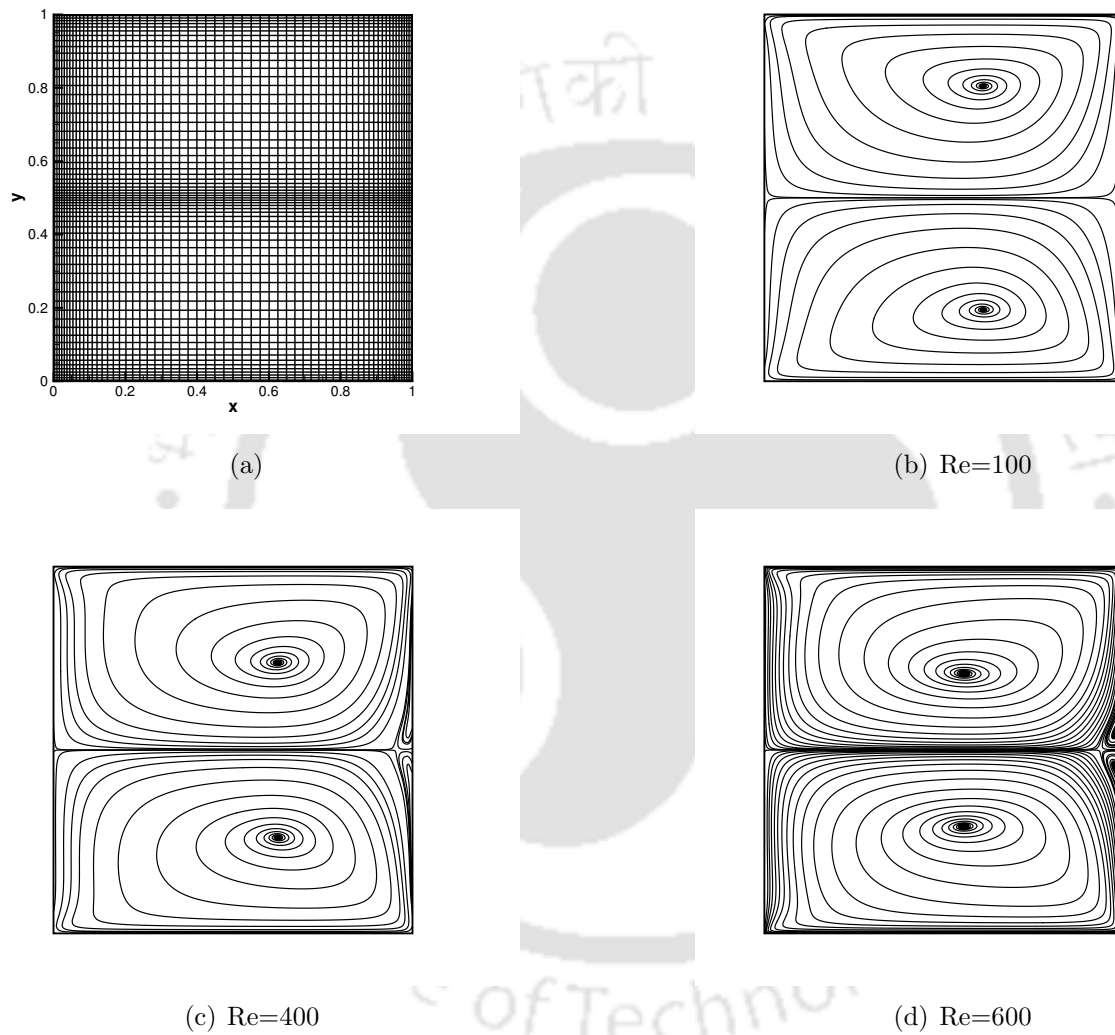


Fig. 4.22: 2D view of the grid in the z -plane and steady-state projected streamline patterns for different Re 's for the facing-wall two-sided cubic lid-driven cavity flow on a $65 \times 65 \times 65$ grid.

indicate that sidewall effects can be perceived at mid-span even at relatively low Reynolds numbers. Similar to the single-sided cavity studied earlier, as Re increases to 400 and 600 the deviations between 2D and 3D results at the mid-span increases (Fig. 4.23(ii) and 4.23(iii)). Again like the single-sided lid-driven cubic cavity the magnitude of the velocity components for the 3D flow are lower than those of the 2D flow owing to the 3D effect of the side walls. This reduction in magnitude increases with an increase in Re .

The transient problem of the two-sided cubic cavity flow studied here consists in the impulsive movement of the top and bottom walls from the left to right. The initial velocity field is obtained by taking u , v and w as zero at all grid-nodes except those on the top and the bottom moving walls where u is taken as unity, v and w as zero. Thus at $t = 0$ we start from a divergence-free flow, which is a necessary condition to be satisfied by an incompressible flow at every instant. To ensure time-accuracy time steps chosen are so small as to make the results time-step independent. Fig. 4.24 depicts, at midspan ($z = 0.5$), the time-wise evolution of the streamline patterns at $Re = 600$ till the steady state is reached. Fig. 4.24(a) shows the streamline pattern at $t = 1.0$ when the primary vortex cores are seen close to the top right and bottom right corner symmetric to each other about the horizontal centreline. At this instant the secondary vortices have not developed as yet. At time $t = 3.0$ (Fig. 4.24(b)) the two primary vortex cores are seen to move away from the nearest corners maintaining their symmetry about the horizontal centreline. Also seen at this instant is the emergence of a pair of two counter-rotating secondary vortices symmetrically placed about the horizontal centreline near the centre of the right wall. At time $t = 5.0$ (Fig. 4.24(c)) the primary vortex cores are seen to continue to move away from their nearest corners whereas the size of the pair of secondary vortices enlarges. Fig. 4.24(d) shows the streamline pattern at time $t = 10.0$, when the movement of the primary vortex cores away from the respective corners of origin is seen to continue and reach near the centres of the top and bottom halves but the secondary vortex sizes do not change much compared with those at $t = 5.0$. For a fall of mass residual below 10^{-5} the steady state is reached at time $t = 42.8$. At this steady state (Fig. 4.24(e)), the two counter-rotating primary vortex cores are stably positioned.

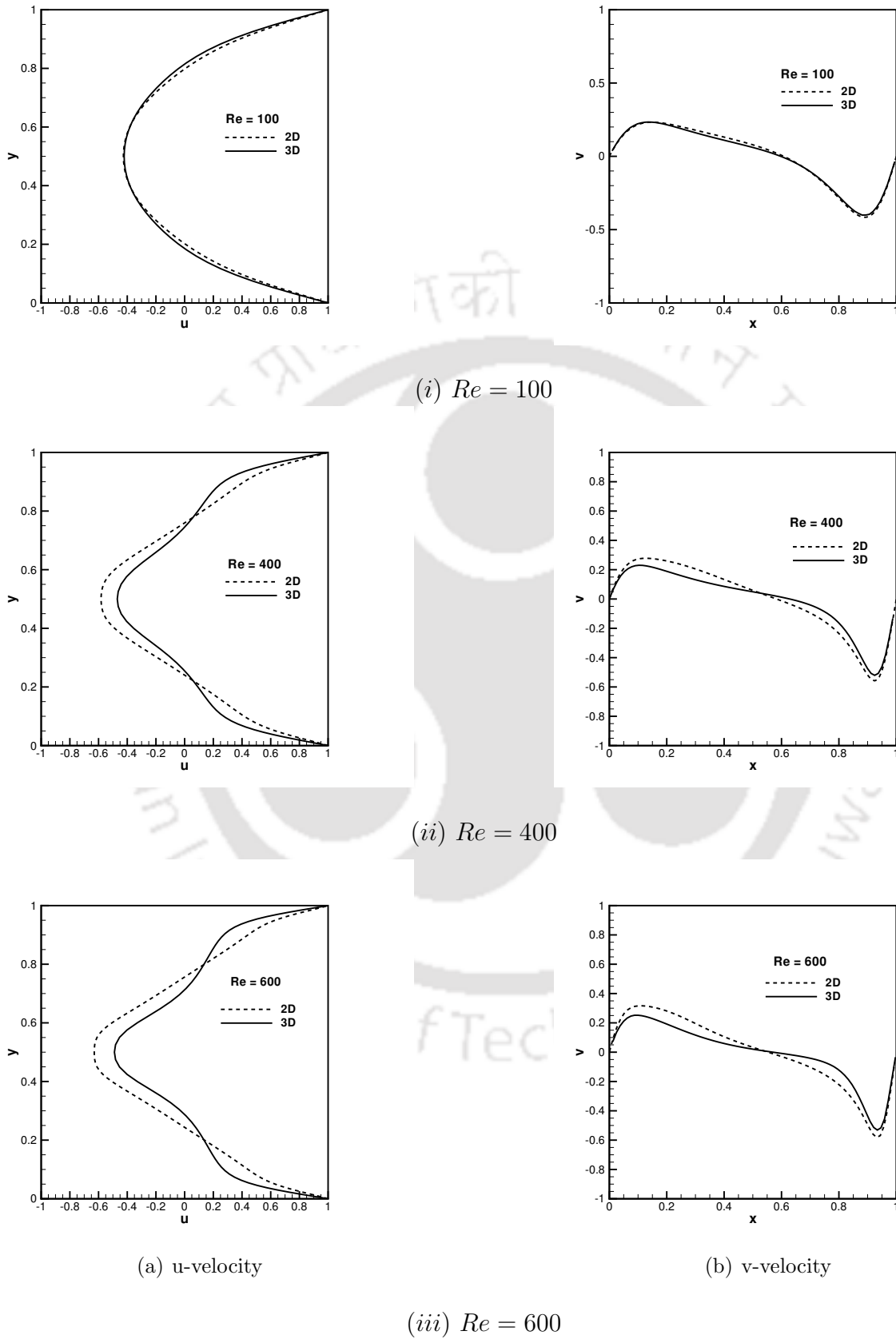


Fig. 4.23: Steady-state (a) Horizontal velocity along the vertical centreline (b) Vertical velocity along the horizontal centreline for the facing-wall two-sided cubic lid-driven cavity flow for $Re = 100, 400$ and 600 at $z = 0.5$.

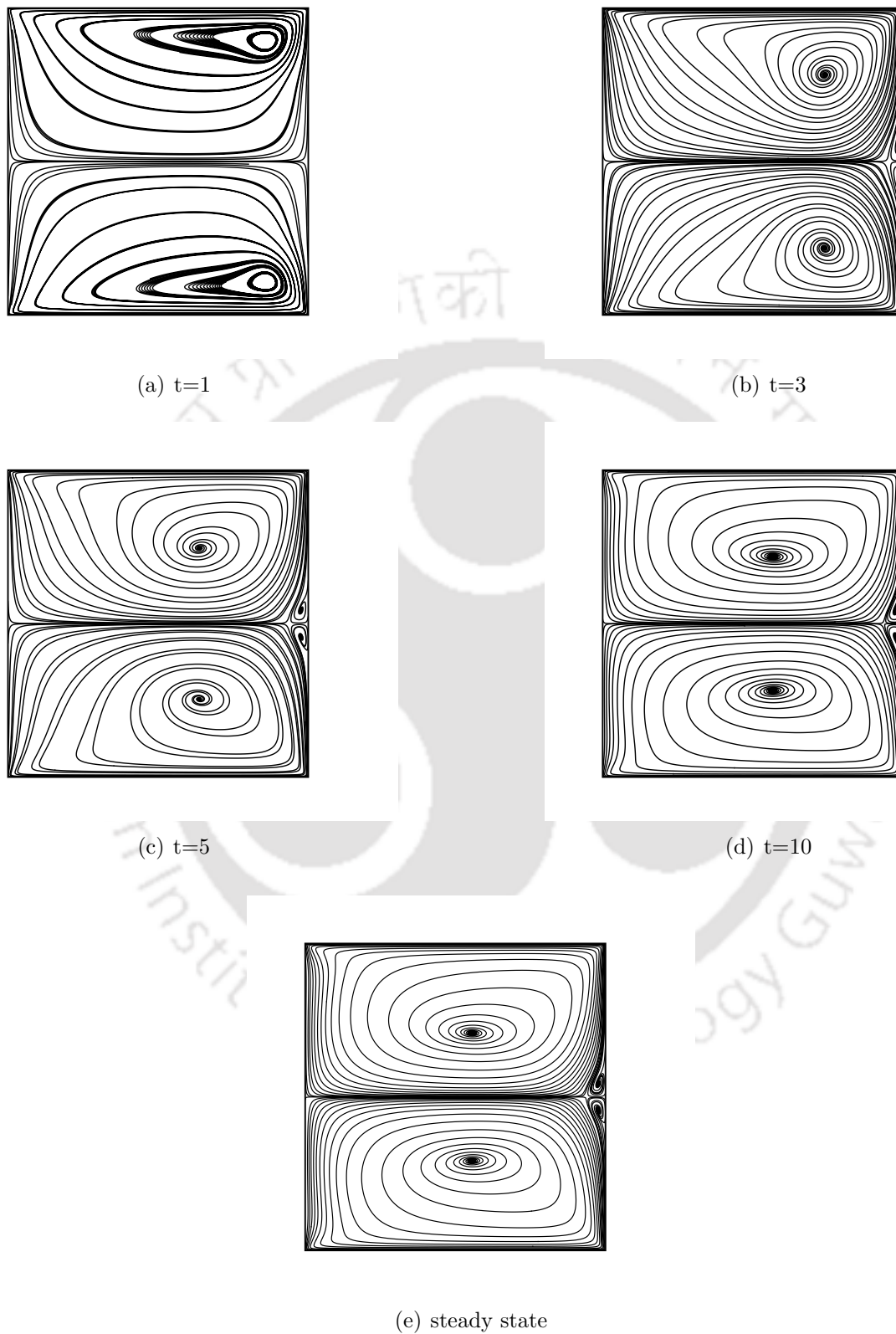


Fig. 4.24: Time evolution of projected streamlines for the facing-wall two-sided cubic lid-driven cavity flow at $Re = 600$.

The top one is located slightly below the geometric centre of the upper half and the bottom one slightly above the geometric centre of the lower half of the cavity. Also there are no appreciable changes in the positions and shapes of the counter-rotating pair of secondary vortices. Observing these figures in the chronological sequence also reveals that the flow gradient in the free shear layer about the horizontal centreline of the cavity becomes sharper with time. Because of grid clustering at wall and free shear layer, the second order time and higher order spatial accuracy of the scheme and the care taken to maintain time-step-independence of the results throughout, the obtained transient computations for this hitherto unexamined flow configuration are highly reliable. Use of high precision arithmetic to avoid possible accumulation of round-off errors lends further credibility to the results. The time-wise efficiency of the transients computations is also high as multigrid has been used to good effect. Overall these results are produced accurately and efficiently with grid economy.

4.4 Conclusions

The present work is about the development and application of a multigrid-accelerated 3D code that accurately and efficiently computes incompressible transient viscous flows especially in geometries that allow graded Cartesian meshes to be used conveniently to resolve the sharp gradients in shear layers by locally clustering the grid there. A fractional-step method is used to obtain second-order temporal accuracy and spatial discretizations of the convective and viscous terms are of third- and fourth-order spatial accuracy respectively. The time-consuming process of computation of the Poisson equation at every time step is accelerated by using a V-cycle multigrid approach. For accurate implementation of the multigrid method, finite difference discretization and grid-transformation of the graded Cartesian meshes are employed. First, computations of transient and steady-state flows are carried out for the single-sided lid-driven cubical cavity and a brief analysis of the results is carried out. The code enjoys the advantage of a good time-wise efficiency bestowed by multigrid. Also the use of graded mesh and good spatial accuracy of the discretization allows the code

to reproduce the highly accurate steady-state results of Albensoeder and Kuhlmann [2] for the single-sided cubical cavity flow using a much coarser-grid. After having used the code to compute the flow in the single-sided cubical cavity to establish its credibility, it is then used to compute transient and steady-state flows in a hitherto unexplored configuration, namely, the two-sided cubical cavity flow induced by the motion of two facing walls in the same direction. The grid here is clustered not only near the walls but also in the free shear layer midway between the moving walls to obtain a good grid-economy. It is observed that for the nonfacing two-sided cubic cavity, flow becomes periodic at $Re = 540$. For the facing wall two-sided cubic cavity, however, steady solutions are obtained much beyond that Reynolds number and steady and transient results are presented here also for such Re 's. Various observations are made about this untested flow configuration. As all the steady-state results are grid independent, the transient results are time-step independent produced on the same grids and the accuracy of the spatio-temporal discretization is sufficiently high, all the steady and transient results reported for this new flow configuration are likely to be highly reliable and accurate.

Chapter 5

Multiple Stable Solutions for Two- and Four-Sided Lid-Driven Cavity Flows using FAS Multigrid Method

5.1 Introduction

In the past two decades the semi-implicit method for pressure linked equation (SIMPLE) algorithm is probably the most widely used method for the calculation of incompressible flows. The SIMPLE algorithm of Patankar and Spalding [86] is used to couple the momentum and continuity equations. In this method the discretization equations for velocity components and a pressure correction equation are solved in a sequential manner. The pressure correction formulation which is used to link the pressure and the velocity fields, however, is weak and leads to slow convergence or even divergence if appropriate under-relaxations are not used. A major factor affecting the performance of a scheme is its ability to smoothen out the errors of all frequencies effectively in the solution. Iterative methods, such as line-by-line solvers, provide excellent smoothening of the errors of wavelengths comparable to the mesh size. As the grid is refined, these solvers show fast convergence in the initial few iterations, but subsequently the convergence rate deteriorates rapidly. Therefore, a full-approximation storage

(FAS) multigrid method which is particularly applicable for solving non-linear equations is used in the present work. In 1988, Sivaloganathan and Shaw [99] applied multigrid method to SIMPLE pressure correction technique of Patankar and Spalding [86] using staggered grid. They adopted the full-approximation storage multigrid method and achieved a large reduction in CPU time. Since then application of multigrid method in SIMPLE pressure correction smoothers has increased. Grid-coarsening process is difficult in staggered grid arrangement because of the use of three different set of cells. Hence a colocated grid layout is used in the present work. The momentum interpolation proposed by Rhie and Chow [93] is employed in the present work for calculating the cell-face mass fluxes to avoid pressure oscillations. The FAS multigrid code developed in the present work for one-sided lid-driven cavity exhibit fast convergence and accuracy. The results are compared with well-established results. After having thus gained confidence in the code it is then applied to compute the multiple solutions for two-sided and four-sided lid-driven cavities.

Kuhlmann *et al.* [68,69] extended the one-sided lid driven cavity problem to a two-sided problem, where the flow is driven by the parallel or antiparallel motion of two facing walls. The facing walls could be either the left and right walls or the upper and lower walls. At low Reynolds number, the flow consists of separate co- or counter-rotating primary vortices that form adjacent to each moving wall. At higher Reynolds numbers, instabilities arise in the flow due to the interaction between the two primary vortices. Moreover, their results showed that multiple flow solutions may exist, depending on the cavity aspect ratio and the value of the Reynolds number [3]. Numerical simulations of two-sided lid driven cavity flow with temperature gradient and accompanied by heat and mass transport were also reported by Alleborn *et al.* [4] and Luo and Yang [77]. More recently, the multiplicity of flow states induced by the motion of two-sided non-facing lid-driven cavity flow and four-sided lid-driven cavity flow have been investigated by Wahba [113]. He found the critical Reynolds number of 1073 for the two-sided lid-driven cavity and 129 for the four-sided lid-driven cavity. We have reproduced the results of [113] in addition with pressure contours to show the ability and accuracy of multigrid method that the symmetric solution exists for all Reynolds numbers

where as multiplicity of states of one symmetric and two asymmetric solutions for the two-sided and four-sided cavity flows are identified above the critical Reynolds number. Here we have captured multiple steady solutions at post-critical Reynolds numbers for both the two-sided and four-sided cavity flows using FAS multigrid method. The strategy employed to obtain these solutions is also described.

5.2 Governing Equations and Numerical Method

The flow is considered to be laminar and incompressible. The fluid physical properties are assumed to be constant. Under these assumptions the governing equations in the traditional primitive variable formulation in the non-dimensional form can be written as

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \quad (5.1)$$

$$u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = -\frac{\partial p}{\partial x} + \frac{1}{Re} \left[\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right] \quad (5.2)$$

$$u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} = -\frac{\partial p}{\partial y} + \frac{1}{Re} \left[\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right] \quad (5.3)$$

Here u and v denote the velocities along x and y directions and p the pressure, respectively. The dimensionless governing equations for a general field variable ϕ for steady flow can be written as

$$\frac{\partial}{\partial x} (u\phi) + \frac{\partial}{\partial y} (v\phi) = \frac{\partial}{\partial x} \left(\Gamma^\phi \frac{\partial \phi}{\partial x} \right) + \frac{\partial}{\partial y} \left(\Gamma^\phi \frac{\partial \phi}{\partial y} \right) + S^\phi \quad (5.4)$$

where ϕ is the dependent variable and, Γ^ϕ and S are the diffusion coefficient and source term respectively. The following substitutions in Eq. (5.4) give the continuity equation:

$$\phi = 1, \quad \Gamma^\phi = 0, \quad S^\phi = 0 \quad (5.5)$$

The x -momentum equation results from the substitution

$$\phi = u, \quad \Gamma^\phi = \frac{1}{Re}, \quad S^\phi = -\frac{\partial p}{\partial x} \quad (5.6)$$

and the y -momentum equation results from the substitution

$$\phi = v, \quad \Gamma^\phi = \frac{1}{Re}, \quad S^\phi = -\frac{\partial p}{\partial y} \quad (5.7)$$

The governing equations are discretized using the finite volume method (FVM) on a colocated grid arrangement. An attractive feature of this method is that the solution would satisfy the integral conservation of mass, momentum and other quantities over any group of control volumes and of course over the whole computational domain. Since the momentum equations are integrated over the colocated control volumes, Rhie and Chow [93] interpolation is used to ensure strong pressure-velocity coupling. To improve the guessed pressure, pressure-correction equation derived from the continuity equation is used. In order to overcome the instability that arises due to the central-difference scheme, higher-order Quadratic Upwind Interpolation for Convective Kinematics (QUICK) scheme is used to discretize the convective part of the governing equations and central-differences are used to compute the diffusive fluxes and the pressure gradient.

5.2.1 Discretization Method

In the cell-centred finite volume method used here the calculation domain is divided into a number of non-overlapping control volumes such that all flow variables ‘sit’ at the centre of the control volume. The differential equation integrated over each control volume yields a discretized equation at each cell-centre. The convective fluxes at the cell interfaces are computed using the QUICK scheme and the diffusive fluxes by central differences. The Quadratic Upwind Interpolation for Convective Kinematics (QUICK) scheme was devised by Leonard [74]. It uses the quadratic interpolation between two upstream neighbours and one downstream neighbour to evaluate ϕ (and hence flux) at the control volume interfaces.

The value of ϕ at the east face e of the control volume (see Fig. 5.1) is given by the quadratic interpolation.

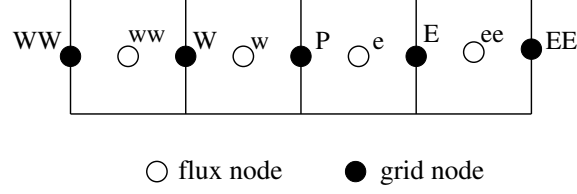


Fig. 5.1: Schematic diagram of uniform collocated grid for interpolation formula.

$$\phi_e = \frac{3}{8}\phi_E + \frac{3}{4}\phi_P - \frac{1}{8}\phi_W \quad \text{if } F_e > 0 \quad (5.8)$$

$$\phi_e = \frac{3}{8}\phi_P + \frac{3}{4}\phi_E - \frac{1}{8}\phi_{EE} \quad \text{if } F_e < 0 \quad (5.9)$$

where $F_e = u\Delta y$ represents the convection strength, which is positive for convection in the positive x -direction. Therefore the convective flux term can be expressed as

$$F_e\phi_e = \left(\frac{3}{8}\phi_E + \frac{3}{4}\phi_P - \frac{1}{8}\phi_W \right) [[0, F_e]] - \left(\frac{3}{8}\phi_P + \frac{3}{4}\phi_E - \frac{1}{8}\phi_{EE} \right) [[0, -F_e]] \quad (5.10)$$

where the operator $[[x, y]]$ denote the maximum of the two arguments x and y . The deferred QUICK scheme [48] employed in the present work is given by

$$F_e\phi_e = \left(\phi_P^n + \frac{1}{8}(3\phi_E - 2\phi_P - \phi_W)^{n-1} \right) [[0, F_e]] - \left(\phi_E^n + \frac{1}{8}(3\phi_P - 2\phi_E - \phi_{EE})^{n-1} \right) [[0, -F_e]] \quad (5.11)$$

where superscript n represents the latest value of the variable and $n - 1$ represents the value of the variable in the previous iteration.

5.2.2 Description of Multigrid Algorithm

In this section a step-by-step procedure used in the present work for solving incompressible N-S equations is given. The governing equations are discretized on a collocated grid, where all the dependent variables are stored at the centre of the same finite volume. The discretized

governing equation for any variable φ_m on a grid-level m is given by

$$[\mathcal{A}_m]\{\varphi_m\} = \{\mathcal{S}_m\} \quad (5.12)$$

where m denotes the grid level for the finest grid, $[\mathcal{A}_m]$ the coefficient matrix and $\mathcal{S}_m$ the source term vector. The present multigrid method, applied to a two-grid system, commences by performing a few finite volume iterations using the SIMPLE algorithm on the finest grid having a mesh size h . The residual R_m after some iterations is given by

$$\{R_m\} = \{S_m\} - [A_m]\{\phi_m\} \quad (5.13)$$

where $[A_m]$ and $\{S_m\}$ are approximations to $[\mathcal{A}_m]$ and $\{\mathcal{S}_m\}$ based on the approximate solution $\{\phi_m\}$. Subtracting Eq. (5.13) from Eq. (5.12) yields

$$[\mathcal{A}_m]\{\varphi_m\} = \{\mathcal{S}_m\} + [A_m]\{\phi_m\} - \{S_m\} + \{R_m\} \quad (5.14)$$

This equation is transferred to the next coarser grid level $(m+1)$ as follows

$$[\hat{A}_{m+1}]\{\hat{\phi}_{m+1}\} = \{\hat{S}_{m+1}\} + \underbrace{[\tilde{A}_{m+1}]\{\tilde{\phi}_{m+1}\} - \{\tilde{S}_{m+1}\} + \{\tilde{R}_{m+1}\}} \quad (5.15)$$

The variables and operators denoted by $(\hat{\ })$ are modified during the iteration on the coarser grid level $m+1$. The vectors that have been transferred from grid level m to $m+1$ are denoted by $(\tilde{\ })$ and are referred as the restricted quantities. The manner in which the variables are restricted will be explained in the next sub-section. The underlined terms remain unchanged during the iteration on the coarser grid and appear as an extra source term in the above equation. The quantities denoted by $(\tilde{\ })$ are used as initial guess for $(\hat{\ })$ quantities, which are changed owing to restricted residuals $\tilde{R}_{m+1}$ when iterating on the coarser grid. If the residual is zero, the solution will be $\hat{\phi}_{m+1} = \tilde{\phi}_{m+1}$, i.e., $\tilde{R}_{m+1} = 0$. It is to be noted that the pressure-correction equation is linear in pressure and therefore no restriction of pressure

from fine to coarse grid is needed. The SIMPLE smoothing scheme applied to the coarse grid equations is as follows:

1. The initial state is

$$\hat{u} = \tilde{u}, \quad \hat{v} = \tilde{v}, \quad p' = 0 \quad (5.16)$$

2. Perform a few iterations on the coarse-grid momentum equations.
3. The cell-face velocities, for example for the east and north faces, are corrected as follows

$$u'_e = \hat{u}_e - \tilde{u}_e + \Delta V_e \overline{\left(\frac{1}{A_P}\right)}_e \left[\overline{\left(\frac{\partial p'}{\partial x}\right)}_e - \left(\frac{\partial p'}{\partial x}\right)_e \right] \quad (5.17)$$

$$v'_n = \hat{v}_n - \tilde{v}_n + \Delta V_n \overline{\left(\frac{1}{A_P}\right)}_n \left[\overline{\left(\frac{\partial p'}{\partial y}\right)}_n - \left(\frac{\partial p'}{\partial y}\right)_n \right] \quad (5.18)$$

Similar corrections can be obtained for the other two faces. The above equations reflect Rhie and Chow [93] interpolation strategy required to secure freedom from odd-even oscillations in a colocated framework. The terms with over-bars are simply arithmetic averages of values of the nodes adjacent to the appropriate faces (P and E or P and N). The mass residuals on coarse grid based on these face velocities are calculated.

4. The face-velocity corrections (denoted by double primes) are derived from the truncated versions of Eqs. (5.17) and (5.18)

$$u''_e = a_e \overline{\left(\frac{1}{A_P}\right)}_e (p''_P - p''_E) \quad (5.19)$$

$$v''_n = a_n \overline{\left(\frac{1}{A_P}\right)}_n (p''_P - p''_N) \quad (5.20)$$

5. The coarse-grid pressure-correction equation is derived by substituting the face-velocity

corrections into the mass-continuity constraint. The pressure-correction equation is

$$\hat{A}_P p_P'' = \sum_{nb} \hat{A}_{nb} p_{nb}'' + R'_{nb} \quad (5.21)$$

where R'_{nb} indicates the mass unbalance.

6. Eq. (5.21) is solved, and the related pressure and face-velocity corrections are updated using the correction Eqs. (5.19) and (5.20).
7. Once the outer relaxation sweep is over, the algorithm returns to step 2.

After performing few relaxation sweeps, the coarse-grid solution is obtained. Hence the coarse-grid correction can be calculated by

$$[\delta\phi]^c = [\hat{\phi}] - [\tilde{\phi}] \quad (5.22)$$

however, if $\phi = p'$ (pressure correction), then

$$[\delta\phi]^c = [\hat{\phi}] \quad (5.23)$$

The coarse-grid corrections are interpolated back (prolongated) to the fine-grid to adjust the fine-grid solutions.

$$[\delta\phi]^f = [I_c^f][\delta\phi]^c \quad (5.24)$$

where I_c^f is the prolongation operator.

These are added to the prevailing variable values

$$[\phi]_{new}^f = [\phi]_{old}^f + \lambda[\delta\phi]^f \quad (5.25)$$

where λ is an under-relaxation factor $0 \leq \lambda \leq 1$, which may be necessary to ensure convergence of the two-level cycles. In the fine-grid again a few iterations are carried out to

smoothen the high-frequency errors arising from the interpolation.

Restriction

In a cell-centre finite volume algorithm no nodes on the fine-grid ever coincide with those of the coarse-grid whether staggered or colocated and therefore the transfer of variables from the fine to the coarse-grid has to be performed by interpolation. The transfer of residuals requires no interpolation because the coarse-grid balance equation is equal to the sum of the four balance equations for the corresponding fine-grid control volume. The grid arrangement shown in Fig. 5.2 can be divided into three regions like the inner region 'a', boundary region ('b' shows west) and corner ('c' shows the north-west). The restriction operator I_f^c can be

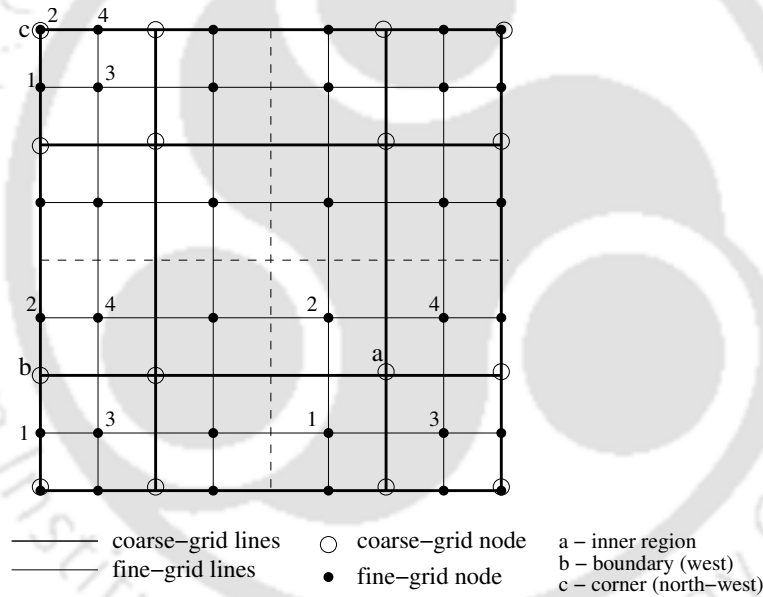


Fig. 5.2: (a) Restriction of mass fluxes and variables.

expressed as

$$\tilde{\phi} = \phi^c = [I_f^c] \begin{bmatrix} \phi^1 \\ \phi^2 \\ \phi^3 \\ \phi^4 \end{bmatrix}^f \quad (5.26)$$

For the inner region away from the boundaries,

$$[I_f^c] = \frac{1}{4} \begin{bmatrix} 1 & 1 & 1 & 1 \end{bmatrix} \quad (5.27)$$

while for the west boundary,

$$[I_f^c] = \frac{1}{4} \begin{bmatrix} 2 & 2 & 0 & 0 \end{bmatrix} \quad (5.28)$$

and for the north-west corner,

$$[I_f^c] = \frac{1}{4} \begin{bmatrix} 0 & 4 & 0 & 0 \end{bmatrix} \quad (5.29)$$

The above equations represent bilinear interpolation formulae. The restriction operator pertaining to the residuals is

$$\tilde{R}_\phi = R_\phi^c = [I_f^c] \begin{bmatrix} R_\phi^1 \\ R_\phi^2 \\ R_\phi^3 \\ R_\phi^4 \end{bmatrix}^f \quad (5.30)$$

where $[I_f^c] = \begin{bmatrix} 1 & 1 & 1 & 1 \end{bmatrix}$.

Prolongation

The grid arrangement can be divided into three parts: A- the inner region, B - the boundary strips and C - the boundary corners (Fig. 5.3). The prolongation operator I_c^f can be expressed

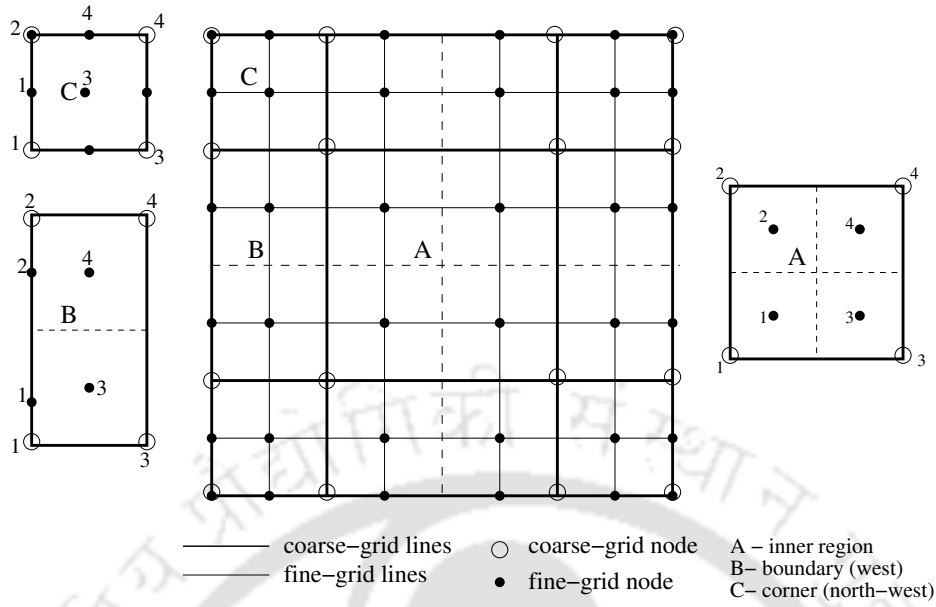


Fig. 5.3: Grid stencil for prolongation operator.

as

$$\begin{bmatrix} \delta\phi^1 \\ \delta\phi^2 \\ \delta\phi^3 \\ \delta\phi^4 \end{bmatrix}^f = [I_c^f] \begin{bmatrix} \delta\phi^1 \\ \delta\phi^2 \\ \delta\phi^3 \\ \delta\phi^4 \end{bmatrix}^c \quad (5.31)$$

The nature of the prolongation operator depends on the region under consideration. It has the following forms:

For the inner region:

$$[I_c^f] = \frac{1}{16} \begin{bmatrix} 9 & 3 & 3 & 1 \\ 3 & 9 & 1 & 3 \\ 3 & 1 & 9 & 3 \\ 1 & 3 & 3 & 9 \end{bmatrix}, \quad (5.32)$$

for the west boundary:

$$[I_c^f] = \frac{1}{16} \begin{bmatrix} 0 & 0 & 6 & 2 \\ 0 & 0 & 2 & 6 \\ 3 & 1 & 3 & 1 \\ 1 & 3 & 1 & 3 \end{bmatrix}, \quad (5.33)$$

and for north-west corner:

$$[I_c^f] = \frac{1}{16} \begin{bmatrix} 0 & 0 & 2 & 2 \\ 0 & 0 & 0 & 4 \\ 1 & 1 & 1 & 1 \\ 0 & 2 & 0 & 2 \end{bmatrix} \quad (5.34)$$

Similar relations can be applied to the other boundaries and corners.

5.3 Results and Discussion

5.3.1 Application to Lid-Driven Cavity Flow

A computer code has been developed using the QUICK scheme for convective terms and central-difference scheme for the viscous terms. In order to evaluate the performance of the code, it has been applied to a standard test problem, namely, 2D single-sided lid-driven cavity, where the fluid in a square cavity is set into motion by the movement of one wall. Since steady equations are solved using the SIMPLE algorithm in the present work, the developed code captures only the final steady-state results. The geometry and boundary conditions are shown in Fig. 3.1, where the top wall is moving from the left to right and the remaining three walls are stationary. No-slip boundary condition is employed on all the four

walls. The dimensionless boundary conditions are given as

$$x = 0 : u = 0.0, \quad v = 0.0$$

$$x = 1 : u = 0.0, \quad v = 0.0$$

$$y = 0 : u = 0.0, \quad v = 0.0$$

$$y = 1 : u = 1.0, \quad v = 0.0$$

Fig. 5.4 presents a comparison of the steady-state horizontal velocities on the vertical centreline and the vertical velocities on the horizontal centreline of the square cavity with those obtained by Ghia *et al.* [41] and Erturk *et al.* [32] and the agreement is quite good. The steady-state streamline patterns obtained with the present code for $Re = 1000, 3200, 5000$ and 7500 are shown in Fig. 5.5, which agree very well with those in [32,41]. It is seen that the results produced by the present FAS multigrid code are accurate and the code stands validated.

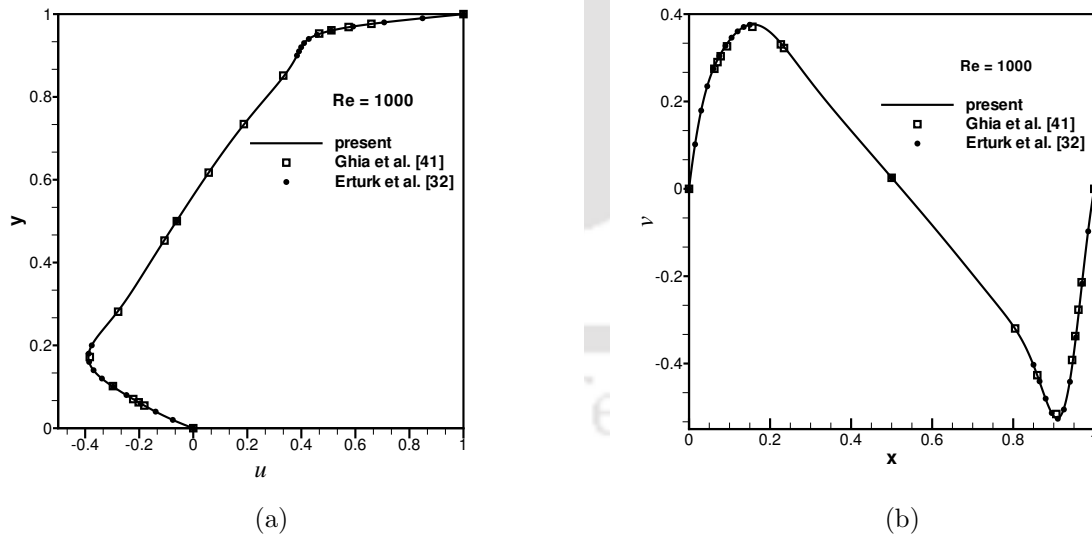


Fig. 5.4: Comparison of steady-state velocity profiles (a) horizontal velocity along the vertical centreline and (b) vertical velocity along the horizontal centreline for the single-sided lid-driven cavity.

Now we carry out a comparison exercise to evaluate the performance of the 4-level multigrid versus the single-grid method. Expectedly full-approximation storage multigrid method

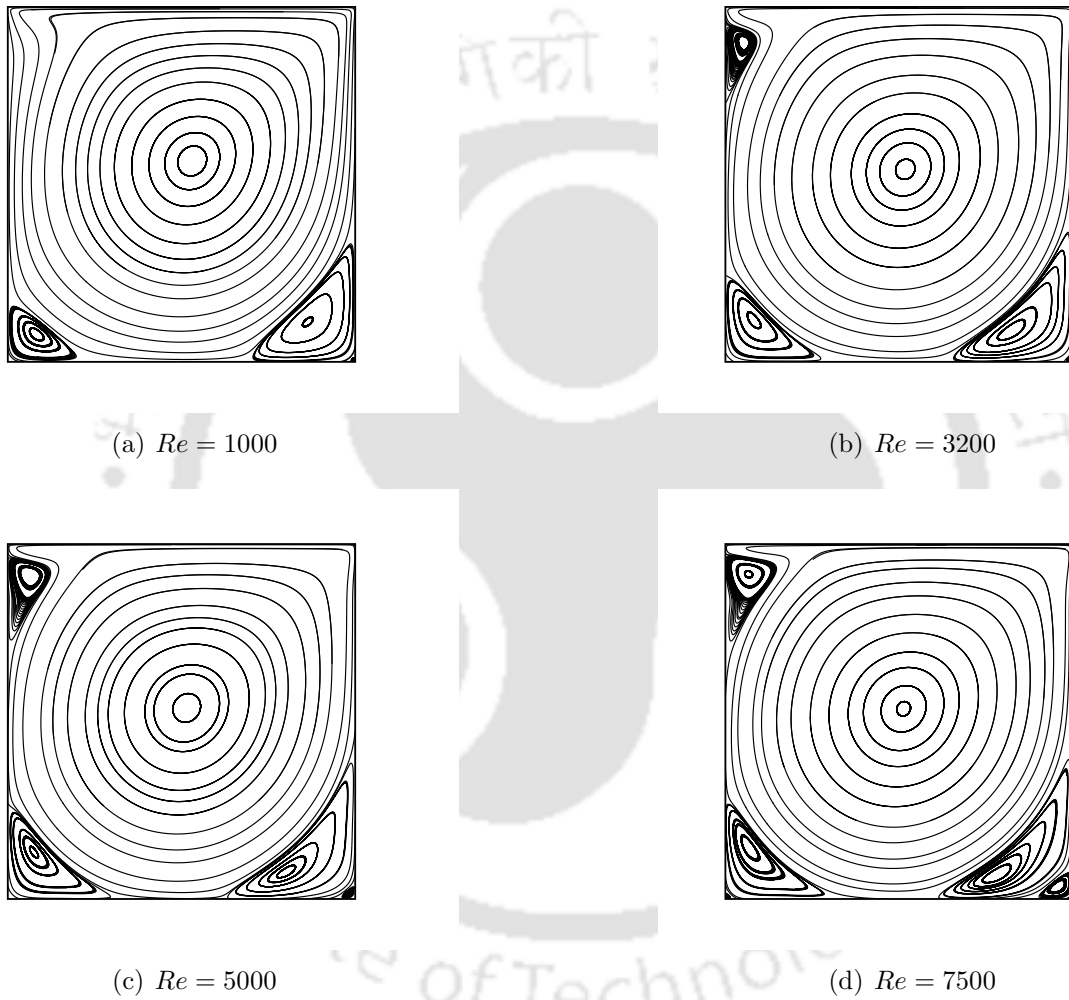


Fig. 5.5: Steady state streamline patterns for the single-sided lid-driven cavity for various Reynolds numbers.

shows a faster convergence. All the results presented here are produced using a 4-level V-cycle strategy (Fig. 5.6) and increasing the level further does not pay any additional dividend. The number of iterations performed at each grid level is shown inside the circles (Fig. 5.6). The computational effort is reported in terms of equivalent fine-grid sweeps that

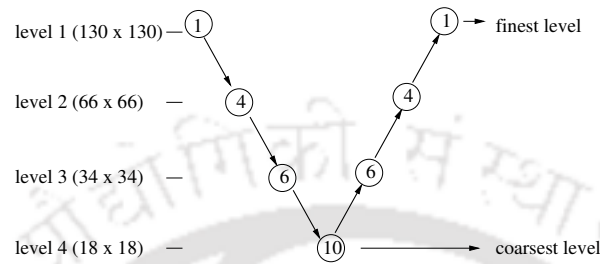


Fig. 5.6: V-cycle multigrid showing iterations at various levels.

are usually referred to as work units. Fig. 5.7 gives the comparison between 4-level multigrid and single-grid for the history of mass residual on a 130×130 grids. Expectedly a 4-level multigrid cycle shows a faster convergence than single grid. Fig. 5.7 shows that in the single-grid the residuals fall faster at the beginning and slows down thereafter whereas in multigrid method the residual falls at a more or less constant rate. It is observed that work units required to reach the steady state increase as Re increases for both single grid and multigrid. This can be attributed to the fact that high Reynolds number flows contain multiplicity of scales, which introduce high frequency errors into the computational process. Table 5.1 summarizes the total CPU time required and speed-up obtained by multigrid for various Reynolds numbers. For the same fall of residual, time-gain by multigrid is impressive. The time-wise speed-up achieved by multigrid at ‘steady state’ is approximately 8 to 10 times for the various Reynolds numbers. All the CPU times given in Table 5.1 correspond to the computations carried out on a Dual Xeon 3.2 GHz based machine. Thus, in terms of computational efficiency, the 4-level multigrid method is substantially faster than the single grid method.

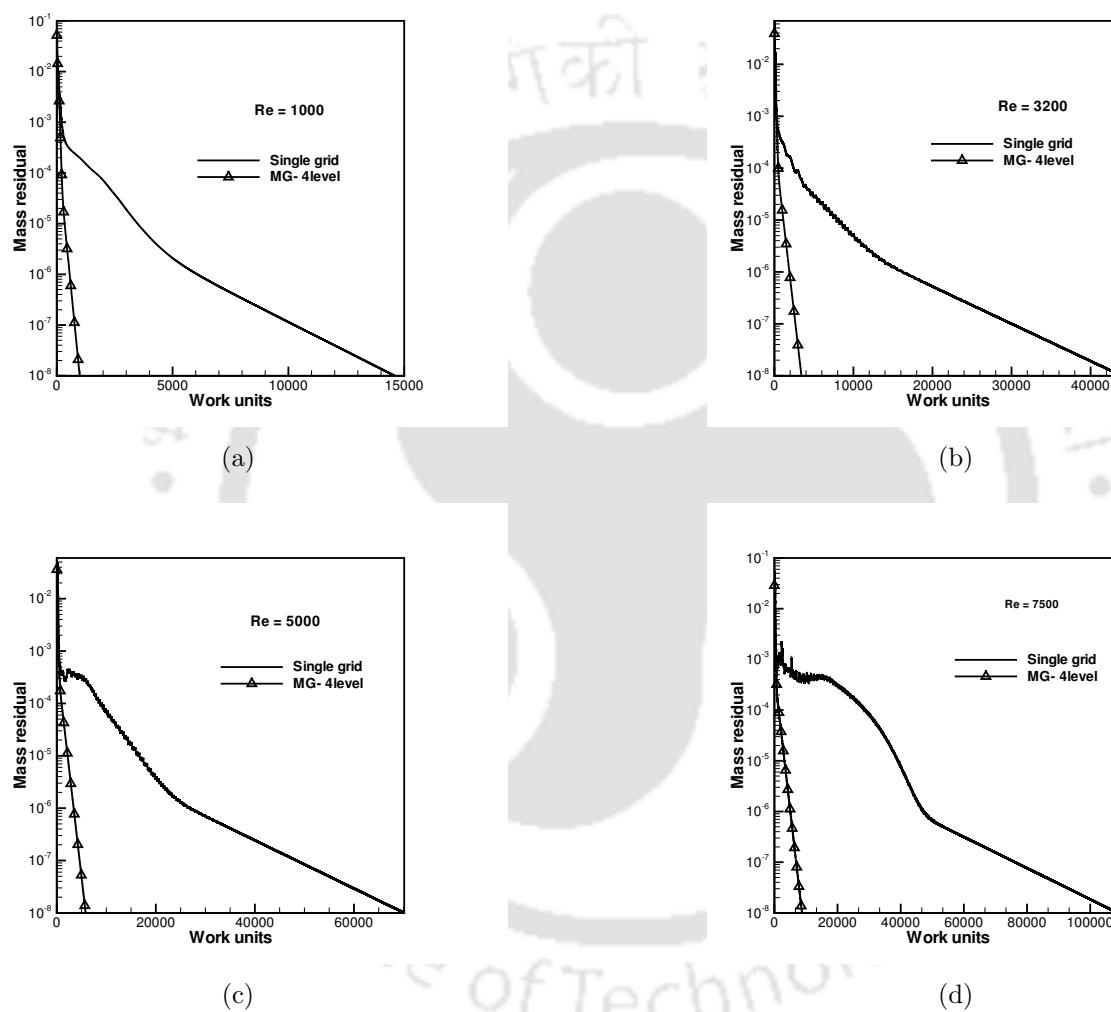


Fig. 5.7: Mass residual history of multigrid methods for single-sided lid-driven cavity for various Reynolds numbers

Table 5.1: Performance of multigrid method for a single-sided lid-driven cavity flow at various Reynolds numbers.

Reynolds number	CPU time (minutes)		Speed-up
	Single-grid	MG 4-level	
1000	26.12	2.30	10.48
3200	64.24	8.06	7.95
5000	104.48	13	8.06
7500	142.30	15.48	9.01

5.3.2 Multiple Solution for Two- and Four-Sided Lid-Driven Cavity Flows

Two-Sided Non-Facing Lid-Driven Cavity Flow

The flow configuration for the two-sided non-facing lid-driven cavity is shown in Fig. 5.8. The dimensionless boundary conditions are given as

$$x = 0 : u = 0.0, \quad v = -1.0$$

$$x = 1 : u = 0.0, \quad v = 0.0$$

$$y = 0 : u = 0.0, \quad v = 0.0$$

$$y = 1 : u = 1.0, \quad v = 0.0$$

Here the flow is driven by the upper wall ($y = 1$) moving from the left to right and the left wall moving downwards. The top wall drags the fluid to the right and this fluid then gets deflected by the right wall that makes the fluid rotate in the clockwise direction. Similarly the left wall drags the fluid to the bottom and it is deflected by the bottom wall that makes the fluid rotate in the counterclockwise direction. Apart from the two counter-rotating primary vortices, two secondary vortices are formed with the resulting flow field being symmetric

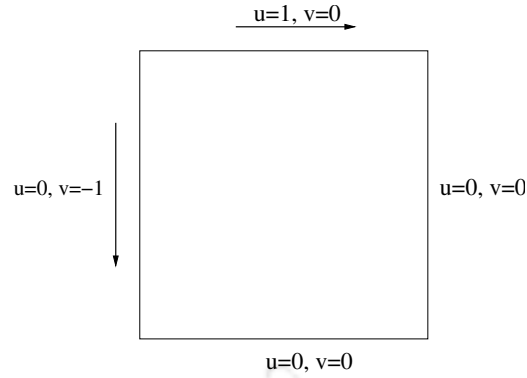


Fig. 5.8: Schematic diagram of the two-sided lid-driven cavity.

about the diagonal passing through the point where the moving walls meet. As Re increases to a critical value and beyond, the flow produces multiple symmetric and asymmetric steady solutions that depend upon the sweeping direction in the line-by-line iterative solver. Fig. 5.9 shows a few streamline patterns for various Reynolds numbers. At lower Reynolds numbers (Fig. 5.9(a-c)) the streamline patterns are symmetric about a diagonal and as the Reynolds number increases, the size of the secondary vortices grow in size at the expense of the primary vortices. As seen from Fig. 5.9(d-f) at larger Reynolds numbers asymmetry may develop. Recently Wahba [113] showed that between Reynolds number values of 1071 and 1075, the flow bifurcates from a stable symmetric state to a stable asymmetric state and the critical Reynolds number for flow bifurcation in two-sided non-facing lid-driven cavity is 1073. Fig. 5.9(d) shows that at $Re = 1075$ there is a departure from symmetry with the upper vortices becoming slightly smaller than the corresponding lower ones if the sweeping direction in the line-by-line iterative solver is from the left to right and our result is consistent with those reported in [113]. This trend continues with the increase in Re with the result that at $Re = 1500$ and 2000 (Fig. 5.9(e-f)) the upper vortices are considerably larger than the corresponding lower ones. If the sweeping direction is reversed the resulting asymmetric flow patterns have upper vortices larger than the corresponding lower ones. Fig. 5.10 depicts the pressure contours corresponding to the flow patterns given in Fig. 5.9. As for flow patterns, it is seen that as Re increases symmetry in pressure is lost and the effect is more prominent at larger Reynolds numbers. The pressure is the maximum in the neighbourhood

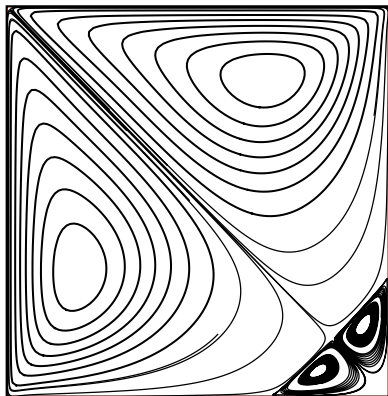
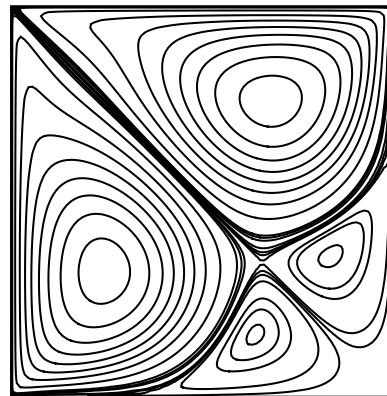
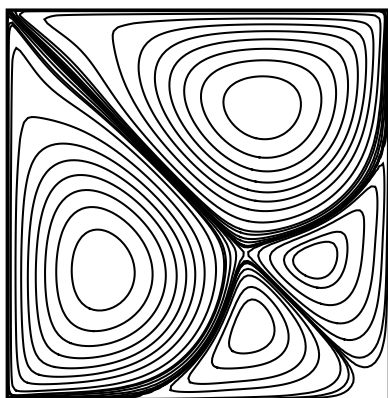
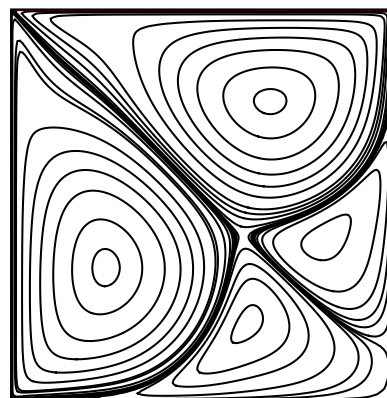
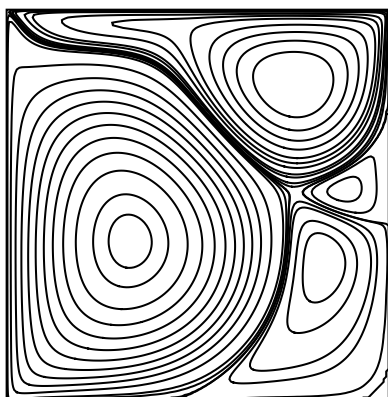
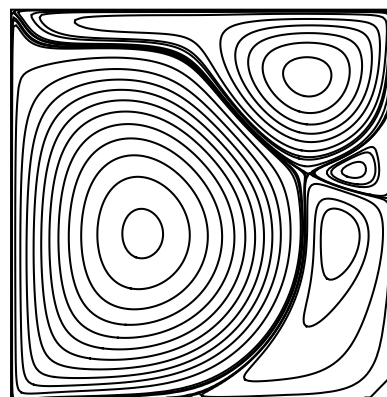
(a) $Re = 100$ (b) $Re = 500$ (c) $Re = 1000$ (d) $Re = 1075$ (e) $Re = 1500$ (f) $Re = 2000$

Fig. 5.9: Steady state streamline patterns for the two-sided lid-driven cavity for different Reynolds numbers.

of the top-right and bottom-left corners as at these locations, for the type of wall motions considered here, the fluid flow impinges on the facing walls with the maximum velocity. Fig. 5.11 shows the multiple-steady flow patterns with one symmetric and two asymmetric solutions at $Re = 2000$. The first asymmetric solution (Fig. 5.11(a)) is obtained when the line-by-line solver sweeps from the left to right. If the direction of sweep is reversed another asymmetric solution (Fig. 5.11(b)) is obtained. That both these asymmetric patterns are valid stable solutions to the problem can be gauged from the fact that there is no change of state if the final solution obtained by sweeping in one direction is given as the input to the solver sweeping in the other direction. The flow geometry clearly suggests the existence of a symmetric solution as well. Fig. 5.11(c) shows this flow pattern, which is obtained by putting $u = -v$ on the symmetry diagonal from the beginning to the point when the mass residual reaches a value of 10^{-3} so as to help the flow develop in the direction that favours the desired solution. After this artificial restriction is removed to obtain a valid solution, the mass residual is allowed to fall below 10^{-8} when Fig. 5.11(c) is plotted. Fig. 5.12 depicts the pressure contours for the multiple solutions obtained for $Re = 2000$.

Four-Sided Lid-Driven Cavity Flow

A schematic representation of the four-sided lid-driven cavity is shown in Fig. 5.13. The dimensionless boundary conditions are given by

$$x = 0 : \quad u = 0.0, \quad v = -1.0$$

$$x = 1 : \quad u = 0.0, \quad v = 1.0$$

$$y = 0 : \quad u = -1.0, \quad v = 0.0$$

$$y = 1 : \quad u = 1.0, \quad v = 0.0$$

Here the upper and the lower walls move to the right and left respectively, while the left wall moves downwards and the right wall upwards. All the four walls move with equal speeds. The top wall drags the fluid to the right and this fluid is then deflected by the right wall

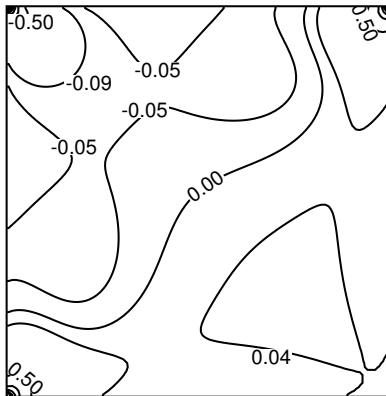
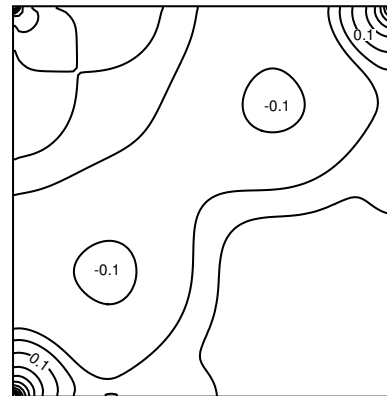
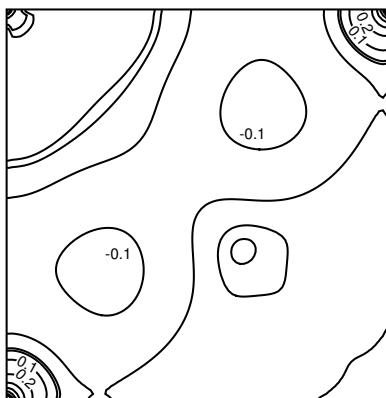
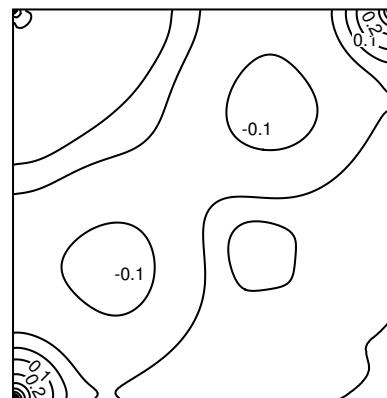
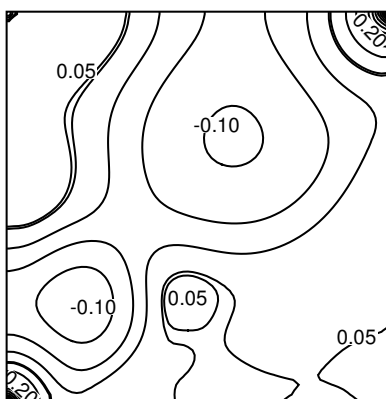
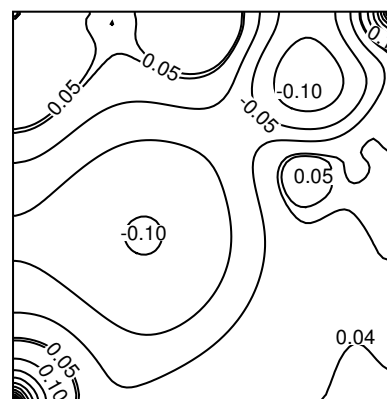
(a) $Re = 100$ (b) $Re = 500$ (c) $Re = 1000$ (d) $Re = 1075$ (e) $Re = 1500$ (f) $Re = 2000$

Fig. 5.10: Steady state pressure contours for the two-sided lid-driven cavity for different Reynolds numbers.

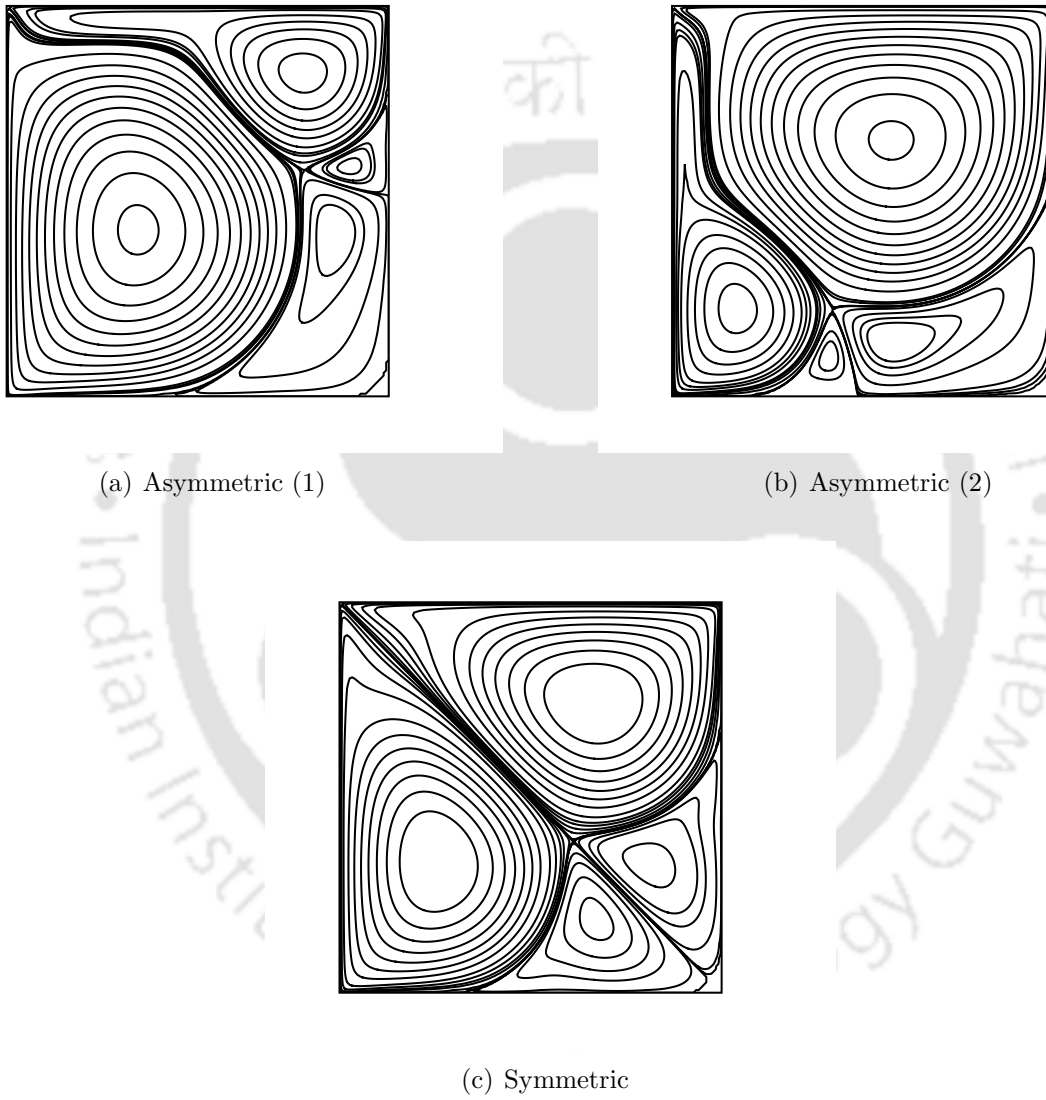
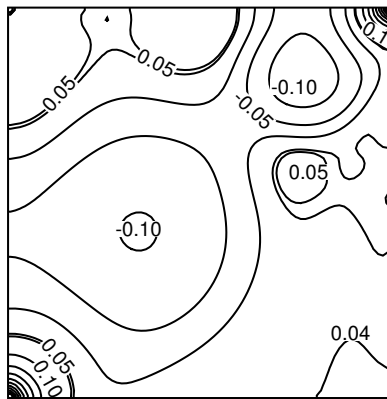
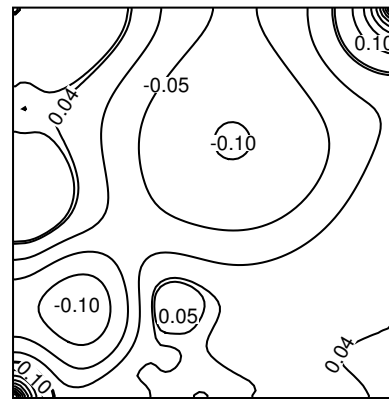


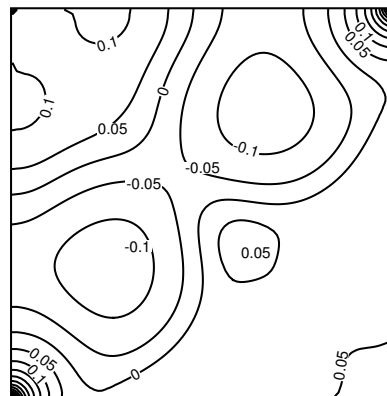
Fig. 5.11: Multiple solutions for a two-sided lid-driven cavity at $Re = 2000$.



(a) Asymmetric (1)



(b) Asymmetric (2)



(c) Symmetric

Fig. 5.12: Multiple solutions of pressure contours for a two-sided lid-driven cavity at $Re = 2000$.

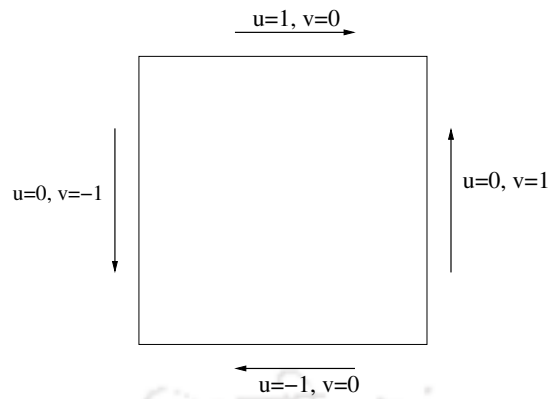


Fig. 5.13: Schematic diagram of the four-sided lid-driven cavity.

that makes the fluid rotate in the clockwise direction. The right wall drags the fluid upwards and the fluid is then deflected by the top wall so that it rotates in the counterclockwise direction. Similarly, the bottom wall moving to the left and the left wall moving downwards make the dragged fluid rotate in the clockwise and counterclockwise directions respectively. Since all the four walls move with equal speed, four distinct vortices of similar size develop symmetric to both the diagonals. Fig. 5.14 shows a few streamline patterns for various Reynolds numbers. At $Re = 100$ and $Re = 127$ (Fig. 5.14(a-b)) the streamline patterns are symmetric about both the diagonals. As seen from Fig. 5.14(c-f) at larger Reynolds number asymmetry may develop. For a four-sided lid-driven cavity Wahba [113] showed that between Reynolds number values of 127 and 131, the flow bifurcates from a stable symmetric state to a stable asymmetric state and the critical Reynolds number for bifurcation in four-sided lid-driven cavity is 129. Fig. 5.14(c) shows that at $Re = 131$ there is a departure from symmetry with the upper and lower vortices showing a tendency to merge into one if the line-by-line iterative solver sweeps vertically and our result is consistent with those reported in [113]. The merging process continues with the increase in Re as can be seen from Fig. 5.14(d-f). Fig. 5.14(f) shows a state when the merging results in a single vortex with a single core. This vortex is considerably larger than the right and left vortices. If a horizontal line-by-line sweep is employed instead of a vertical sweep the resulting asymmetric flow patterns would have merged right and left vortices and the resultant vortex would be larger than the upper

and lower vortices. Fig. 5.15 depicts the pressure contours corresponding to the flow patterns given in Fig. 5.14. As for flow patterns, it is seen that as Re increases symmetry in pressure pattern is lost and the effect is more prominent at larger Reynolds numbers. Fig. 5.16 shows the multiple-steady flow patterns with one symmetric and two asymmetric solutions at $Re = 200$. The first asymmetric solution (Fig. 5.16(a)) is obtained when the line-by-line solver sweeps from the left to right. If the direction of sweep is changed to vertical another asymmetric solution (Fig. 5.16(b)) is obtained. That both these asymmetric patterns are valid stable solutions to the problem can be gauged from the fact that there is no change of state if the final solution obtained by sweeping in one direction is given as the input to the solver sweeping in the other direction. The flow geometry clearly suggests the existence of a symmetric solution as well. Fig. 5.16(c) shows this flow pattern, which is obtained by putting $u = -v$ on the symmetry diagonal from the top-left to the bottom-right corner and putting $u = v$ on the symmetry diagonal from the bottom-left to the top-right corner till when the mass residual reaches a value of 10^{-3} so as to help the flow develop in the direction that favours the desired solution. After this artificial restriction is removed to obtain a valid solution, the mass residual is allowed to fall below 10^{-8} when Fig. 5.16(c) is plotted. Fig. 5.17 depicts the corresponding pressure contours for the multiple solutions obtained for $Re = 200$.

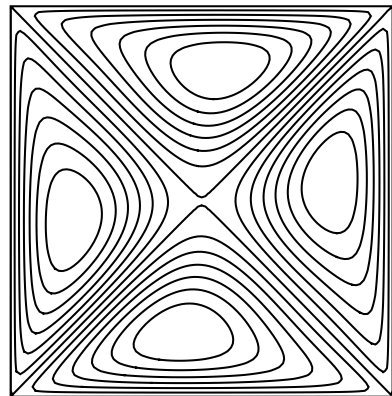
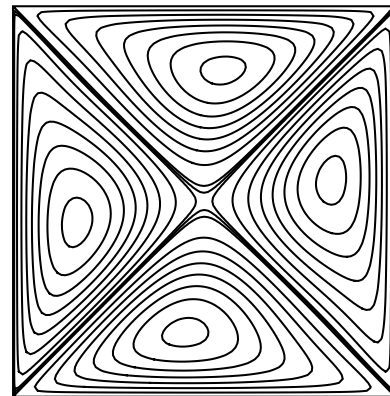
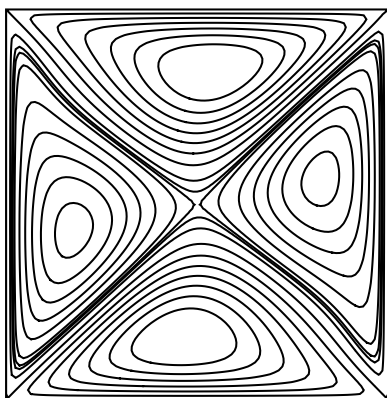
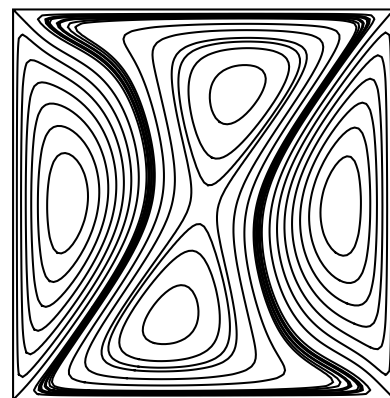
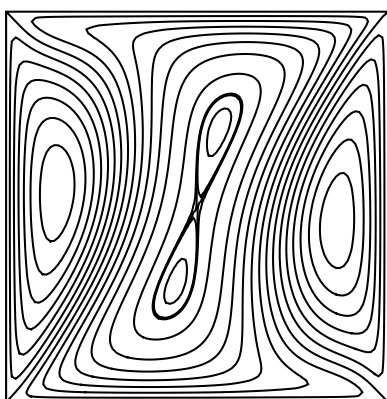
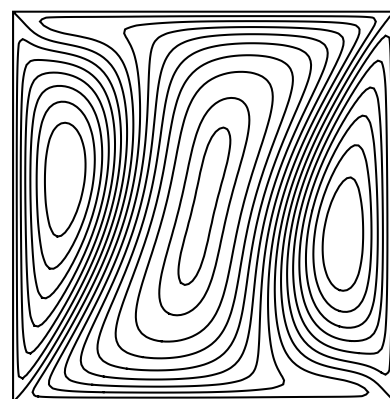
(a) $Re = 100$ (b) $Re = 127$ (c) $Re = 131$ (d) $Re = 150$ (e) $Re = 200$ (f) $Re = 300$

Fig. 5.14: Steady state streamline patterns for the four-sided lid-driven cavity for different Reynolds numbers.

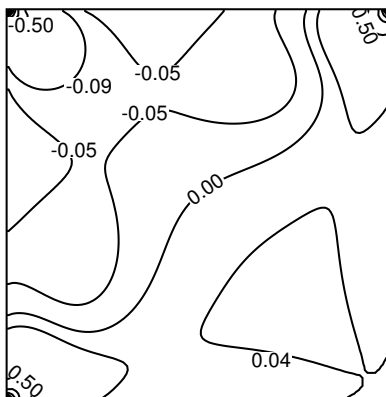
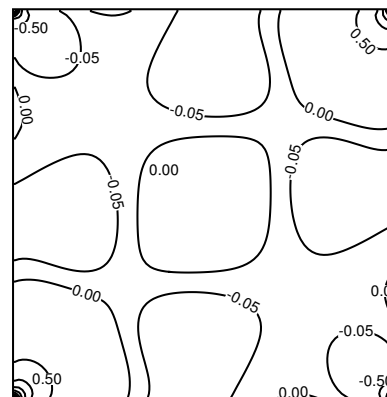
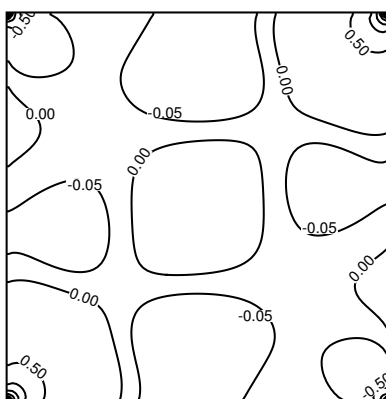
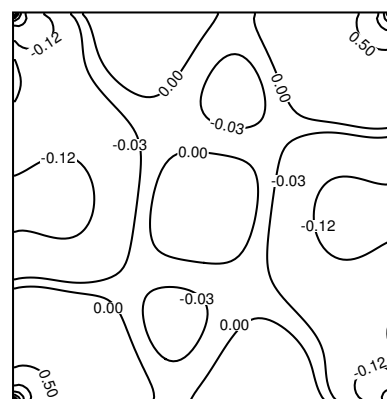
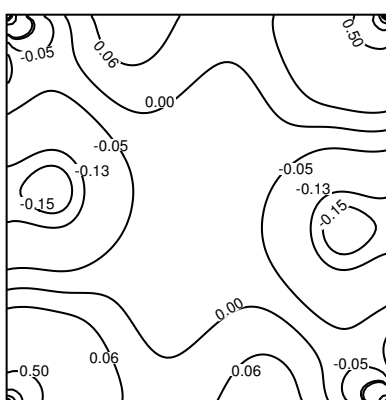
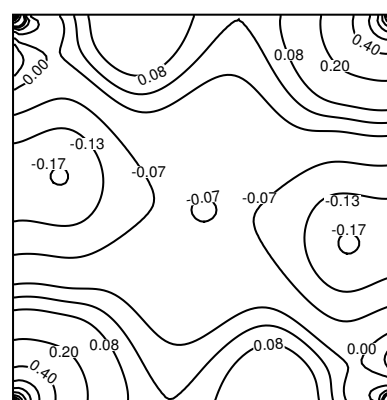
(a) $Re = 100$ (b) $Re = 127$ (c) $Re = 131$ (d) $Re = 150$ (e) $Re = 200$ (f) $Re = 300$

Fig. 5.15: Steady state pressure contours for the four-sided lid-driven cavity for different Reynolds numbers.

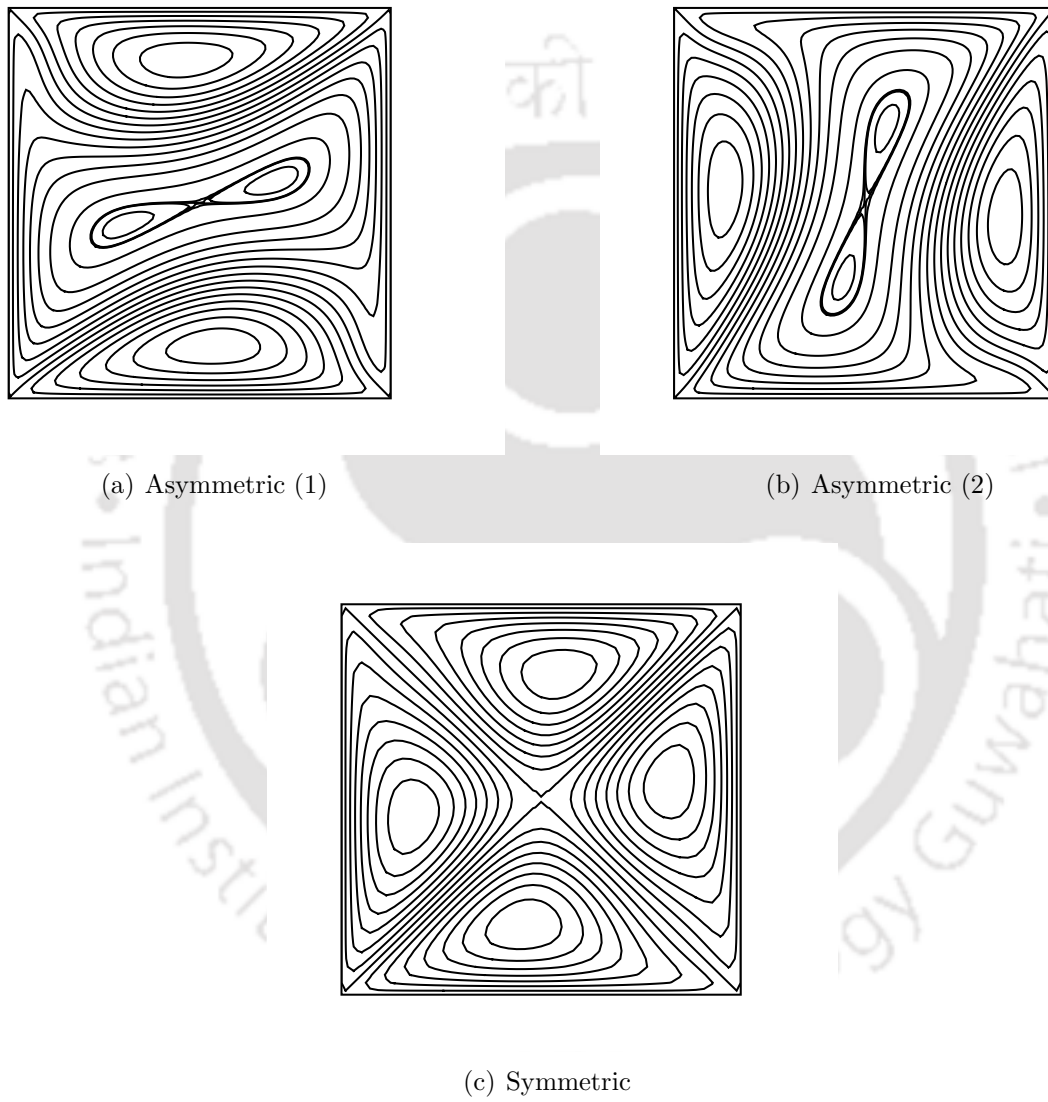
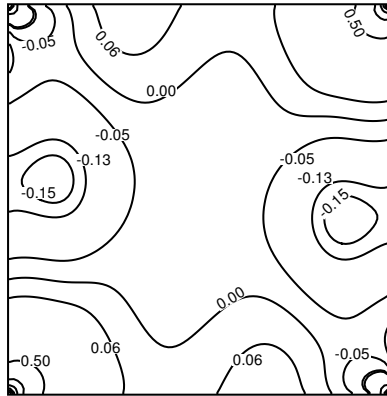
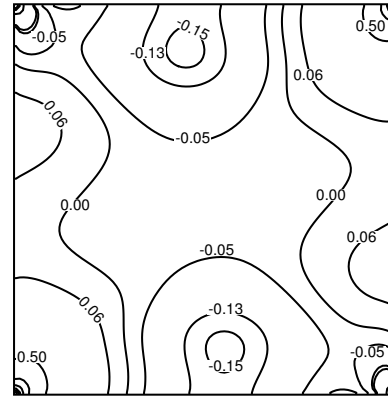


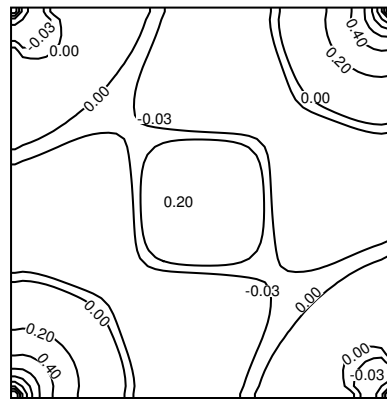
Fig. 5.16: Multiple solutions for a four-sided lid-driven cavity at $Re = 200$.



(a) Asymmetric (1)



(b) Asymmetric (2)



(c) Symmetric

Fig. 5.17: Multiple solutions of pressure contours for a four-sided lid-driven cavity at $Re = 200$.

5.3.3 Multigrid Performance

Now we carry out a comparison exercise to evaluate the performance of the 4-level multigrid versus the single-grid method for two-sided and four-sided lid-driven cavities. Fig. 5.18 gives for the two-sided lid-driven cavity the comparison between 4-level multigrid and single-grid for the history of mass residual on a 130×130 grid. Fig. 5.19 gives the same for the four-

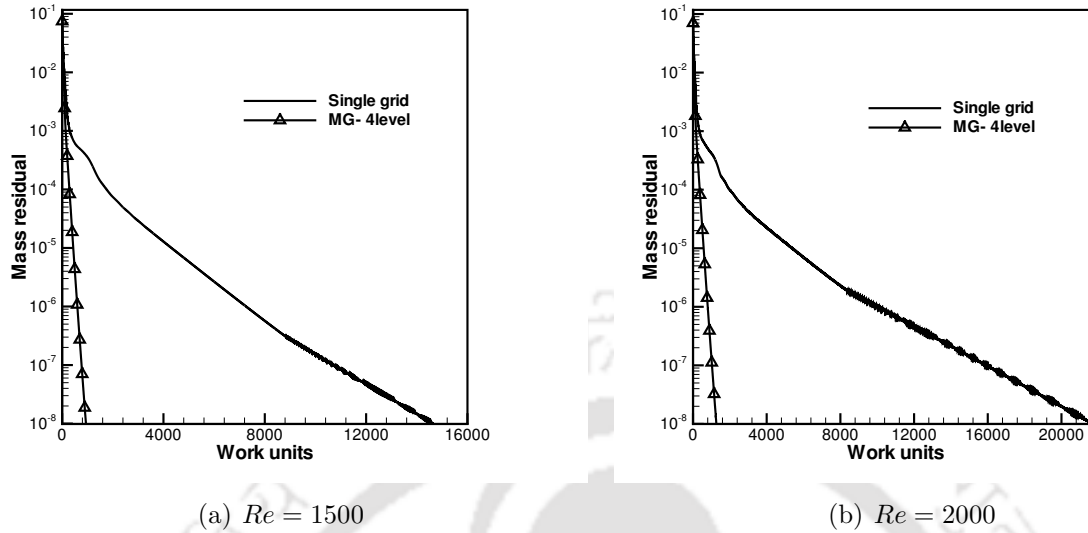


Fig. 5.18: Mass residual history of multigrid methods for a two-sided lid-driven cavity.

sided lid-driven cavity. Expectedly a 4-level multigrid cycle shows a faster convergence than a single grid. As for single lid-driven cavity, the residuals in the two- and four-sided cavities falls faster at the beginning and slows down thereafter whereas in the multigrid method the residual falls at a more or less constant rate. Tables 5.2 and 5.3 summarize the total CPU time required and speed-up obtained by the multigrid for 2-sided and 4-sided cavity flows, respectively. It can be seen that the time-wise speed-up achieved by the multigrid is approximately 9 to 10 times. Thus in terms of computational efficiency, the multigrid method is substantially faster than the single-grid method. It may be noted that multigrid pay-off is likely to be higher for finer grids. These tables exemplify the excellent potential of the multigrid technique for accelerating the convergence even for flow geometries giving multiple solutions.

5.4 Conclusions

A full-approximation storage (FAS) multigrid code using the finite volume method and SIMPLE algorithm on a colocated grid arrangement is developed to obtain stable multiple solutions for two- and four-sided lid-driven cavities. Since the convective terms in the Navier-

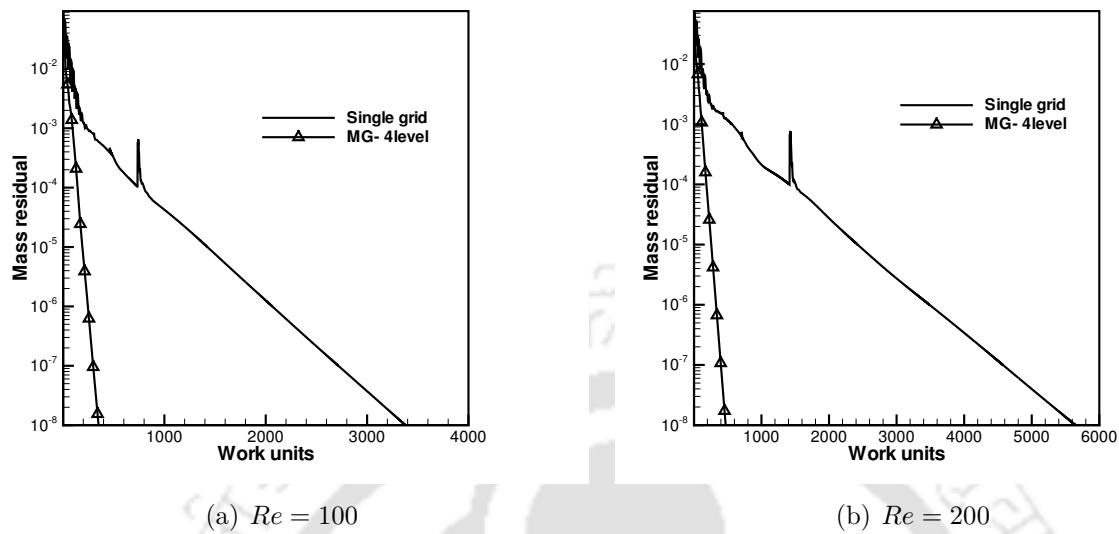


Fig. 5.19: Mass residual history of multigrid methods for a four-sided lid-driven cavity.

Table 5.2: Performance of multigrid method for a two-sided lid-driven cavity flow at various Reynolds numbers

Reynolds number	CPU time (minutes)		Speed-up
	Single-grid	MG 4-level	
1500	32.30	3.12	10.15
2000	45.6	4.12	10.73

Table 5.3: Performance of multigrid method for a four-sided lid-driven cavity flow at various Reynolds numbers

Reynolds number	CPU time (minutes)		Speed-up
	Single-grid	MG 4-level	
100	17.12	1.48	9.55
200	20.12	2.12	9.18

Stokes equations are discretized using the QUICK scheme and the viscous terms are central-differenced to second-order accuracy, the code offers accurate space resolution. To establish the credibility and performance of the code it is first used to compute the flow in a standard 2D single-lid-driven cavity to demonstrate that the results closely match with highly reliable existing results. Ability of the 4-level multigrid method to obtain speed-up of 8 to 10 compared with the single-grid method exhibits the good time-wise performance of the code. After having thus gained an insight into various aspects of the code it is then used to compute the flows in two- and four-sided 2D lid-driven cavities. These two flow configurations are fraught with many interesting flow features in that they exhibit multiple stable solutions. How these multiple solutions are obtained for the same flow configuration is described in some detail. That the solutions are indeed valid and stable is established. It may be noted that nonlinear problems are known at times to give multiple solutions and the traditional mathematical concept of wellposedness does not apply here. For the first time a multigrid technique is used to compute the solutions for such peculiar fluid-mechanical situations with substantial time-wise acceleration of convergence.

Chapter 6

Analysis of Non-Darcy Models for Mixed Convection in a Porous Cavity using Multigrid Approach

6.1 Introduction

This chapter investigates the performance of two models, namely Brinkman-extended Darcy model (BDM) and Brinkman-Forchheimer Darcy model (BFDM) in a problem involving mixed convection in a square cavity filled with a porous medium using FAS multigrid method. The analysis of convection heat transfer in porous media has attracted the attention of many researchers during the past three decades. This type of flow is encountered in geothermal energy systems, prevention of sub-oil water pollution, storage of nuclear waste, separation process in chemical industries etc. Early works on fluid flow and heat transfer through porous media have used the Darcy law [26] which neglects inertia and viscous effects which are not significant for a low velocity and low permeability flows. However, the Darcy law does not hold good for high velocity and high permeability flows. To overcome the shortcomings of the Darcy's equation, two notable modifications are proposed namely, the Forchheimer's equation [36] and Brinkman's equation [15]. The former modification considers nonlinear

drag effect due to the solid matrix which should be considered when Reynolds number or Darcy number is large and the latter accounts for the viscous stresses adjacent to the bounding walls and this equation is valid only when the porosity is high. Brinkman-Forchheimer equation was derived by the local volume averaging technique that includes the inertia and viscous terms [82].

Some investigators have used combinations of these models. Vafai and Tien [109] highlighted the importance of boundary layer study and inertia effects in forced convection through porous medium using Brinkman-Forchheimer Darcy model (BFDM) without the divergent inertial term and it was pointed out that in some situations it should not be neglected. Lauriat and Prasad [72] accounted for nonlinear drag by the Forchheimer approach and studied the relative importance of inertia and viscous forces on natural convection in porous media. Lage [71] and Jue [54] analyzed Benard convection in porous media via BFDM. Recently Roy [9] studied natural convection in porous media with various thermal boundary conditions using Darcy-Forchheimer model without Forchheimer inertia term for a wide range of Rayleigh and Prandtl numbers.

Mixed convection with heat transfer in a porous medium via Brinkman-extended Darcy model (BDM) was numerically investigated and the effects of Richardson and Darcy numbers were discussed in [62, 112]. Al-Amiri [1] analyzed the momentum and energy transfer under stable thermal stratification with emphasis on the influence of the quadratic inertial effects. Khanafer and Vafai [63] employed BFDM in a double-diffusive mixed convection lid-driven cavity filled with porous medium by the finite volume approach. They discussed various pertinent dimensionless numbers in detail.

In a mixed convection flows, Richardson number (Ri) (expressed as $\frac{Gr}{(Re)^2}$) represents the importance of natural convection relative to the forced convection and viceversa. Here Gr denotes Grashof number and Re the Reynolds number. When $Ri < 0.1$, the flow due to natural convection is negligible. Similarly, when $Ri > 10$, the flow due to forced convection is negligible. In the present case we analysed the two models, namely BFDM and BDM, for $Ri = 0.01$ in which the forced convection becomes the dominated parameter. After gaining

confidence that BFDM produces better results than BDM, we carry out for $Ri = 100$, where natural convection becomes the dominating parameter using BFDM. The objective of the present work is to investigate through a multigrid method the effects of the nonlinear inertial term in the BFDM which is absent in the BDM on the mixed convection in a lid-driven cavity filled with porous medium. The study has been conducted for different values of Darcy number and Grashof number to show the true effects on momentum and heat transfer inside the cavity. To enhance the computational efficiency, the FAS-multigrid method was incorporated in the present work. The present work constitutes an efficient method for computation of flow for mixed convection in a porous cavity.

6.2 Problem Description

The flow configuration considered in the present investigation is shown in Fig. 6.1. A 2D square cavity of height H is filled with a saturated porous medium whose horizontal walls are insulated. The cold left wall is moving up and the hot right wall is moving down with equal velocities. The sliding lids induce a clockwise fluid motion, whereas the buoyancy force tries to rotate the fluid in the counter-clockwise direction. The directions of movement of the vertical plates are so chosen as to oppose the current due to pure buoyancy so that interesting flow patterns develop. The flow is assumed to be incompressible, laminar and steady, and the porous medium is taken as homogeneous with isotropic properties. The fluid physical properties are considered constant except the density in the body force term in the momentum equation for which the Boussinesq approximation is invoked. Under these assumptions the nondimensional equations governing the conservation of mass, momentum and energy can be written as follows:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \quad (6.1)$$

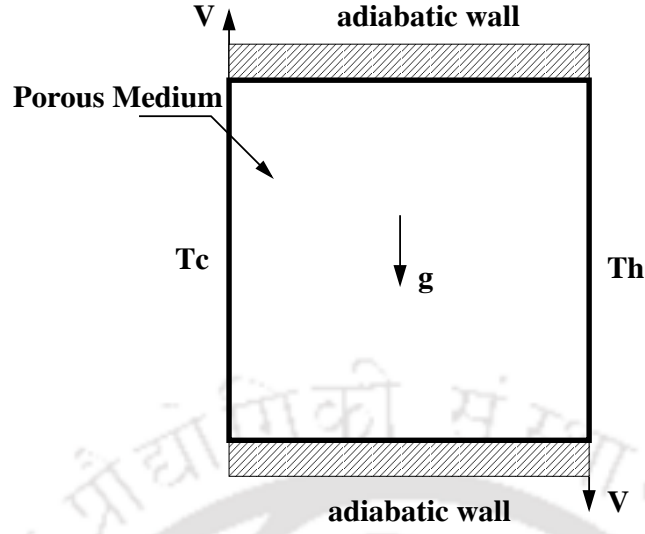


Fig. 6.1: Schematic diagram of the flow configuration.

$$\begin{aligned} \frac{1}{\epsilon} \frac{\partial u}{\partial t} + A \left[\frac{1}{\epsilon^2} u \frac{\partial u}{\partial x} + \frac{1}{\epsilon^2} v \frac{\partial u}{\partial y} \right] = & -\frac{\partial p}{\partial x} + B \frac{1}{(\epsilon.Re)} \left[\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right] \\ & -C \left[\frac{1.75}{\sqrt{150}} \frac{\sqrt{(u^2 + v^2)}}{\sqrt{Da}} \frac{u}{\epsilon^{3/2}} \right] - \frac{u}{Da.Re} \end{aligned} \quad (6.2)$$

$$\begin{aligned} \frac{1}{\epsilon} \frac{\partial v}{\partial t} + A \left[\frac{1}{\epsilon^2} u \frac{\partial v}{\partial x} + \frac{1}{\epsilon^2} v \frac{\partial v}{\partial y} \right] = & -\frac{\partial p}{\partial y} + B \frac{1}{(\epsilon.Re)} \left[\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right] \\ & -C \left[\frac{1.75}{\sqrt{150}} \frac{\sqrt{(u^2 + v^2)}}{\sqrt{Da}} \frac{v}{\epsilon^{3/2}} \right] - \frac{v}{Da.Re} + \frac{Gr}{Re^2} \theta \end{aligned} \quad (6.3)$$

$$\sigma \frac{\partial \theta}{\partial t} + u \frac{\partial \theta}{\partial x} + v \frac{\partial \theta}{\partial y} = \frac{1}{Re.Pr} \left(\frac{\partial^2 \theta}{\partial x^2} + \frac{\partial^2 \theta}{\partial y^2} \right) \quad (6.4)$$

The scaled variables and physical parameters are defined as follows:

$$\begin{aligned} x = \frac{x^*}{H}, \quad y = \frac{y^*}{H}, \quad u = \frac{u^*}{V}, \quad v = \frac{v^*}{V}, \quad p = \frac{p^*}{\rho V^2}, \\ \theta = \frac{T - T_c}{T_h - T_c}, \quad \sigma = \left[\epsilon (\rho c_p)_{fl} + (1 - \epsilon) (\rho c_p)_s \right] / (\rho c_p)_{fl} \end{aligned}$$

$$Re = \frac{VH}{\nu}, \quad Gr = \frac{g\beta H^3 \Delta T}{\nu^2}, \quad Da = \frac{\kappa}{H^2}, \quad Pr = \frac{\nu}{\alpha_e}$$

where asterisked variables are dimensional and other dimensional variables without the stars are explained in the nomenclature. The coefficients A, B, C can be set equal to 0 or 1 in order to obtain the existing porous media flow models such as the Darcy or the Brinkman and Forchheimer models. In this study the coefficients A and B are set equal to 1 because the flow velocity is high, where inertia and boundary effects are significant. In addition if $C = 1$ it gives the Brinkman-Forchheimer Darcy equation (BFDE). Brinkman-extended Darcy equation (BDE) can be deduced from Brinkman-Forchheimer Darcy equation by setting the coefficient $C = 0$. The dimensionless boundary conditions are given as:

$$x = 0 : u = 0.0, \quad v = 1.0, \quad \theta = 0.0$$

$$x = 1 : u = 0.0, \quad v = -1.0, \quad \theta = 0.0$$

$$y = 0 : u = 0.0, \quad v = 0.0, \quad \theta = 0.0$$

$$y = 1 : u = 0.0, \quad v = 0.0, \quad \theta = 0.0$$

The dimensionless governing equations for a general field variable ϕ can be written as

$$\alpha_\phi \frac{\partial \phi}{\partial t} + \delta_\phi \left(\frac{\partial}{\partial x} (u\phi) + \frac{\partial}{\partial y} (v\phi) \right) = \frac{\partial}{\partial x} \left(\Gamma^\phi \frac{\partial \phi}{\partial x} \right) + \frac{\partial}{\partial y} \left(\Gamma^\phi \frac{\partial \phi}{\partial y} \right) + S^\phi \quad (6.5)$$

where Γ^ϕ denotes diffusion coefficient and S^ϕ the source term.

The following substitutions in Eq. (6.5) give x -momentum equation:

$$\begin{aligned} \phi &= u, \quad \Gamma^\phi = \frac{1}{\epsilon \cdot Re}, \quad \alpha_\phi = \frac{1}{\epsilon}, \quad \delta_\phi = \frac{1}{\epsilon^2}, \\ S^\phi &= -\frac{\partial p}{\partial x} - \left[\frac{1.75}{\sqrt{150}} \frac{\sqrt{(u^2 + v^2)}}{\sqrt{Da}} \frac{u}{\epsilon^{3/2}} \right] - \frac{u}{Da \cdot Re} \end{aligned} \quad (6.6)$$

The y -momentum equation results from the substitution

$$\begin{aligned} \phi &= v, \quad \Gamma^\phi = \frac{1}{\epsilon \cdot Re}, \quad \alpha_\phi = \frac{1}{\epsilon}, \quad \delta_\phi = \frac{1}{\epsilon^2}, \\ S^\phi &= -\frac{\partial p}{\partial y} - \left[\frac{1.75}{\sqrt{150}} \frac{\sqrt{(u^2 + v^2)}}{\sqrt{Da}} \frac{v}{\epsilon^{3/2}} \right] - \frac{v}{Da \cdot Re} + \frac{Gr}{Re^2} \theta \end{aligned} \quad (6.7)$$

and the energy equation results from

$$\phi = \theta, \quad \Gamma^\phi = \frac{1}{Pr.Re}, \quad \alpha_\phi = \sigma, \quad \delta_\phi = 1, \quad S^\phi = 0 \quad (6.8)$$

The average Nusselt number $\overline{Nu}$ is calculated by integrating the local Nusselt number Nu along the left wall and is given by

$$\overline{Nu} = \int_0^1 Nu \, dy \quad (6.9)$$

where, the local Nusslet number Nu is given by

$$Nu = -\frac{1}{A} \frac{(\partial\theta/\partial x)_w}{\theta_h - \theta_c} \quad (6.10)$$

6.3 Numerical Method

The present problem of mixed convection in a cavity filled with porous medium is numerically solved using the full approximation storage multigrid method. The FAS multigrid method has been described in Section 5.2.2. The code is based on the SIMPLE algorithm on a colocated grid arrangement using finite volume method. The convective fluxes at the cell interface in both momentum and energy equations are computed by using the Quadratic Upwind Interpolation for Convective Kinematics (QUICK) scheme and central differences are used to compute the diffusive fluxes and the pressure gradient. The convergence criterion used here requires that the mass residual should be less than 10^{-10} . To test the grid independence of the solution numerical experiments were performed for various uniform grids for $Ri = 0.01$, $Gr = 10^4$ and $Da = 10^{-1}$ and the corresponding results are shown in Fig. 6.2. It is clear that an equally spaced grid of 130×130 is adequate and refining the grid further produces the same results (Figs. 6.2(a) and 6.2(b)).

Besides discussing the other aspects, we also examine the accuracy of the present code by comparing its result with other existing results. The validation of the code against the work

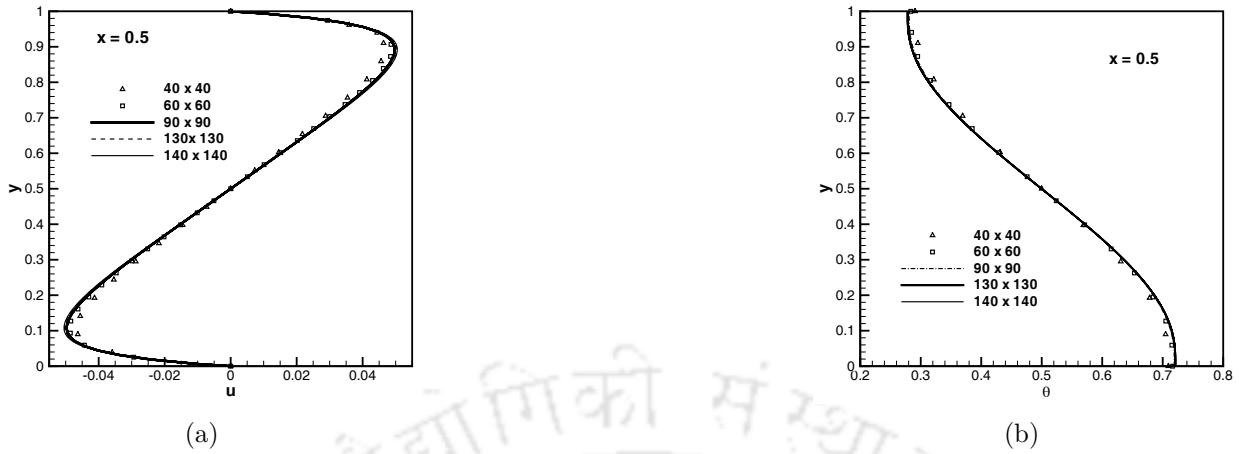


Fig. 6.2: Grid-independence study (a) horizontal velocity and (b) temperature along the vertical centre line at $Ri = 0.01$, $Gr = 10^4$ and $Da = 10^{-3}$.

of Khanafer and Vafai [63] in a double-diffusive mixed convection in a lid-driven cavity filled with porous medium was performed (Fig. 6.3). It can be seen from Figs. 6.3(a) and 6.3(b) that the solution of the present numerical code is in excellent agreement with the numerical results of [63] for various buoyancy ratios at $Re = 100$, $Gr = 10^2$, $Le = 1.0$, $Da = 0.1$ and $\epsilon = 0.95$. When $\epsilon \rightarrow 1$ and $Da \rightarrow \infty$, the Brinkman-Forchheimer Eqs. (6.2) and (6.3) reduce

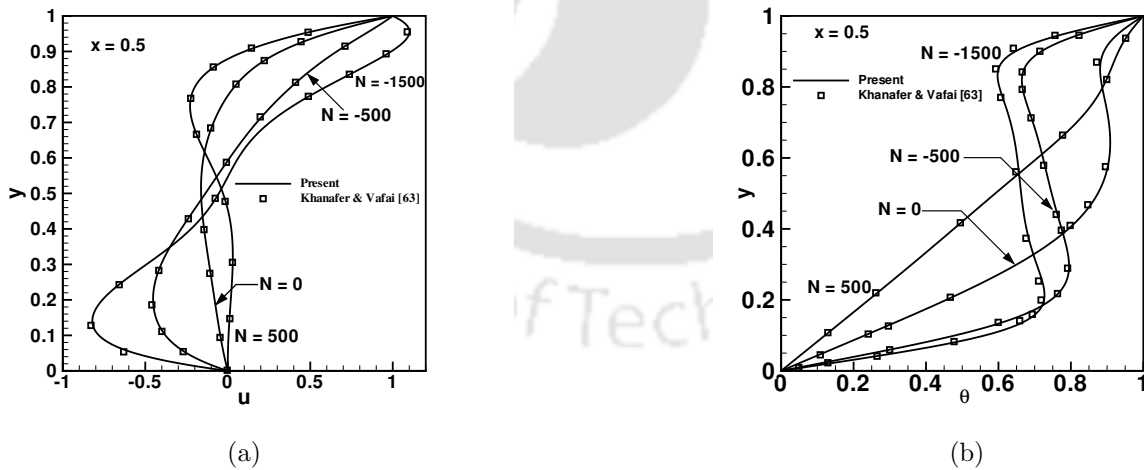


Fig. 6.3: Comparison of (a) horizontal velocity and (b) temperature profile along the vertical centre line for various buoyancy ratios (N) with Khanafer and Vafai [63].

to the Navier-Stokes equation for free fluid flows (mixed convection in the absence of the porous medium). By taking $\epsilon = 0.99999$ and $Da = 10^7$ in the Eqs. (6.1)-(6.4) and using

the present code a natural convection flow problem is also computed in order to gain further testimony in support of the code. Fig. 6.4 shows the streamline and isotherm pattern for various values of Rayleigh numbers. These pattern bear very close resemblance with similar figures presented by de Vahl Davis [30] and Kalita *et al.* [55]. The predicted average Nusselt number on the hot wall are compared with those given by de Vahl Davis [30] and Nithiarasu *et al.* [83] for various Rayleigh numbers (Table 6.1). The results of the present study agree well with those of the earlier investigators over the entire range of Rayleigh numbers, thus lending credibility to the present code and numerical method.

Table 6.1: Comparison of average Nusselt number for natural convection in a square cavity with available literature

Ra	Average Nusselt number		
	deVahl Davis [30]	Nithiarasu et al [83]	Present
10^3	1.116	1.127	1.117
10^4	2.238	2.245	2.246
10^5	4.509	4.521	4.530
10^6	8.817	8.800	8.822

6.4 Results and Discussion

In the present work, the flow due to forced convection and the flow due to natural convection are investigated which depends upon Richardson number (Ri). When $Ri = 0.01$ forced convection becomes the dominated parameter and $Ri = 100$ then the natural convection becomes the dominated parameter. The multigrid-accelerated code developed is used to compute the flows in the range of Darcy numbers $10^{-3} \leq Da \leq 10^{-1}$ and Grashof numbers 10^2 and 10^4 . The other parameters such as Prandtl number ($Pr = 0.71$) and porosity ($\epsilon = 0.9$) are assumed to remain constant. We analysed the two models, namely BFDM and BDM, for $Ri = 0.01$. After gaining confidence that BFDM produces better results than

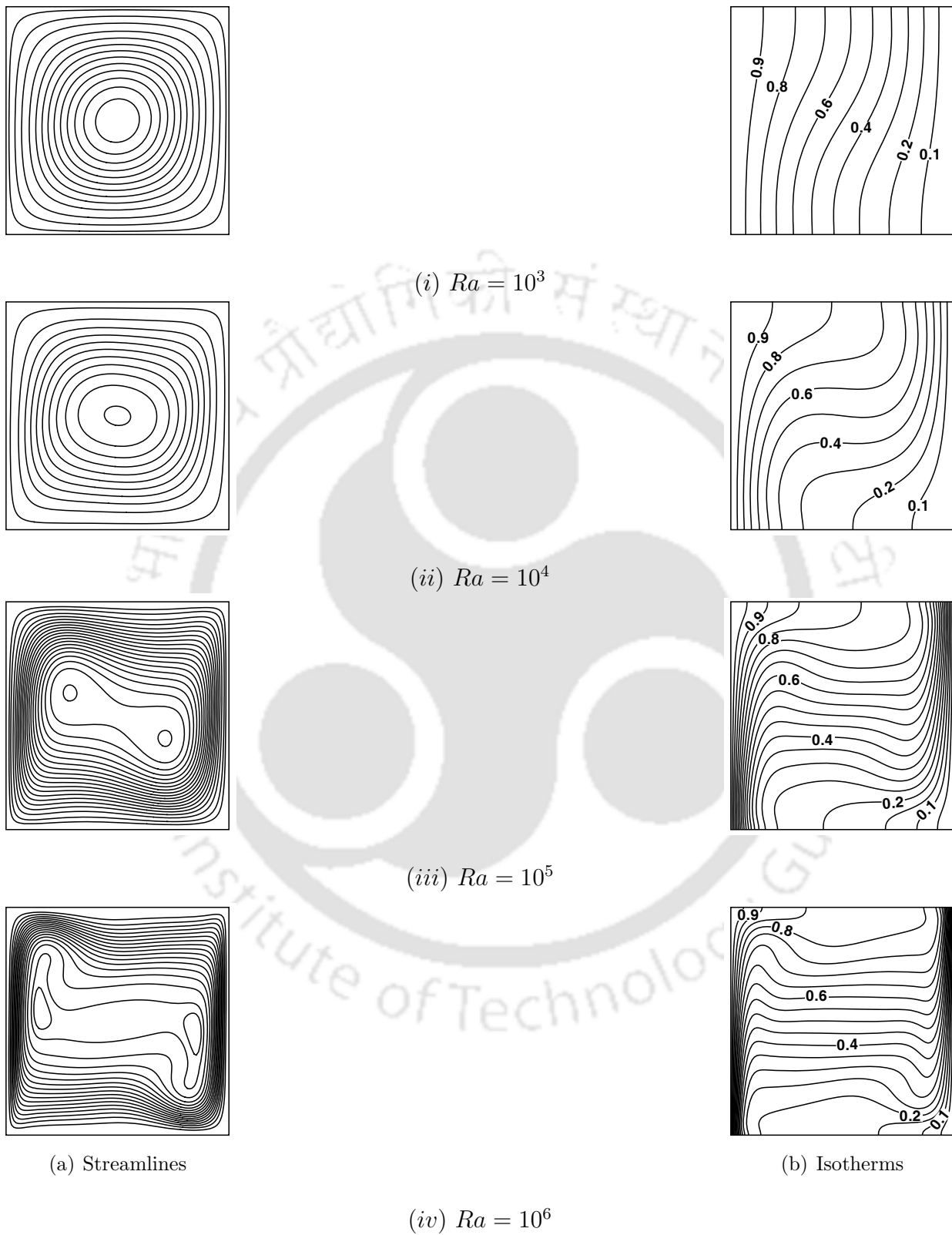


Fig. 6.4: (a) Streamline and (b) isotherm patterns for various Rayleigh numbers for natural convection in a square cavity.

BDM, we carry out the computations for $Ri = 100$.

6.4.1 Flow Due to Forced Convection

Effects of Inertial Term at Low Grashof Number ($Gr = 10^2$)

In the first case, when $Gr = 10^2$ and $Ri = 0.01$, the corresponding Reynolds number given by $\sqrt{Gr/Ri}$ equals 100. A comparison of BFDM and BDM is shown in Fig. 6.5 for the midplane horizontal velocity and temperature at constant $Ri = 0.01$ and $Gr = 10^2$ for various Darcy numbers. When Darcy number is low ($Da = 10^{-3}$) both models produce almost identical velocity profiles because of negligible convective effects (Fig. 6.5(a)). When Darcy number, and hence convection, increases - expectedly the velocity profiles deviate from each other proving the inadequacy of BDM at $Da = 10^{-1}$ and 10^{-2} . The magnitudes of velocity and temperature predicted by BDM are higher than those predicted by BFDM owing to the absence of inertial effect in BDM. This is in conformity with the observations in [1], which mentions that the presence of inertial effects in BFDM hinders momentum and energy transport. Understandably this deviation is relatively more pronounced for higher permeability of the porous medium represented by the Darcy number, as the flow conductance increases with permeability.

Fig. 6.6 shows the streamlines and isotherm patterns for BFDM and Fig. 6.7 shows those for BDM. At $Da = 10^{-3}$ the flow and temperature patterns (Figs. 6.6(i) and 6.7(i)) have a very close resemblance. Increase in Darcy number induces the flow activity deeper into the cavity, which causes more energy to be carried away from the sliding walls, thus causing marked changes in the flow behaviour. As the Darcy number increases the vortices generated near the vertical sliding walls move away from the walls and they eventually coalesce into a single vortex (Figs. 6.6(ii,iii) and 6.7(ii,iii)). Another observation that can be made from Figs. 6.6 and 6.7 is that as the sidewall vortices have a clockwise sense of rotation, the right vortex drags hotter fluid to the bottom wall and the left vortex drags colder fluid to the top wall with the result that with an increase in Darcy number, and hence convection, average

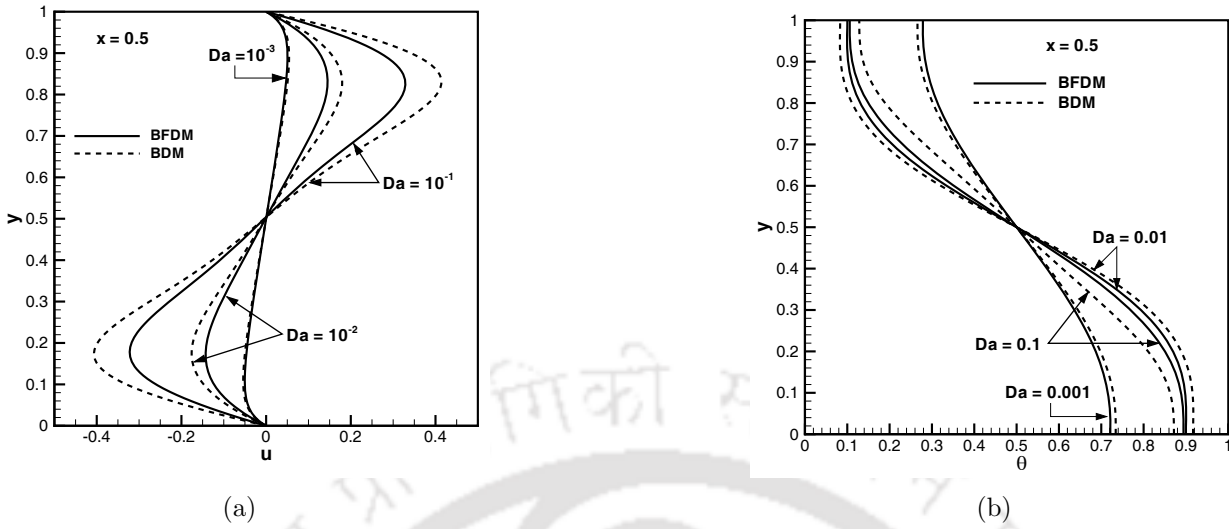


Fig. 6.5: (a) Horizontal velocity and (b) temperature along the vertical centre line at different Darcy numbers (Da) for $Ri = 0.01$, $Gr = 100$ and $\epsilon = 0.9$.

temperature of the top wall decreases and that of the bottom wall increases. This conclusion can be made from the fact that with Darcy number the hotter isotherms become more and more concentrated near the top right corner whereas colder isotherms become more and more concentrated near the bottom left corner. At $Da = 0.1$, both the models show that the sidewall vortices merge into one. This isotherm patterns for both the models demonstrate that with an increase in Darcy number the heat transfer between the two vertical walls is influenced more and more by convection compared with conduction. However, fractional influence of convection in the heat transfer appears to be somewhat higher for BDM compared with BFDM.

Effects of Inertial Term at High Grashof Number ($Gr = 10^4$)

In the second case, when Grashof number is increased to 10^4 keeping Richardson number ($Ri = 0.01$), unchanged, the corresponding Reynolds number given by $\sqrt{Gr/Ri}$ increases to 1000. A comparison of BFDM and BDM is shown in Fig. 6.8 for the midplane horizontal velocity and temperature at constant $Ri = 0.01$ and $Gr = 10^4$ for various Darcy numbers. When Darcy number is low ($Da = 10^{-3}$), both the velocity and temperature profiles show only a small variation between these two models (Figs. 6.8(a) and 6.8(b)) because of low

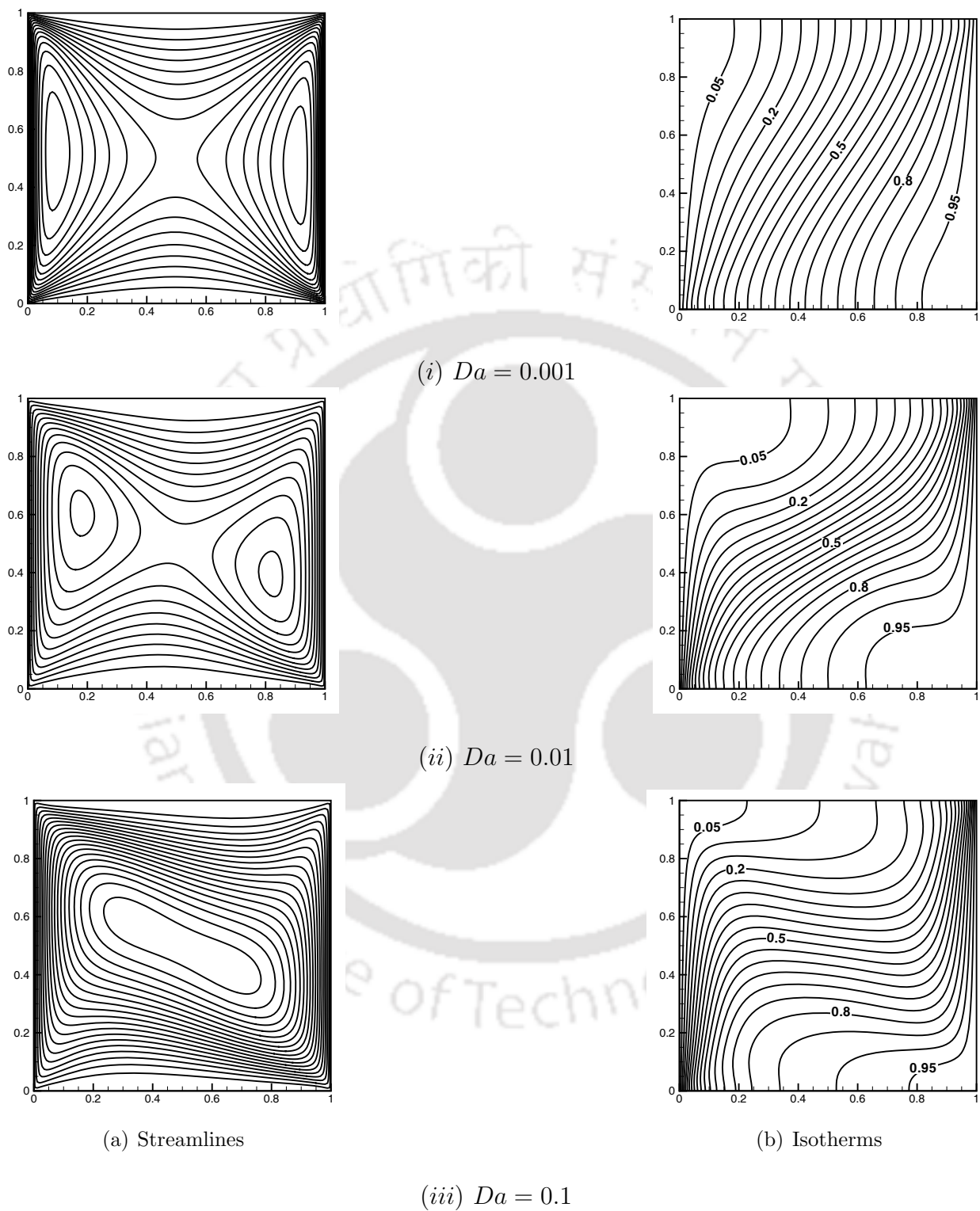


Fig. 6.6: BFD (a) streamline and (b) isotherm patterns for $Ri = 10^{-2}$, $Gr = 10^2$, $\epsilon = 0.9$ and for various Darcy numbers (Da).

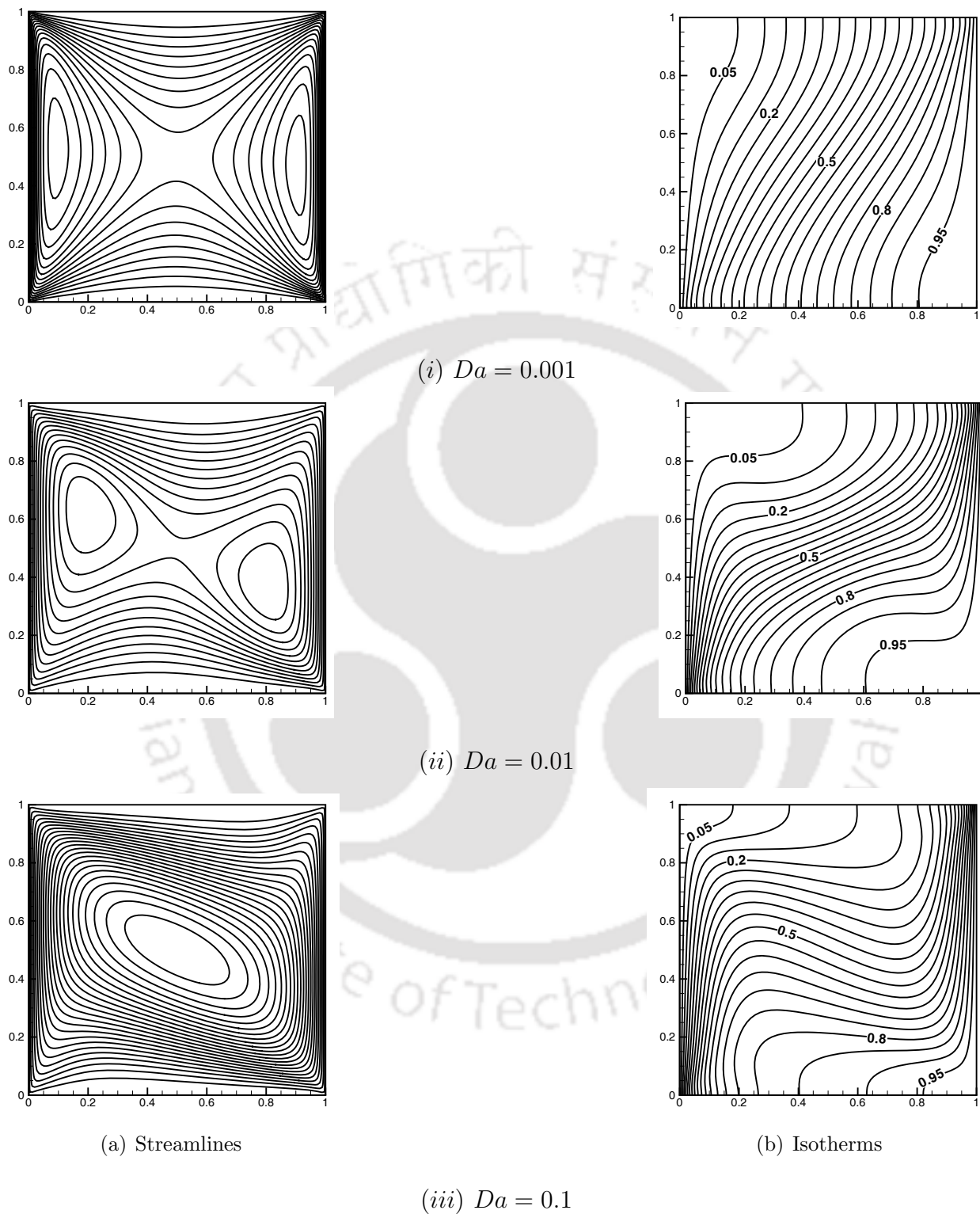


Fig. 6.7: BDM (a) streamline and (b) isotherm patterns for $Ri = 10^{-2}$, $Gr = 10^2$, $\epsilon = 0.9$ and for various Darcy number (Da).

convective effects. However, compared with the case for $Gr = 10^2$ (Figs. 6.5(a) and 6.5(b)), this variation is higher. This is understandable as at higher values of Gr effect of convection is more pronounced and so is the discrepancy between the two models. As Darcy number increases, expectedly the velocity profiles deviate more and more from each other. In conformity with the observations in [1] the magnitudes of velocity and temperature predicted by BDM is higher than those predicted by BFDM. The difference is again higher for $Gr = 10^4$ than for $Gr = 10^2$ (see Figs. 6.5 and 6.8). The presence of quadratic nature of the inertial effects in BFDM makes their contribution more noteworthy in hindering the fluid motion as flow activities intensify. The discrepancy in velocity prediction between the two models has subsequently impacted the temperature results. Fig. 6.9 shows the streamlines and

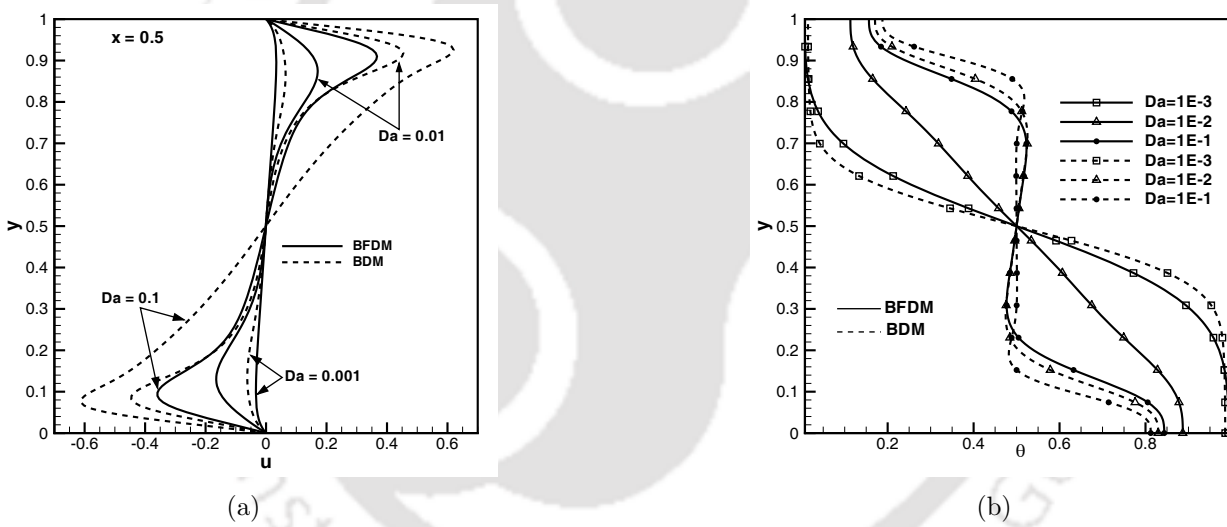
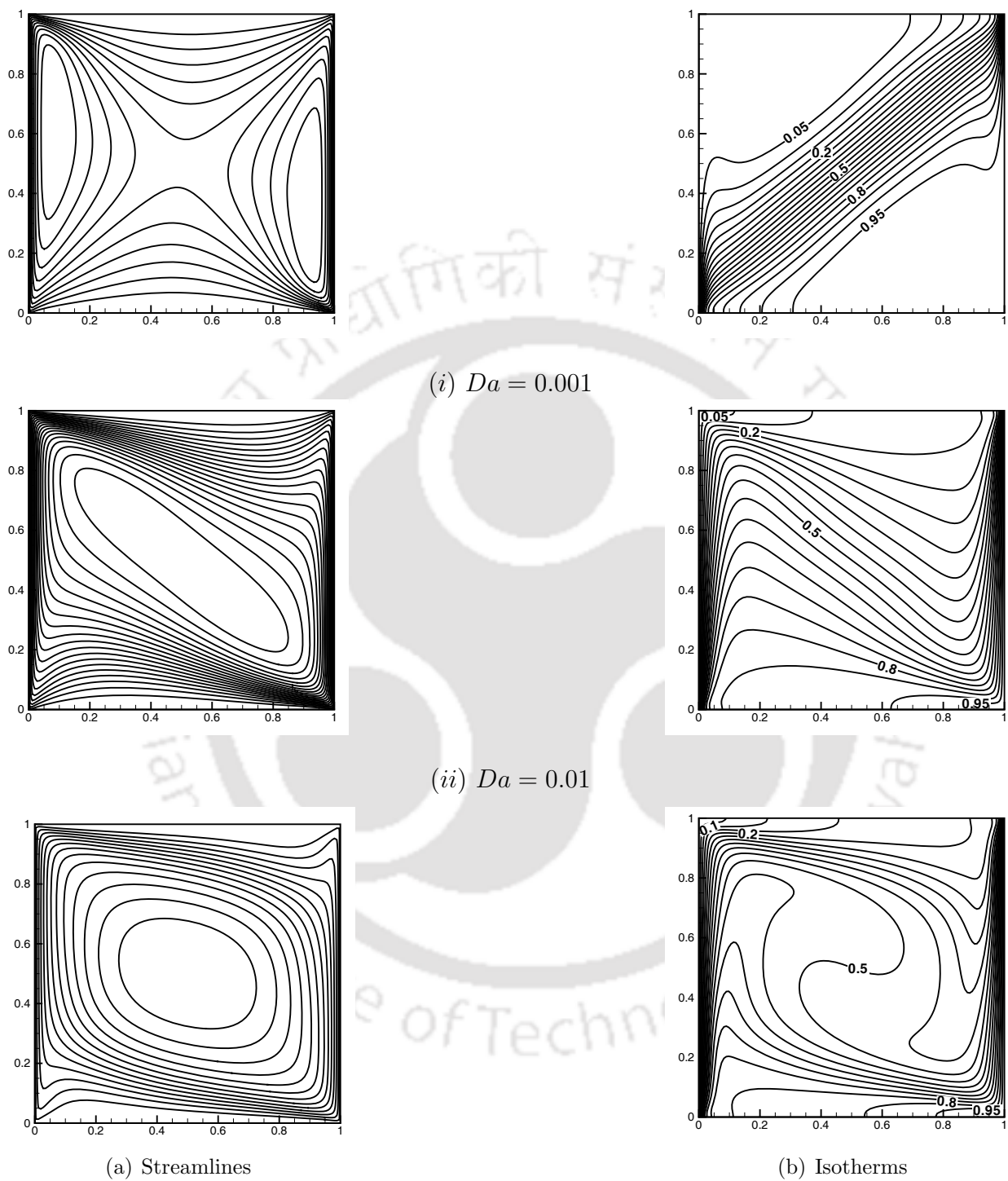


Fig. 6.8: (a) Horizontal velocity and (b) temperature along the vertical centre line at different Darcy numbers (Da) for $Ri = 0.01$, $Gr = 10^4$ and $\epsilon = 0.9$.

isotherm pattern for BFDM and Fig. 6.10 shows those for BDM. At lower Darcy number ($Da = 10^{-3}$), the streamlines and temperature pattern shows relatively smaller variations (Figs. 6.9(i) and 6.10(i)). As seen from Figs. 6.9(a) and 6.10(a), the streamlines come closer together near the left and right walls forming regions of high normal velocity gradient. As the left wall moves upwards and right wall moves downwards, the thickness of these shear layers increases from bottom to top on the left wall and from top to bottom on the right wall. As Da increases from 0.001 (as for $Gr = 10^2$) the two sidewall vortices merge into one.

For $Da = 0.001$ (BFDM, Fig. 6.9(a)) the vortex-core has a somewhat elliptical shape and for $Da = 0.1$ it becomes somewhat circular just like the case of antiparallel lid motion in a purely fluid flow problem [70]. For BDM (Fig. 6.10(a)) the streamline patterns at $Da = 0.01$ itself looks like that of $Da = 0.1$ for BFDM (Fig. 6.9(a)). This is in conformity with the fact that the presence of the additional inertial term in BFDM hinders the flow with the result that flow pattern given by BDM at lower Da looks like the flow pattern given by BFDM at a higher Da . In other words the effects of convection are more pronounced for the predictions given by BDM than those given by BFDM. From Figs. 6.9(b) and 6.10(b) it can be concluded that at low values of $Da = 0.001$ the isotherms cluster near the top right and the bottom left walls, which develop into thermal boundary layers at higher values of $Da = 0.01$ and 0.1 as convection starts playing a dominant role. As for $Gr = 10^2$, at $Gr = 10^4$ also, with an increase in Da the average temperature of the top wall shows a decreasing trend while that of the bottom wall shows an increasing trend. Also from the isotherm patterns given by both the models it can be concluded that BDM demonstrates higher convective activity compared with BFDM.

To understand the heat transfer effect of Da and Gr for both the models, the cold (left) wall local Nusselt number (Nu) is plotted at $Ri = 0.01$ for $Gr = 10^2$ (Fig. 6.11(a)) and $Gr = 10^4$ (Fig. 6.11(b)). At $Gr = 10^2$ it is seen that Nu increases from the top to bottom and since Gr represents the ratio of heat transfer by convection to that by conduction, the fraction of heat transfer by convection decreases with the height of the wall. Expectedly, BDM without the inertia term that hinders convective effects, gives a higher value of Nu compared with BFDM. At $Gr = 10^4$ similar trends in Nu is generally observed with two exceptions. First the Nu at any point in the wall is much higher than that for $Gr = 10^2$. Second the monotonic increase of Nu with Da is not observed especially towards the lower part of the wall. With an increase in Gr , buoyancy effects become competitive with the effects of the plate movement and this results in heat transfer trend that is in variance with the trend for the lower value of $Gr = 10^2$. The average Nusselt number $\overline{Nu}$ for various values of Da at $Gr = 10^2$ and 10^4 is shown in Table 6.2. Since BDM does not have the inertia



(iii) $Da = 0.1$

Fig. 6.9: BFDm (a) Streamline and (b) isotherm patterns for $Ri = 10^{-2}$, $Gr = 10^4$, $\epsilon = 0.9$ and for various Darcy numbers (Da).

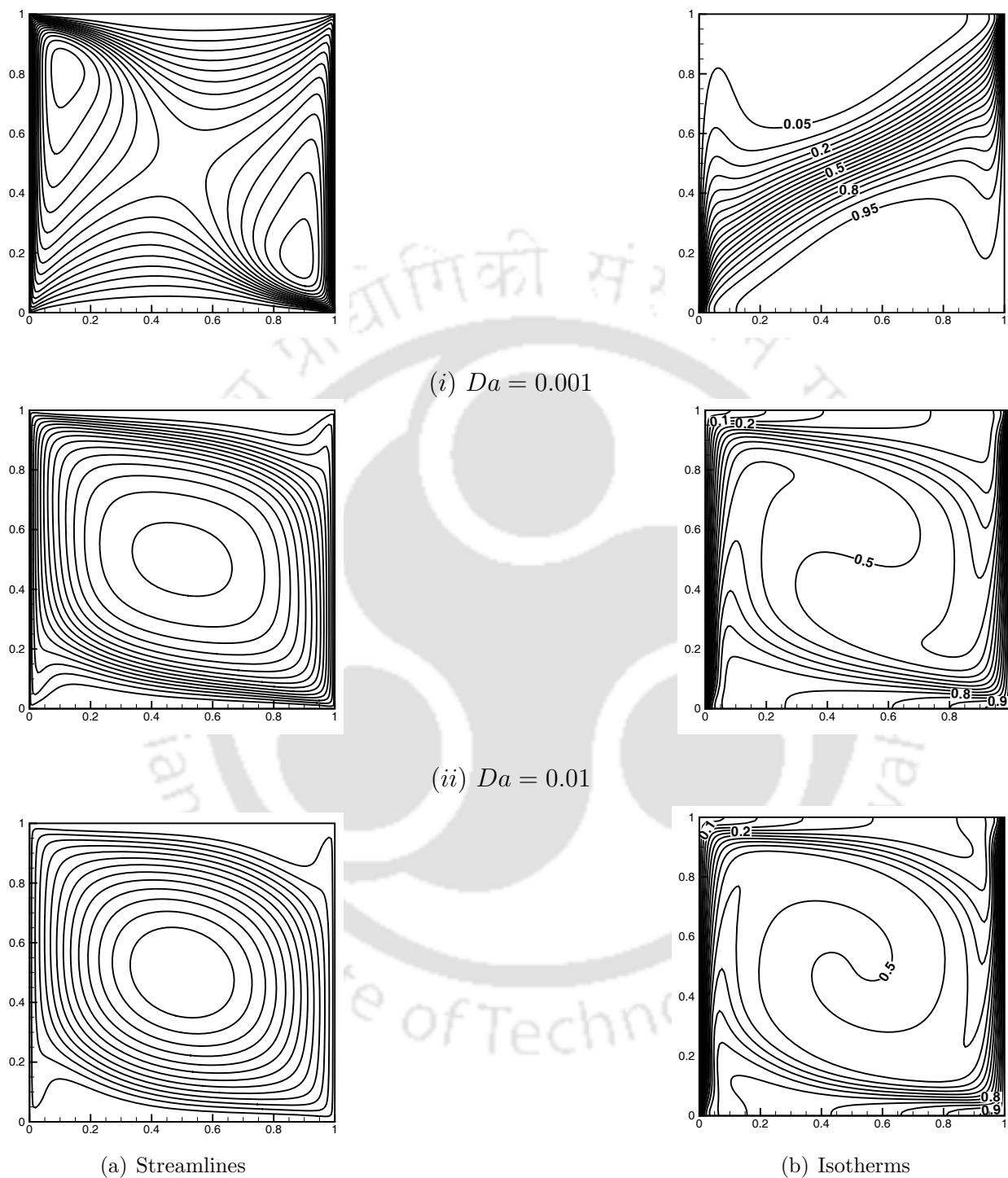


Fig. 6.10: BDM (a) Streamline and (b) isotherm patterns for $Ri = 10^{-2}$, $Gr = 10^4$, $\epsilon = 0.9$ and for various Darcy numbers (Da).

term that hinders convective activity, it gives a higher value of $\overline{Nu}$ for both the values of Gr . Also it is seen that $\overline{Nu}$ generally increases with an increase in Da with a small exception at $Gr = 10^4$ (last column in Table 6.2). As mentioned earlier, the complex flow pattern that develops because of the competition between the plate movement and buoyancy is seemingly the reason for it.

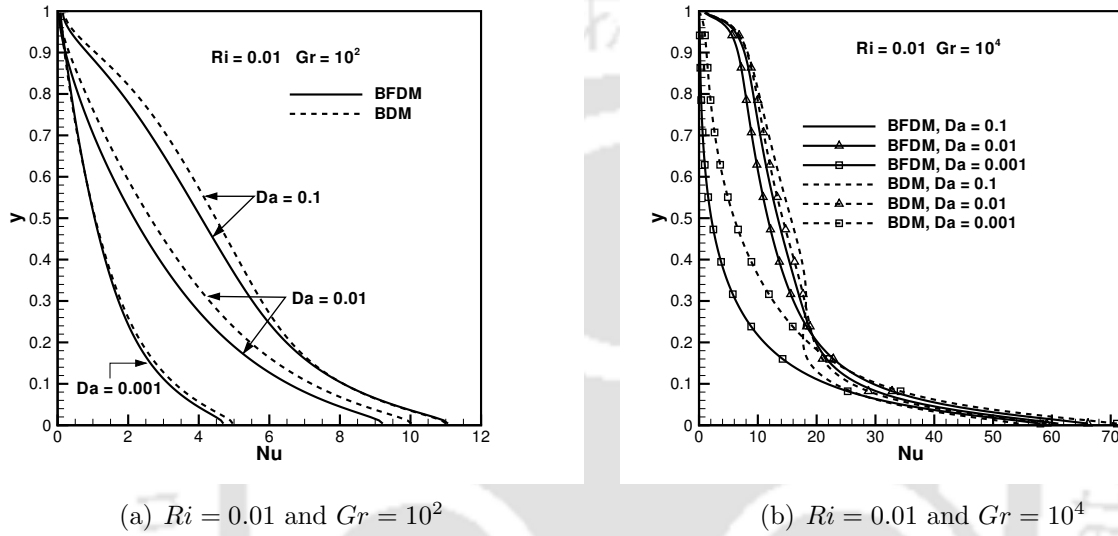


Fig. 6.11: Local Nusselt number at the cold (left) wall for various Darcy numbers (Da): (a) $Gr = 10^2$ (b) $Gr = 10^4$.

Convergence Acceleration by Multigrid

We now describe the performance of the multigrid method used for the present computations. Expectedly the multigrid full approximation scheme shows a faster convergence. All the results presented in the present work were performed using V-cycle (Fig. 5.6). Tests were conducted on both 3-level and 4-level V-cycles and increasing the level further did not pay additional dividend. The number of iterations performed at each grid level is shown inside the circles (Fig. 5.6). The computational effort is reported in terms of equivalent fine-grid sweeps that are usually referred to as work units. Figs. 6.12 and 6.13 present the history of mass residual on a 130×130 grid as a function of the number of work units 1-, 3- and 4-level computations. A 4-level multigrid cycle shows better convergence than 3-level

Table 6.2: Average Nusselt number $\overline{Nu}$ given by Brinkman-Forchheimer Darcy model and Brinkman-extended Darcy model at $Ri = 0.01$

Darcy number	$\overline{Nu} (Gr = 10^2)$		$\overline{Nu} (Gr = 10^4)$	
	BFDM	BDM	BFDM	BDM
10^{-3}	1.379	1.429	7.181	11.690
10^{-2}	2.807	3.228	15.146	16.255
10^{-1}	4.305	4.571	16.008	16.112

Table 6.3: Performance of multigrid for a 130×130 grid for $Ri = 0.01$ and $Gr = 10^4$ and a mass residual of 10^{-10} for BFDM

Darcy number	CPU time (min)			Speed-up	
	Single-grid	MG-3 level	MG-4 level	MG-3 level	MG-4 level
10^{-3}	55.48	10.42	7.54	5.21	7.06
10^{-2}	95.36	17.24	14.36	5.49	6.54
10^{-1}	117.12	21.54	18.43	5.35	6.26

multigrid cycle. Figs. 6.12 and 6.13 show that in the single-grid the residuals fall faster at the beginning and slows down thereafter whereas in multigrid method the residual falls at a more or less constant rate. Tables 6.3 and 6.4 summarize the total CPU time required and speed-up obtained by the multigrid over single grid method for $Gr = 10^2$ and $Gr = 10^4$ for various Darcy numbers. It is seen that multigrid method shows approximately 4 – 5 times faster than single grid method when $Gr = 10^2$ and 6 – 7 times faster when $Gr = 10^4$.

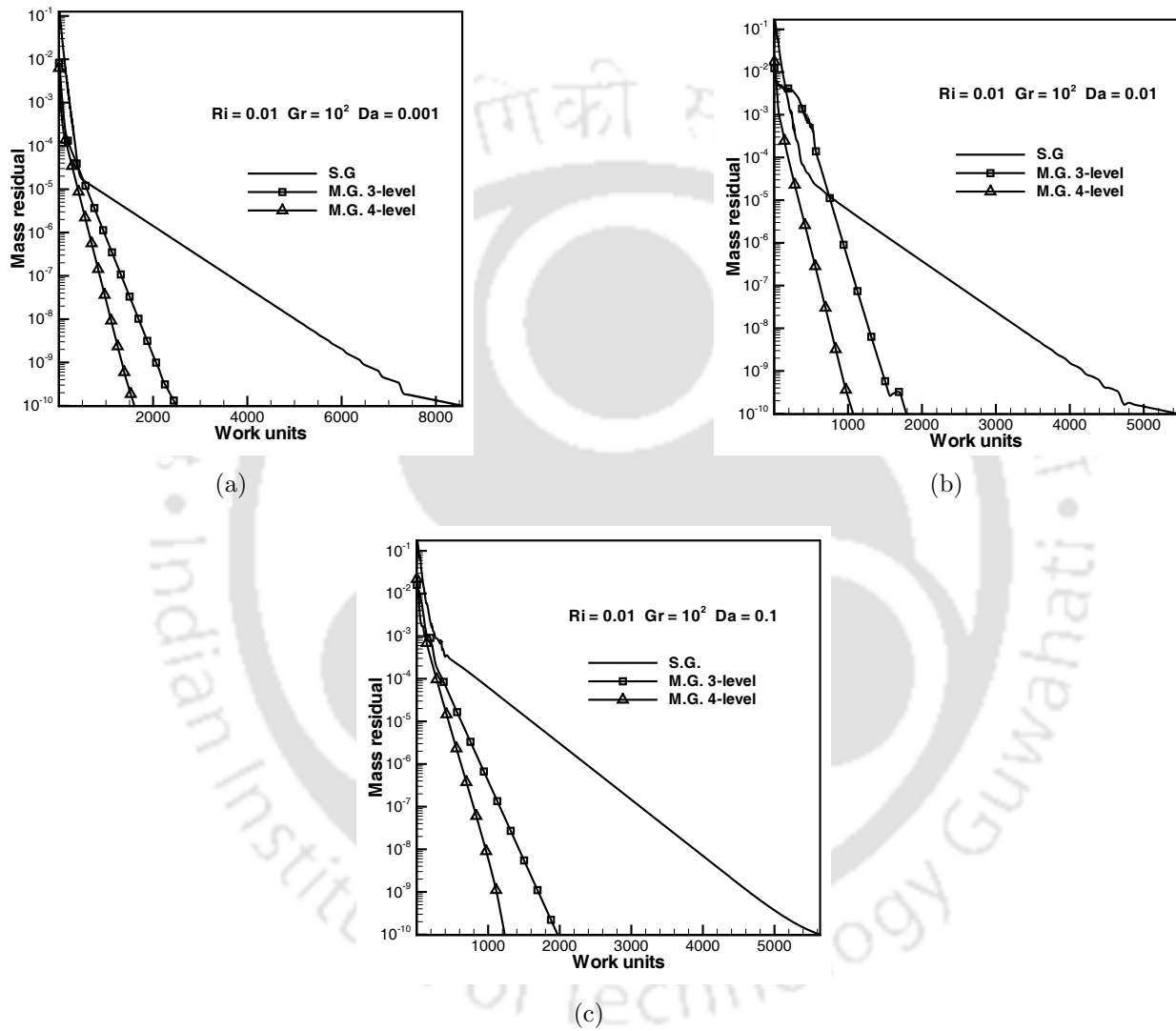


Fig. 6.12: Mass residual history of multigrid methods for $Ri = 0.01$, $Gr = 10^2$ and $\epsilon = 0.9$
 (a) $Da = 0.001$ (b) $Da = 0.01$ (c) $Da = 0.1$ for BFDM.

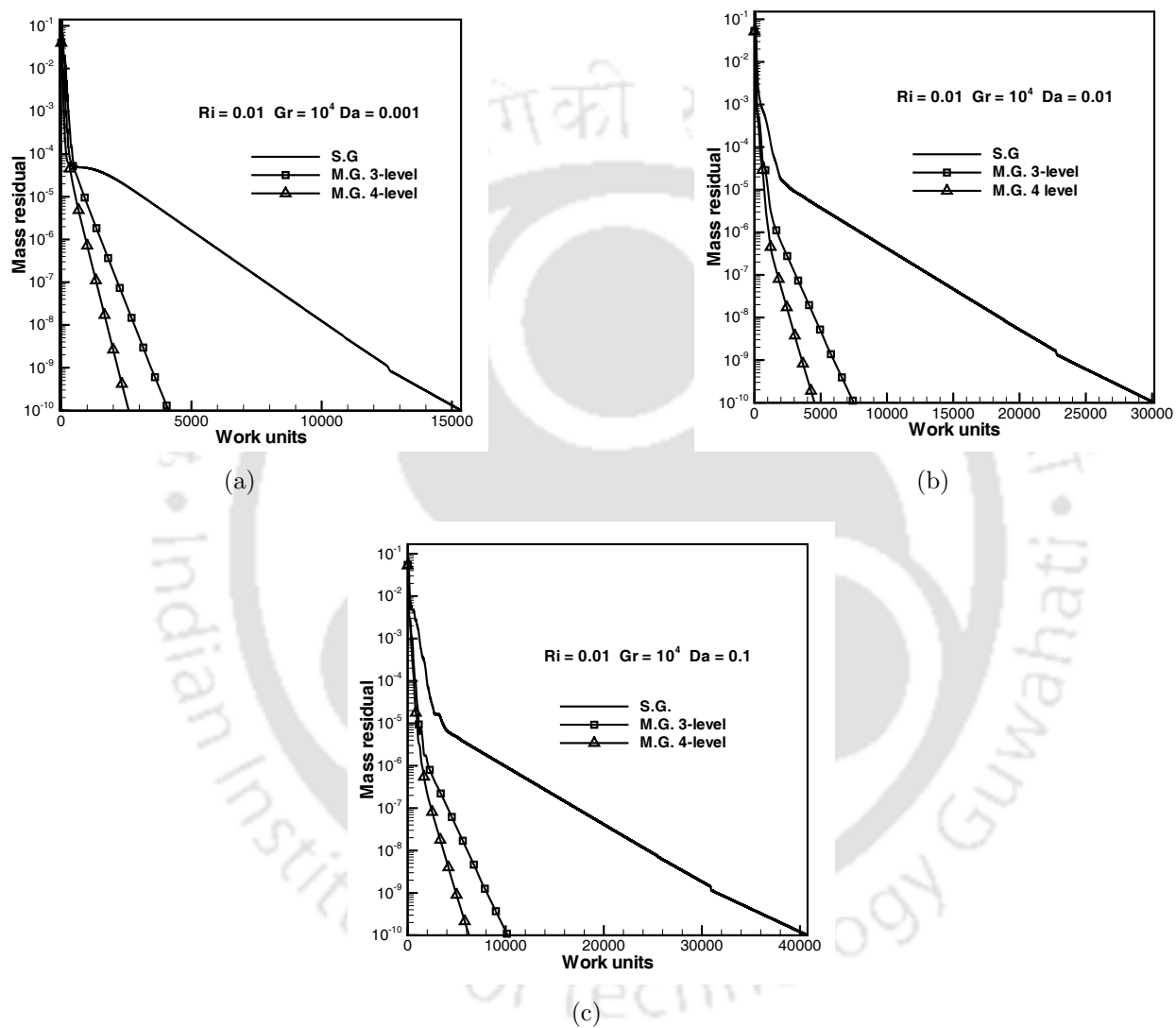


Fig. 6.13: Mass residual history of multigrid methods for $Ri = 0.01$, $Gr = 10^4$ and $\epsilon = 0.9$ (a) $Da = 0.001$ (b) $Da = 0.01$ (c) $Da = 0.1$ for BFDm.

Table 6.4: Performance of multigrid for a 130×130 grid for $Ri = 0.01$ and $Gr = 10^2$ and a mass residual of 10^{-10} for BFDM

Darcy number	CPU time (min)			Speed-up	
	Single-grid	MG-3 level	MG-4 level	MG-3 level	MG-4 level
10^{-3}	23.18	5.25	4.30	4.24	5.17
10^{-2}	19.42	5.06	4.12	3.86	4.69
10^{-1}	18.42	5.18	3.48	3.52	4.92

6.4.2 Flow Due to Natural Convection

In Section 6.4.1 we showed that BFDM gives better results than BDM for the forced convection dominated flows ($Ri = 0.01$). Therefore, we used BFDM for $Ri = 100$, where the natural convection becomes a dominating parameter in the flow. At $Gr = 10^2$ and 10^4 , Da is varied from 10^{-3} to 10^{-1} . The streamlines and isotherms for the natural-convection dominated flows are reported for various Darcy numbers.

Flow at Low Grashof Number ($Gr = 10^2$)

When $Ri = 100$ and $Gr = 100$ the corresponding Re ($Re = \sqrt{Gr/Ri}$) is unity. Fig. 6.14(a-b) depicts the centre-line horizontal velocity and temperature for various Darcy numbers. Increase in Da does not cause much change in velocity (Fig. 6.14(a)). The temperature has an increasing trend from the top to the bottom wall, but the variation in magnitude is small (Fig. 6.14(b)). Increase in Da , results in small increase in temperature. Fig. 6.15 presents the streamlines and temperature contours respectively for various Darcy numbers. As Ri is high, there is a relative dominance of natural convection over the forced convection heat transfer. At $Da = 10^{-3}$, most of the flow is confined near the sliding lids and vortices are generated near these lids (Fig. 6.15(i)). When Da increases, vortex cores move away from the sliding lids and rotation is clockwise due to the sliding lids (Fig. 6.15(ii)). As Da is

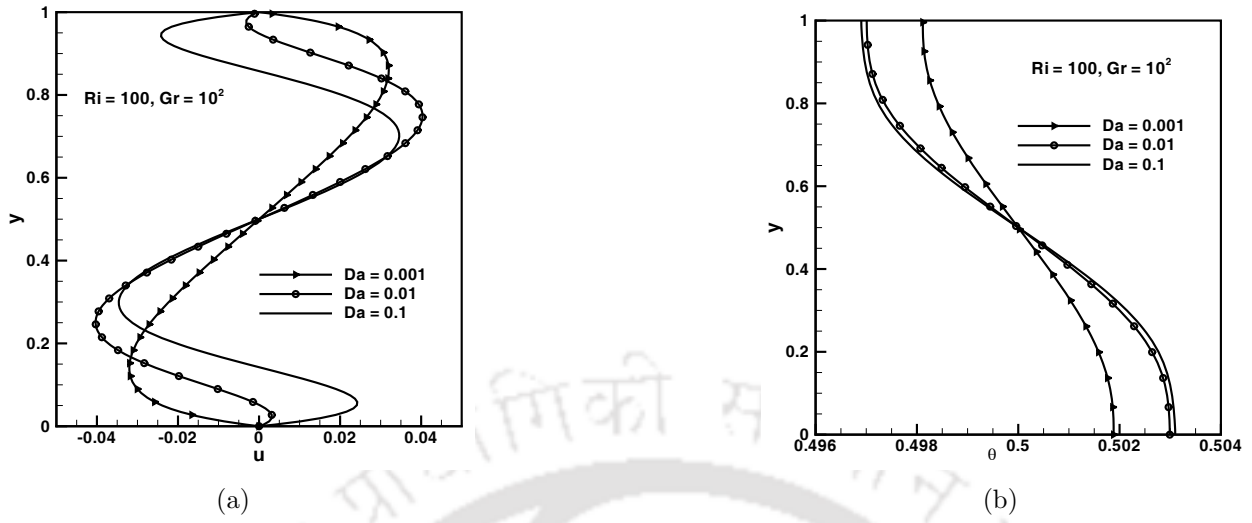


Fig. 6.14: (a) Horizontal velocity and (b) temperature along the vertical centre line at different Darcy numbers (Da) for $Ri = 100$, $Gr = 10^2$ and $\epsilon = 0.9$.

increased to 10^{-1} new vortices are generated at the top and bottom walls of the cavity due to the natural convection (Fig. 6.15(iii)). Direction of these vortices is anti-clockwise because of the effect of natural convection. The isotherms for all Darcy number are nearly vertical, which means that convection is low.

Flow at High Grashof Number ($Gr = 10^4$)

When Gr is increased to 10^4 , keeping $Ri = 100$, then the corresponding Re is 10. Fig. 6.16(a-b) depicts the horizontal velocity and temperature along the vertical centre-line for different Darcy numbers. Increase in Darcy number increases the velocity. Since the flow is dominated by natural convection ($Ri = 100$, $Gr = 10^4$), the flow rotates in the anti-clockwise direction. Therefore the midplane horizontal velocity profile in Fig. 6.16(a) has negative velocity above the centre of the cavity and positive velocity below the centre of the cavity. Increase in Da increases the natural convection heat transfer as depicted in Fig. 6.16(b). Due to the effect of natural convection hot fluid at the right wall reaches the top wall and cooler fluid at the left wall reaches the bottom wall. Therefore the average temperature of the top wall is higher than that of the bottom wall. Fig. 6.17 presents the streamlines and isotherm patterns for various Darcy numbers. When $Da = 10^{-3}$ a vortex due to natural convection rotating in the

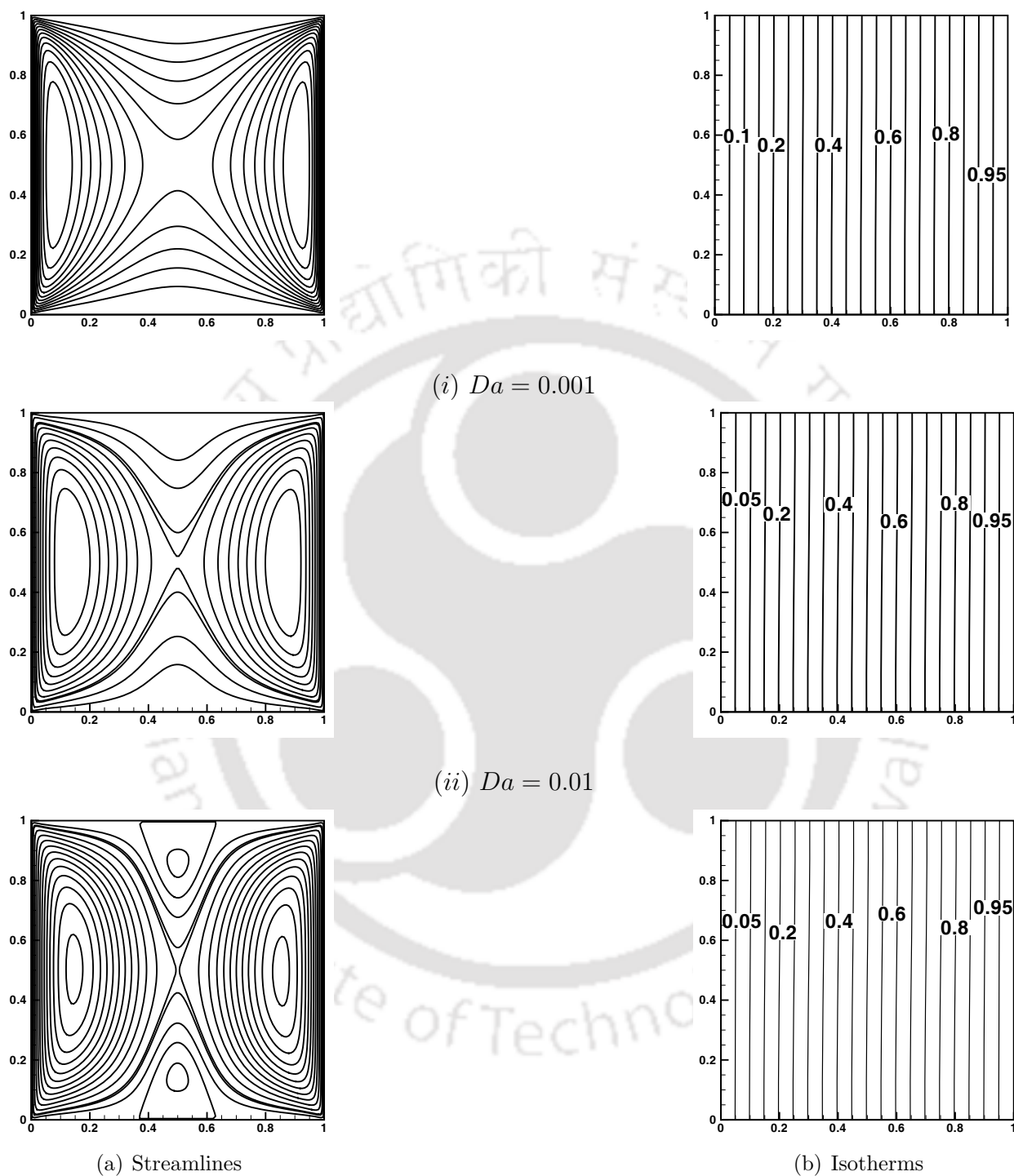


Fig. 6.15: (a) Streamline and (b) isotherm patterns for $Ri = 100$, $Gr = 10^2$, $\epsilon = 0.9$ and for various Darcy numbers (Da).

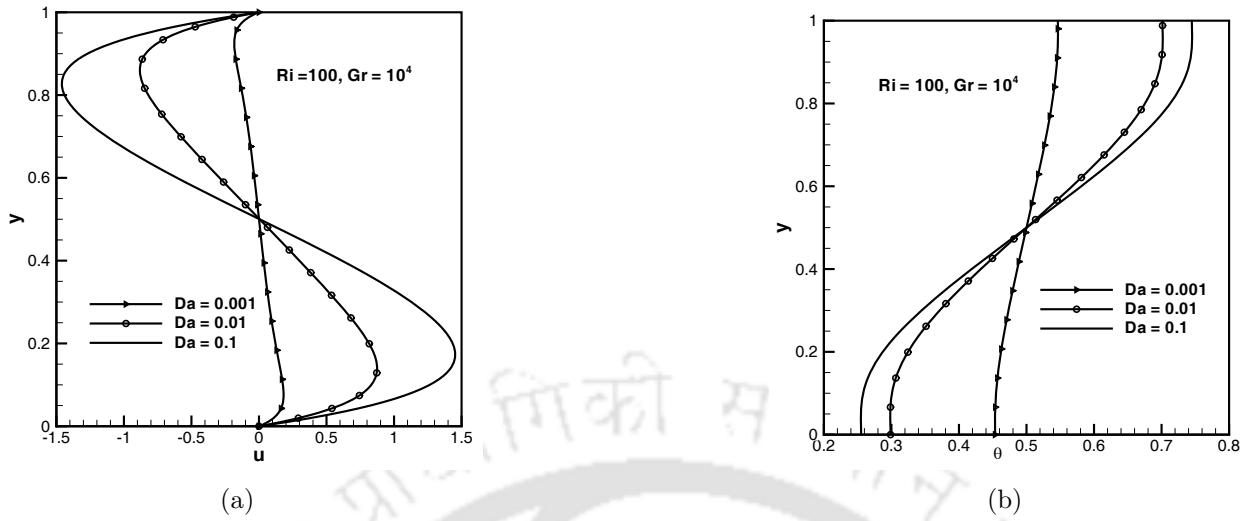


Fig. 6.16: (a) Horizontal velocity and (b) temperature along the vertical centre line at different Darcy numbers (Da) for $Ri = 100$, $Gr = 10^4$ and $\epsilon = 0.9$.

anti-clockwise direction is formed at the centre and two vortices are generated due to sliding lid which rotates in the clockwise direction (Fig. 6.17(i)). As Da is increased the size of the vortices due to sliding lids gets reduced, and each of them breakup into two vortices. The size of the vortex due to natural convection increases which clearly shows the dominance of natural convection over forced convection (Fig. 6.17(ii-iii)). Not surprisingly, the isotherms for higher Darcy number resemble those for the purely thermally driven square cavity flow problem of de Vahl Davis [30].

To understand the heat transfer effect the average Nusselt number is calculated at the cold (left) wall for various value of Da at $Ri = 100$ for both $Gr = 10^2$ and $Gr = 10^4$ and shown in Table 6.5. Two important observations can be made from Table 6.5. First it is obvious here that the increase in the Da value will bring about an increase in Nusselt number since more fluid can penetrate into the porous medium. Secondly an increase in Gr increases the $\overline{Nu}$ because Gr increases the convection.

Convergence Acceleration by Multigrid

This section describes the performance of the multigrid method for natural convection dominated flows. Fig. 6.18(a-c) shows the history of mass residual on a 130×130 grid as a

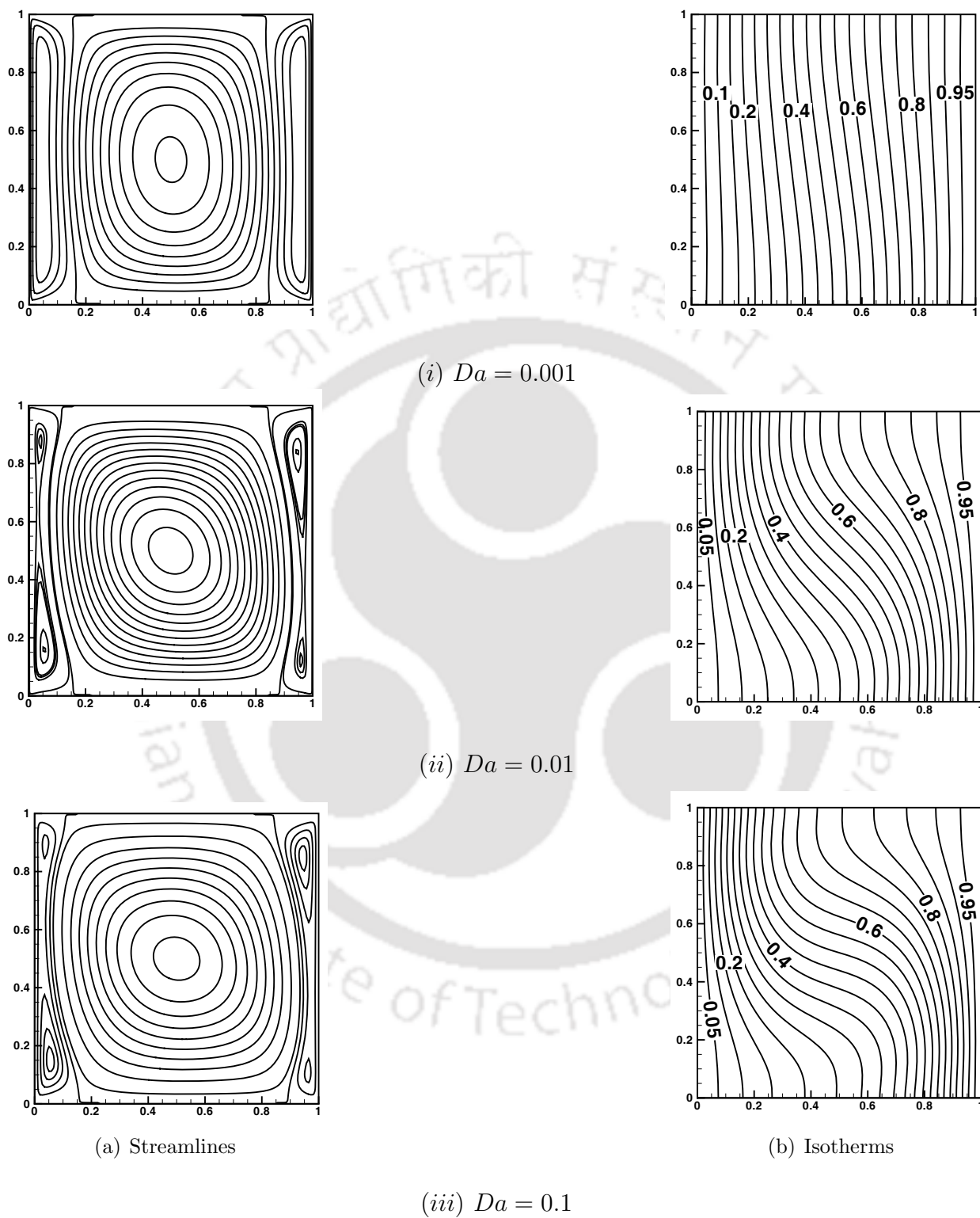


Fig. 6.17: (a) Streamline and (b) isotherm patterns for $Ri = 100$, $Gr = 10^4$, $\epsilon = 0.9$ and for various Darcy numbers (Da).

Table 6.5: Average Nusselt number $\overline{Nu}$ for various Darcy number and Grashof number at $Ri = 100$

Da	$Gr = 10^2$	$Gr = 10^4$
10^{-3}	1.0007	1.010104
10^{-2}	1.00080	1.285542
10^{-1}	1.0015	1.610664

function of number of work units at $Ri = 100$ for various Darcy numbers at $Gr = 10^4$ for single grid and 4-level multigrid computation. Similar to forced convection dominated flows, the FAS multigrid method in this case also shows a faster convergence than the single-grid method. Table 6.6 summarizes the total CPU time required and speed-up obtained by the multigrid over single grid method at $Ri = 100$ and $Gr = 10^4$ for various Darcy numbers. It is seen that multigrid method shows approximately 7 – 8 times faster than single grid method. In terms of computational efficiency, the 4-level multigrid method is substantially faster than single grid method. It is known that multigrid pay-off is higher for finer grids, which is why the multigrid technique is a very useful device for accelerating the convergence.

Table 6.6: CPU time required by the single-grid and multigrid method at $Ri = 100$, $Gr = 10^4$ for various Darcy numbers

Da	CPU time (min)		speed-up
	single grid	multigrid	
10^{-3}	66.06	8.48	7.51
10^{-2}	58.30	7.12	8.12
10^{-1}	49	7.42	6.36

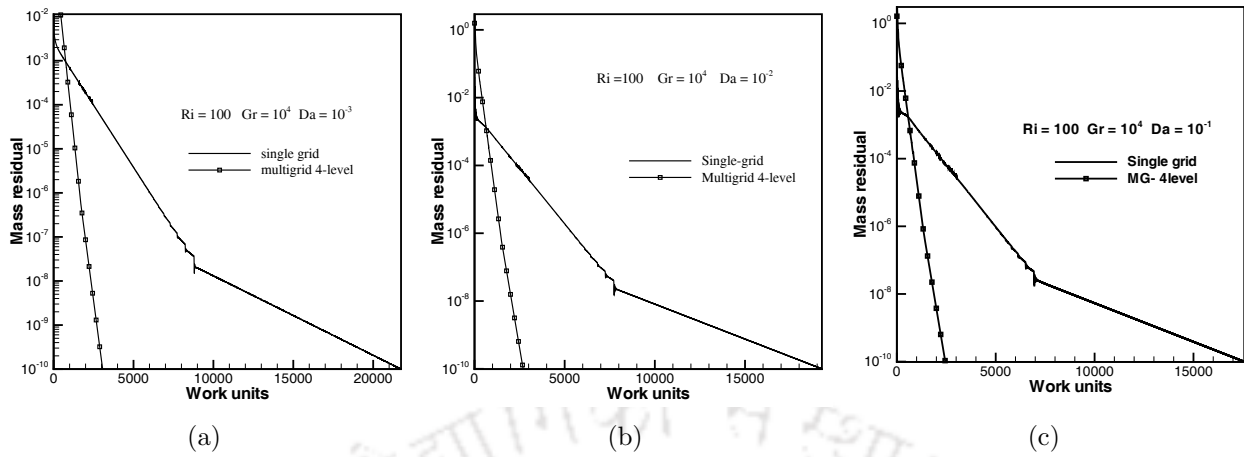


Fig. 6.18: Mass residual history of multigrid methods for $Ri = 100, Gr = 10^4$ and $\epsilon = 0.9$ (a) $Da = 0.001$ (b) $Da = 0.01$ (c) $Da = 0.1$ for BFDM.

6.5 Conclusions

The mixed convection in a square cavity with antiparallel wall motion and filled with a porous medium has been investigated using Brinkman-extended Darcy model and Brinkman-Forchheimer Darcy model to study the effect of the extra inertia term in the momentum equation present in the Forchheimer model. The direction of the plate movements have been so chosen as to oppose the buoyancy effects so that interesting flow patterns develop especially when both the effects have similar levels of influence. A multigrid accelerated computer code has been developed to compute the flow which has not been earlier investigated for the present geometry and the range of parameters using these models. To lend credibility to the results produced, a thorough code validation exercise has been carried out by using the code to reproduce various results available in literature. The present results then can be expected to enjoy a high level of credibility. It is generally seen in the present investigation that as Darcy number increases the discrepancy between the results given by the two models increases. The flow pattern, isotherm pattern and heat transfer pattern represented by the Nusselt number given by the two models for a wide range of parameters are presented. Some general observations can be made from these results. For the low value of Da and Gr Brinkman-extended Darcy model appear to be sufficient. However, for higher values of the parameters Brinkman-Forchheimer Darcy model is expected to give more accurate results

because of the hindering effect of the additional term. In case of $Ri = 0.01$ the vortex formed is completely due to the sliding lid and in case of $Ri = 100$ it is due to both sliding lids and natural convection.

The results plotted and tabulated are in keeping with the trend consistent with the physics of the flow that involves mixed convection. The average Nusselt number calculated is high for forced convection dominated flows relative to the natural convection dominated flows. Increase in Gr increases the effect of natural convection. The performance of the full approximation storage multigrid method, not frequently used in literature for similar type of problems is also investigated in some details. It is seen that the 4-level multigrid method employed here has the ability to speed-up the convergence considerably for the present problem involving both free and forced convection. Overall this chapter presents the results of high credibility for a relatively unexplored flow configuration using an efficient algorithm.

Chapter 7

Summary and Conclusions

This chapter describes the major achievements of the thesis and what future work can be taken up based on the experience gained from this work. Whenever an observation is made or an inference drawn, to prepare the background the work done is also briefly described.

A number of computer codes have been developed. The transient codes on graded Cartesian meshes computes mainly the following flows: (1) Steady and periodic flows in a single-sided lid-driven square cavity (2) Transient and steady flows in a two-sided 2D lid-driven square cavity, for which no earlier transient computations exist (3) Steady flow in a 2D backward-facing step in a channel (4) Flow past a square prism confined in a channel to study the vortex-shedding and, lift and drag behaviour (5) Transient flow in single- and two-sided cubical cavities. All the results for which previous data exist are validated through a rigorous comparison exercise. Sufficient care has been taken to obtain high accuracy in the other results which are obtained for the first time. These results are also produced with good time-wise and grid economy. The FAS multigrid codes are used to compute (1) thermal flows in a porous cavity and (2) multiple steady solution in two- and four-sided square cavity flows. As flows of various complexities are computed accurately and efficiently, the developed codes can be considered good numerical tools for flow computation.

In order to establish the necessity and usefulness of the linear multigrid in the transient fluid-flow computations, the numerical experiments for the 2D heat conduction problem

shows that though successive overrelaxation (SOR) with the optimum relaxation parameter substantially accelerates Gauss-Seidel iterations, linear multigrid does it even better and its performance relative to SOR improves with grid refinement. Thus the selection of multigrid as the accelerator is not arbitrary but carefully thought out.

As already mentioned the 2D and 3D multigrid-accelerated transient NS codes on graded Cartesian meshes use grid transformation to obtain second-order accuracy in time and at least third-order accuracy in space as convective and viscous terms are discretized to third- and fourth-order accuracy respectively. Performance of multigrid on uniform grid is slightly better than that on graded Cartesian meshes with the same meshsize. This fact, however, is not significant as it is seen that for the flows examined same quality of solution can be obtained using a graded Cartesian mesh with much fewer number of grid points.

The hitherto unexamined transient flow computation in the two-sided lid-driven square cavity demonstrates that flow at $Re = 2000$ asymptotically approaches the steady state, settling the issue as to whether the flow at $Re = 2000$ has a steady solution or not.

In the backward facing step flow problem, the issues as to whether the flow at $Re = 800$ and 1000 is steady or not is addressed and our computations with the transient code suggests that both the flows are steady.

The problem of flow over a square prism confined in a channel has many interesting features that include the famous von Kármán vortex street. The present work demonstrates that these flow features can be efficiently captured using the multigrid method. The demonstration of periodicity through the power spectrum analysis, phase plot and, the C_D and C_L vs time plot conclusively show the periodic nature of the flow. The C_D vs time plot is particularly interesting in that in a period there are two troughs and two crests of unequal magnitudes.

FAS multigrid is seen to produce speed-ups ranging from 7 to 10 for steady flows with or without thermal gradients. Linear multigrid used to solve the pressure-Poisson equation in transient flows that asymptotically move to the steady state, produces substantial acceleration (of the order of 21) in the initial phase; however the gain tapers off as time progresses.

For purely transient flows like those over a square prism, the gain produced by linear multigrid remains steady. Multigrid, therefore, is more useful for these later kind of flows which has no steady solution.

For the first time FAS multigrid on a colocated grid has been used to compute multiple steady solutions in two- and four-sided cavity flows, demonstrating that multigrid has the ability to compute flows in those class of problems that exhibit nonunique solutions.

FAS multigrid algorithm on a colocated uniform grid is used to compute the thermal flow in a 2D porous square cavity to show that Brinkman-Forchheimer Darcy model gives better results than Brinkman-extended Darcy model for higher Darcy numbers and at lower Darcy number both the models give similar results. Thus it may be concluded that for the flow configuration considered it is better to use the Brinkman-Forchheimer Darcy model that includes the extra inertia term. Probably this is the first instance where a multigrid algorithm has been effectively used to compute thermal flows in a cavity filled with a porous medium.

Multigrid method has been used to compute several 2D and 3D cavity flows with Dirichlet boundary conditions and also two channel flows that use Neumann boundary conditions. As the performance of multigrid is impressive for all the cases, it may be concluded that multigrid works well with both types of boundary conditions.

Overall, in the present work multigrid methods are used to carry out accurate and time-efficient computations not only for different types of already-examined flow configurations, but also for some hitherto-unexamined flows. Also the scope of using the algorithm has been extended to thermal flows in a porous cavity and flow configuration having nonunique solutions. Sufficient care has been taken to obtain highly accurate solutions. Performance of multigrid, which is quantified in terms of speed-ups for several problems is found to be impressive. The ability of multigrid to efficiently compute the wide variety of flows included in this thesis demonstrates not only the utility of the algorithm but also its robustness.

Finally, in course of what follows we describe some possible scopes for future work as the present study opens up some interesting possibilities for further investigations.

1. The 3D code developed for the present transient computations based on finite-difference method can be effectively used for other 3D rectangular geometries. Also the code may have the potential of being gainfully used for LES and DNS in those geometries that allow effective use of graded Cartesian meshes.
2. It is observed that the solver on nonuniform meshes takes more time for convergence as it is associated with more number of terms compared with uniform grid solvers. Hence parallelization of the code can be attempted to couple the advantages of high speed-up and grid economy.
3. The finite volume based FAS multigrid codes that use the SIMPLE algorithm can be extended to turbulent flows.

Appendix A

Transformation of Governing Equations

For incompressible flow the governing equations (3.1) - (3.3) are in the Cartesian coordinates and their general form can be written as

$$a\phi_{xx} + b\phi_{xy} + c\phi_{yy} + d\phi_x + e\phi_y + f\phi = g(x, y) \quad (\text{A.1})$$

where $g(x, y)$ denotes the source term. Here we present some details of the transformation. The transformation is of the form

$$x = x(\xi, \eta), \quad y = y(\xi, \eta) \quad (\text{A.2})$$

from the physical $x - y$ plane to the computational $\xi - \eta$ plane.

In the discretized N-S equations the derivatives are in the physical domain. These operators are related to those in the computational domain and the key is the simple chain rule of differentiation.

$$\frac{\partial \phi}{\partial x} = \left(\frac{\partial \xi}{\partial x} \right) \left(\frac{\partial \phi}{\partial \xi} \right) + \left(\frac{\partial \eta}{\partial x} \right) \left(\frac{\partial \phi}{\partial \eta} \right) \quad (\text{A.3})$$

A similar relation for the second derivative can be written as

$$\begin{aligned}\frac{\partial^2 \phi}{\partial x^2} &= \frac{\partial}{\partial x} \left(\frac{\partial \phi}{\partial x} \right) = \frac{\partial}{\partial x} \left(\frac{\partial \xi}{\partial x} \frac{\partial \phi}{\partial \xi} + \frac{\partial \eta}{\partial x} \frac{\partial \phi}{\partial \eta} \right) \\ &= \frac{\partial^2 \xi}{\partial x^2} \frac{\partial \phi}{\partial \xi} + \frac{\partial \xi}{\partial x} \frac{\partial^2 \phi}{\partial \xi \partial x} + \frac{\partial^2 \eta}{\partial x^2} \frac{\partial \phi}{\partial \eta} + \frac{\partial \eta}{\partial x} \frac{\partial^2 \phi}{\partial \eta \partial x}\end{aligned}\quad (\text{A.4})$$

Application of the chain rule to the second and fourth terms of Eq. (A.4) and rearrangement gives,

$$\frac{\partial^2 \phi}{\partial x^2} = \frac{\partial^2 \xi}{\partial x^2} \frac{\partial \phi}{\partial \xi} + \frac{\partial^2 \eta}{\partial x^2} \frac{\partial \phi}{\partial \eta} + \left(\frac{\partial \xi}{\partial x} \right)^2 \frac{\partial^2 \phi}{\partial \xi^2} + \left(\frac{\partial \eta}{\partial x} \right)^2 \frac{\partial^2 \phi}{\partial \eta^2} + 2 \frac{\partial \xi}{\partial x} \frac{\partial \eta}{\partial x} \frac{\partial^2 \phi}{\partial \xi \partial \eta} \quad (\text{A.5})$$

Similar equations can be written for the y derivatives. Under this transformation, Eq. (A.1) in the physical space assumes the following form in the computational space:

$$A \phi_{\xi\xi} + B \phi_{\xi\eta} + C \phi_{\eta\eta} = f(\phi_\xi, \phi_\eta, \phi, \xi, \eta) \quad (\text{A.6})$$

where,

$$A = a\xi_x^2 + b\xi_x\xi_y + c\xi_y^2 \quad (\text{A.7})$$

$$B = 2a\xi_x\eta_x + b(\xi_x\eta_y + \xi_y\eta_x) + 2c\xi_y\eta_y \quad (\text{A.8})$$

$$C = a\eta_x^2 + b\eta_x\eta_y + c\eta_y^2 \quad (\text{A.9})$$

Appendix B

Derivation of Pressure-Poisson Equation

The fourth-step of the fractional-step method (Eq. (3.11)) described in Section 3.3 is given by

$$\frac{(u_i^{n+1} - u_i^*)}{\Delta t} = -\frac{\partial p^{n+1}}{\partial x_i} \quad (\text{B.1})$$

where u_i^* (i.e., u^* and v^*) represents the intermediate velocity field. Taking divergence of Eq. (B.1) yields

$$\frac{\partial u_i^{n+1}}{\partial x_i} - \frac{\partial u_i^*}{\partial x_i} = -\Delta t \left(\frac{\partial^2 p^{n+1}}{\partial x_i \partial x_i} \right) \quad (\text{B.2})$$

Eq. (B.2) is subjected to continuity constraint for $(n+1)$ time-step, i.e., $\frac{\partial u_i^{n+1}}{\partial x_i} = 0$. Therefore Eq. (B.2) becomes

$$\frac{\partial^2 p}{\partial x_i \partial x_i} = \frac{1}{\Delta t} \frac{\partial u_i^*}{\partial x_i} \quad (\text{B.3})$$

Eq. (B.3) is the pressure-Poisson equation.

Appendix C

Power Spectrum Analysis

The power spectral density (PSD) is estimated using the fast Fourier transform (FFT) algorithm which describes how a time series is distributed with frequency. If we represent the time series of a particular dynamical variable by

$$Y^{(k)}; \quad k = 0, 1, \dots, N - 1 \quad (\text{C.1})$$

Then the discrete Fourier transform of the N points $Y^{(k)}$ is given by

$$\hat{Y}^{(l)} = \sum_0^{N-1} Y^{(k)} e^{\frac{2\pi jkl}{N}} \quad (\text{C.2})$$

where, $j = \sqrt{-1}$. For a real time series, we need to consider only $l = 0, 1, \dots, N/2$. The transformation (Eq. C.2) can be carried out efficiently by using a fast Fourier transform (FFT) algorithm [89]. For this purpose, the number of points N in the time series (Eq. C.1) must be a power of 2.

The periodogram estimate of the power spectral density (PSD) at $(\frac{N}{2} + 1)$ frequency f_l

is then defined as

$$\begin{aligned} PSD(0) &\equiv PSD(f_0) = \left| \hat{Y}^{(0)} \right|^2 \\ PSD(f_l) &= 2 \left| \hat{Y}^{(l)} \right|^2, \quad l = 1, 2, \dots, \left(\frac{N}{2} - 1 \right) \\ PSD(f_c) &\equiv PSD(f_{\frac{N}{2}}) = \left| \hat{Y}^{(\frac{N}{2})} \right|^2 \end{aligned} \quad (C.3)$$

where

$$f_l = \frac{l}{N\Delta t}, \quad l = 0, 1, \dots, \frac{N}{2} \quad (C.4)$$

Δt = sampling interval for the time series (Eq. C.1), and

$f_c = \frac{1}{2\Delta t}$, the Nyquist frequency.

By Parseval's theorem, Eq. (C.4) is normalized so that the sum of the $\frac{N}{2} + 1$ values of PSD is equal to the mean squared amplitude of the function $Y^{(k)}$. Thus the discrete form of Parseval's equation is written as

$$N \sum_{k=0}^{N-1} |Y^{(k)}|^2 = \sum_{l=0}^{\frac{N}{2}} PSD(f_l) \quad (C.5)$$

Eq. (C.5) is useful in checking overall accuracy of the transformation as well as in detecting any error due to misinterpretation of the output array from an FFT algorithm. The latter possibility arises as the input real array is replaced by an output array containing components of a complex transform with some re-arrangement which can be implementation dependent.

While data filtering or smoothing of the spectral estimates is generally not required in numerical simulation studies of dynamical systems governed by ODEs, careful attention must be paid to aliasing the frequency resolution. All the spectral power that lies outside the Nyquist frequency range is aliased (falsely translated or folded back) into that range by discrete sampling. Thus the sampling interval Δt must be such that the higher frequency harmonics beyond $f_c = \frac{1}{2\Delta t}$ are negligible. If f^* denotes characteristic fundamental frequency associated with the time series, then Δt is selected such that $\frac{f_c}{f^*} = L$ is a large

number.



References

- [1] A.M. Al-Amiri. Analysis of momentum and energy transfer in a lid-driven cavity filled with a porous medium. *International Journal of Heat and Mass Transfer*, 43:3513–3527, 2000.
- [2] S. Albensoeder and H.C. Kuhlmann. Accurate three-dimensional lid-driven cavity flow. *Journal of Computational Physics*, 206:536–558, 2005.
- [3] S. Albensoeder, H.C. Kuhlmann, and H.J. Rath. Multiplicity of steady two-dimensional flows in two-sided lid-driven cavities. *Theoretical and Computational Fluid Dynamics*, 14:223–241, 2001.
- [4] N. Alleborn, H. Raschiller, and F. Durst. Lid-driven cavity with heat and mass transport. *International Journal of Heat and Mass Transfer*, 42:833–853, 1999.
- [5] B.F. Armaly, F. Durst, J.C.F. Pereira, and B. Schonung. Experimental and theoretical investigation of backward-facing step flow. *Journal of Fluid Mechanics*, 127:473–496, 1983.
- [6] S. Armfield and R. Street. Modified fractional-step methods for the Navier-Stokes equations. *ANZIAM*, 45(E):C364–C377, 2004.
- [7] F. Auteri, L. Quartapelle, and L. Vigevano. Accurate $\psi - \zeta$ spectral solution of the singular driven cavity problem. *Journal of Computational Physics*, 180:597–615, 2002.
- [8] E. Barragy and G.F. Carey. Streamfunction vorticity driven cavity solution using p finite elements. *Computers and Fluids*, 26(5):453–468, 1997.

- [9] T. Basak, S. Roy, T. Paul, and I. Pop. Natural convection in a square cavity filled with a porous medium: Effects of various thermal boundary conditions. *International Journal of Heat and Mass Transfer*, 49:1430–1441, 2006.
- [10] B.B. Beya and T. Lili. Three-dimensional incompressible flow in a two-sided non-facing lid-driven cubical cavity. *Comptes Rendus Mecanique*, 336:863–872, 2008.
- [11] B. Botella and R. Peyret. Benchmark spectral results on the lid-driven cavity flow. *Computers and Fluids*, 27:421–433, 1998.
- [12] A. Brandt. Multilevel adaptive solutions to boundary-value problems. *Mathematics of Computation*, 31:333–390, 1977.
- [13] M. Breuer, J. Bernsdorf, T. Zeiser, and F. Durst. Accurate computations of the laminar flow past a square cylinder based on two different methods: lattice-Boltzmann and finite volume. *International Journal of Heat and Fluid Flow*, 21:186–196, 2000.
- [14] W.L. Briggs. *A multigrid tutorial*. SIAM, second edition, 1987.
- [15] H.C. Brinkman. A calculation of the viscous force exerted by a flowing fluid in a dense swarm of particles. *Applied Science Research*, A1:27–34, 1947.
- [16] C.H. Bruneau and C. Jouron. An efficient scheme for solving steady incompressible Navier-Stokes equations. *Journal of Computational Physics*, 89:389–413, 1990.
- [17] C.H. Bruneau and M. Saad. The 2D lid-driven cavity problem revisited. *Computers and Fluids*, 35:326–348, 2006.
- [18] D. Calhoun. A cartesian grid method for solving the two-dimensional streamfunction-vorticity equations in irregular regions. *Journal of Computational Physics*, 176:231–275, 2002.
- [19] N.B. Cheikh, B.B. Beya, and T. Lili. Benchmark solution for time-dependent natural convection flows with an accelerated full-multigrid method. *Numerical Heat Transfer, Part B*, 52:131–151, 2007.

- [20] N.B. Cheikh, B.B. Beya, and T. T Lili. Aspect ratio effect on natural convection flow in a cavity submitted to a periodical temperature boundary. *Journal of Heat Transfer*, 129:1060–1068, 2007.
- [21] H. Choi and P. Moin. Effects of the computational time step on numerical solutions of turbulent flow. *Journal of Computational Physics*, 113:1–4, 1994.
- [22] A.J. Chorin. A numerical method for solving incompressible viscous flow problem. *Journal of Computational Physics*, 2:12–26, 1967.
- [23] A.J. Chorin. A numerical solution of the Navier-Stokes equations. *Mathematics of Computation*, 22:745–762, 1968.
- [24] C. Cornelius, W. Volgmann, and H. Stoff. Calculation of three dimensional turbulent flow with a finite volume multigrid method. *International Journal for Numerical Methods in Fluids*, 31:703–720, 1999.
- [25] M.A. Cruchaga. A study of the backward-facing step problem using a generalized streamline formulation. *Communications in Numerical Methods in Engineering*, 14:697–708, 1998.
- [26] H. Darcy. Les Fontaines Publiques de la ville de Dijon,. *Victor Dalmond, Paris*, 1856.
- [27] R.W Davis and E.F. Moore. A numerical study of vortex shedding from rectangles. *Journal of Fluid Mechanics*, 116:475–506, 1982.
- [28] R.W. Davis, E.F. Moore, and L.P. Purtell. A numerical-experimental study of confined flow around rectangular cylinders. *Physics of Fluids*, 27:46–59, 1984.
- [29] A.K. De and A. Dalal. Numerical study of laminar forced convection fluid flow and heat transfer from a triangular cylinder placed in a channel. *Transcations of the ASME*, 129:646–656, 2007.

- [30] G. de Vahl Davis. Natural convection of air in a square cavity: a bench mark numerical solution. *International Journal for Numerical Methods in Fluids*, 3:249–264, 1983.
- [31] H. Ding, C. Shu, K.S. Yeo, and C. Xu. Numerical computation of three-dimensional incompressible viscous flows in the primitive variable form by local multiquadric differential quadrature method. *Computer Methods in Applied Mechanics and Engineering*, 195:516–533, 2006.
- [32] E. Erturk, T.C. Corke, and C. Gokcol. Numerical solutions of 2-D steady incompressible driven cavity flow at high Reynolds numbers. *International Journal for Numerical Methods in Fluids*, 48(7):747–774, 2005.
- [33] R.P. Fedorenko. A relaxation method for solving elliptic difference equations. *Computational Mathematics and Mathematical Physics*, 1:1092–1096, 1962.
- [34] R.P. Fedorenko. The speed of convergence of one iteration process. *Computational Mathematics and Mathematical Physics*, 4:227–235, 1964.
- [35] H. Foester and K. Witsch. On efficient multigrid software for elliptic problems on rectangular domain. *Mathematics and Computers in Simulation*, XXIII:293–298, 1981.
- [36] P. Forchheimer. Wasserbewegung durch Boden. *Zeitschrift des Verines Deutscher Ingenieure*, 45:1781–1788, 1901.
- [37] A. Fortin, M. Jardak, J.J. Gervais, and R. Pierre. Localization of Hopf bifurcations in fluid flow problems. *International Journal for Numerical Methods in Fluids*, 24(11):1185–1210, 1997.
- [38] R. Franke, W. Rodi, and B. Schonung. Numerical calculation of laminar vortex shedding flow past cylinders. *Journal of Wind Engineering and Industrial Aerodynamics*, 35:237, 1990.

- [39] L. Fuchs. Multigrid solution of the Navier-Stokes equation on nonuniform grids. In *Multigrid methods, NASA Conference Publication 2202, Ames research Center, Moffett Fields, CA*, pages 83–100, 1981.
- [40] D.K. Gartling. A test problem for outflow boundary conditions - flow over a backward-facing step. *International Journal for Numerical Methods in Fluids*, 11:953–967, 1990.
- [41] U. Ghia, K.N. Ghia, and C.T. Shin. High-Resolutions for incompressible Navier-Stokes equation and a multigrid method. *Journal of Computational Physics*, 48:387–411, 1982.
- [42] P.M. Gresho, D.K. Gartling, J.R. Torczynski, K.A. Cliffe, K.H. Winters, T.J. Garratt, A. Spence, and J.W. Goodrich. Is the steady viscous incompressible two-dimensional flow over a backward-facing step at $re = 800$ stable ? *International Journal for Numerical Methods in Fluids*, 17:501–541, 1993.
- [43] J.L. Guermond, C. Migeon, G. Pineau, and L. Quartapelle. Start-up flows in a three dimensional rectangular driven cavity fo aspect ratio 1:1:2 at $Re = 1000$. *Journal of Fluid Mechanics*, 450:169–199, 2002.
- [44] W. Hackbusch. On the multigrid method applied to difference equation. *Computing*, 20:291–306, 1978.
- [45] W. Hackbusch. *Multigrid Methods and Applications*. Springer-verlag, Berlin, 1985.
- [46] L.P. Hackman, G.D. Raithby, and A.B. Strong. Numerical predictions of flows over backward-facing steps. *International Journal for Numerical Methods in Fluids*, 4:711–724, 1984.
- [47] F.H. Harlow and J.E. Welch. Numerical calculation of time-dependent viscous incompressible flow of fluid with free surface. *Physics of Fluids*, 8:2182–2189, 1965.
- [48] T. Hayase, J. A. C. Humphrey, and R. Grief. A consistently formulated QUICK scheme for fast and stable convergence using finite-volume iterative calculation procedures. *Journal of Computational Physics*, 98:108–118, 1992.

- [49] M. Hortmann and M. Peric. Finite volume multigrid prediction of laminar natural convection: Bench-mark solutions. *International Journal for Numerical Methods in Fluids*, 11:189–207, 1990.
- [50] R.R. Hwang and C.C. Yao. A numerical study of vortex shedding from a square cylinder with ground effect. *Journal of fluids engineering*, 119:512–518, 1997.
- [51] A. Jameson. Solution of the euler equations by a multigrid method. *Applied Mathematics and Computation*, 13:327–356, 1983.
- [52] A. Jameson. Multigrid algorithms for compressible flow calculations. In W. Hackbush and V. Trottenberg, editors, *Proceedings of the 2nd European conference on multigrid methods*, Cologne, Springer- verlag, 1985.
- [53] P. Johansson and L. Davidson. A full multigrid method applied to turbulent flow using the SIMPLEC algorithm together with a colocated arrangement. *Multigrid Methods IV. Proceedings of the Fourth European Multigrid Conference*,, 116:245–256, 1994.
- [54] T.C. Jue. Analysis of Benard convection in rectangular cavities filled with a porous medium. *Acta Mechanica*, 146:21–29, 2001.
- [55] J.C. Kalita, D.C. Dalal, and A.K. Dass. Fully compact higher-order computation of steady-state natural convection in a square cavity. *Physical Review E*, 64 (066703):1–13, 2001.
- [56] J.C. Kalita, D.C. Dalal, and A.K. Dass. A class of higher order compact schemes for the unsteady two-dimensional convection diffusion equation with variable convection coefficients. *International Journal for Numerical Methods in Fluids*, 38(11):1111–1131, 2002.
- [57] J.C. Kalita, A.K. Dass, and D.C. Dalal. A transformation-free HOC scheme for steady convectiondiffusion on nonuniform grids. *International Journal for Numerical Methods in Fluids*, 44:33–53, 2004.

- [58] J.C. Kalita and M.M. Gupta. A streamfunction-velocity approach for 2D transient incompressible viscous flows. *International Journal for Numerical Methods in Fluids*, 62:237–266, 2010.
- [59] P.R. Kanna and M.K. Das. A short note on the reattachment length for bfs problem. *International Journal for Numerical Methods in Fluids*, 50:683–692, 2006.
- [60] T. Kawamura and K. K Kuwahara. Computation of high Reynolds number flow around a circular cylinder with surface roughness. *AIAA Journal*, 84:0340, 1984.
- [61] K.M. Kelkar and S.V. Patankar. Numerical prediction of vortex shedding behind a square cylinder. *International Journal for Numerical Methods in Fluids*, 14:327–341, 1992.
- [62] K. Khanafer and A.J. Chamkha. Mixed convection flow in a lid-driven enclosure filled with a fluid-saturated porous medium. *International Journal of Heat and Mass Transfer*, 42:2465–2481, 1999.
- [63] K. Khanafer and K. Vafai. Double-diffusive mixed convection in a lid-driven enclosure filled with a fluid-saturated porous medium. *Numerical Heat Transfer: Part A*, 42:465–486, 2002.
- [64] J. Kim and P. Moin. Application of a fractional-step method to incompressible Navier-Stokes equation. *Journal of Computational Physics*, 59:308–323, 1985.
- [65] J.R. Koseff and R.L. Street. On endwall effects in a lid-driven cavity flow. *Transaction ASME: Journal of Fluids Engineering*, 106:385–389, 1984.
- [66] J.R. Koseff, R.L. Street, P.M. Gresho, C.D. Upson, J.A.C. Jumphrey, and W.M. To. A three-dimensional lid-driven cavity flow: experiment and simulation. *Proceedings 3rd international Conference on Numerical Methods in Laminar and Turbulent Flow*, pages 564–581, 1983.

- [67] H.C. Ku, R.S. Hirsh, and T.D. Taylor. A pseudospectral method for solution of the three-dimensional incompressible Navier-Stokes equations. *Journal of Computational Physics*, 70:439–462, 1987.
- [68] H.C. Kuhlmann, M. Wanschura, and H.J. Rath. Flow in two-sided lid-driven cavities: Non-uniqueness, instabilities, and cellular structures. *Journal of Fluid Mechanics*, 336:267–299, 1997.
- [69] H.C. Kuhlmann, M. Wanschura, and H.J. Rath. Elliptic instability in two-sided lid-driven cavity flow. *European Journal of Mechanics - B/Fluids*, 17:561–569, 1998.
- [70] N. Kumar, J.C. Kalita, and A.K. Dass. HOC computation of two-sided lid-driven cavity flow. *International conference on computational mechanics and simulation, Proceedings IIT Guwahati*, Vol II:1695–1700, 2006.
- [71] J.L. Lage. Effect of the convective inertia term on Benard convection in a porous medium. *Numerical Heat Transfer: Part A*, part A 22:469–485, 1992.
- [72] G. Lauriat and V. Prasad. Non-Darcian effects on natural convection in a vertical porous enclosure. *International Journal of Heat and Mass Transfer*, 32(11):2135–2148, 1989.
- [73] H. Le and P. Moin. An improvement of fractional-step methods for the incompressible Navier-Stokes equations. *Journal of Computational Physics*, 92(2):369–379, 1991.
- [74] B. Leonard. A stable and accurate convective modeling procedure based on quadratic upstream interpolation. *Computing Methods in Applied Mechanics and Engineering*, 19:59–98, 1979.
- [75] F.S. Lien and M.A. Leschziner. Multigrid acceleration for recirculating laminar and turbulent flows computed with a non-orthogonal collocated finite volume scheme. *Computational Methods in Applied Mechanics and Engineering*, 118:351–371, 1994.

- [76] Z. Lilek, S. Muzaferija, and M. Peric. Efficiency and accuracy aspects of a full-multigrid SIMPLE algorithm for three-dimensional flows. *Numerical Heat Transfer: Part A*, 31:23–42, 1997.
- [77] W.J. Luo and R.J. Yang. Multiple fluid flow and heat transfer solutions in a two-sided lid-driven cavity. *International Journal of Heat and Mass Transfer*, 50:2394–2405, 2007.
- [78] R. Mei, W. Shyy, D. Yu, and L.S. Luo. Lattice Boltzmann method for 3-D flows with curved boundary. *Journal of Computational Physics*, 161:680–699, 2000.
- [79] T.F. Miller and F.W. Schmidt. Evaluation of a multilevel technique applied to the poisson and navier-stokes equations. *Numerical Heat Transfer: Part A*, 13:1–26, 1988.
- [80] A. Mukhopadhyay, G. Biswas, and T. Sundararajan. Numerical investigation of confined wakes behind a square cylinder in a channel. *International Journal for Numerical Methods in Fluids*, 14:1473–1484, 1992.
- [81] M.J. Ni, S. Komori, and N. Morley. Projection methods for the calculation of incompressible unsteady flows. *Numerical Heat Transfer, Part B*, 44:533–551, 2003.
- [82] D.A. Nield and A. Bejan. *Convection in porous media*. Springer, 1992.
- [83] P. Nithiarasu, K.N. Seetharamu, and T. Sundararajan. Natural convective heat transfer in a fluid saturated variable porosity medium. *International Journal of Heat and Mass Transfer*, 40(16):3955–3967, 1997.
- [84] A. Okajima. Strouhal numbers of rectangular cylinders. *Journal of Fluid Mechanics*, 123:379–398, 1982.
- [85] S.V. Patankar. *Numerical Heat Transfer and Fluid Flow*. Taylor and Francis, 1980.
- [86] S.V. Patankar and D.B. Spalding. A calculation procedure for heat, mass and momentum transfer in three-dimensional parabolic flows. *International Journal of Heat and Mass Transfer*, 15:1787–1806, 1972.

- [87] D.A. Perumal and A.K. Dass. Simulation of flow in two-sided lid-driven square cavities by the lattice Boltzmann method. *International conference on advances in fluid mechanics VII; proceedings, WIT United Kingdom, May*, pages 45–54, 2008.
- [88] R Peyret and T.D. Taylor. *Computational methods for fluid flow*. springer, New York, 1983.
- [89] W. H. Press, S. A. Teukolsky, W.T. Wetterling, and B.P. Flannery. *Numerical Recipes in C*. Cambridge University Press, 1993.
- [90] A.R. Rahmati and M. Ashrafizaadeh. A generalized lattice Boltzmann method for three-dimensional incompressible fluid flow simulation. *Journal of Applied Fluid Mechanics*, 2:71–96, 2009.
- [91] M.M. Rai and P. Moin. Direct simulations of turbulent flow using finite-difference schemes. *Journal of Computational Physics*, 96:15–53, 1991.
- [92] M. Ramsak and L. Skerget. A subdomain boundary element method for high-Reynolds laminar flow using stream function-vorticity formulation. *International Journal for Numerical Methods in Fluids*, 46:815–847, 2004.
- [93] C.M. Rhie and W.L. Chow. A numerical study of the turbulent flow past an isolated airfoil with trailing edge separation. *AIAA Journal*, 21:1525–1532, 1983.
- [94] A. Roy and G. Bandyopadhyay. Numerical computation of flow past a rectangular cylinder placed at different ground clearances from a fixed wall. *Journal of Aerospace Sciences and Technologies*, 55:112–117, 2003.
- [95] U. Ruede. Multigrid tutorials. <http://www.mgnet.org/mgnet/tutorials/xwb/cycles.html>, 1995.
- [96] A.K. Saha, K. Muralidhar, and G. Biswas. Vortex structures and kinetic energy budget in two-dimensional flow past a square cylinder. *Computers and Fluids*, 29:669–694, 2000.

- [97] P.N. Shankar and M.D. Deshpande. Fluid mechanics in the driven cavity. *Annual Review Fluid Mechanics*, 32:93–136, 2000.
- [98] J. Shen. Hopf bifurcation of the unsteady regularized driven cavity flow. *Journal of Computational Physics*, 95:228–245, 1991.
- [99] S. Sivaloganathan and G.J. Shaw. A multigrid method for recirculating flows. *International Journal for Numerical Methods in Fluids*, 8:417–440, 1988.
- [100] A. Sohankar, C. Norberg, and L. Davidson. *A numerical study of unsteady two-dimensional flow around rectangular cylinders at incidence*. PhD thesis, Chalmers University of Technology-Gothenburg, Sweden, 1996.
- [101] J.L. Sohn. Evaluation of FIDAP on some classical laminar and turbulent benchmarks. *International Journal for Numerical Methods in Fluids*, 8:1469–1490, 1988.
- [102] H.L. Stone. Iterative solution of implicit approximations of multidimensional partial differential equations. *SIAM Journal of Numerical Analysis*, 5:530–558, 1968.
- [103] K. Stuben and U. Trottenberg. *Multigrid methods: fundamental algorithms, model problem analysis and applications*, in *Lecture Notes in Mathematics 960*, ed. W. Hackbusch and U. Trottenberg. Springer, Berlin, 1982.
- [104] S. Sunderesan. *Clustered grids and mesh independence in numerical simulation of 2D lid driven cavity flow*. PhD thesis, Indian Institute of Science Bangalore, 1996.
- [105] L.Q. Tang, T. Cheng, and T.T.H. Tsang. Transient solutions for three-dimensional lid-driven cavity flows by a least-squares finite element method. *International Journal for Numerical Methods in Fluids*, 21:413–432, 1995.
- [106] J.C. Tannehill, D.A. Anderson, and R.H. Pletcher. *Computational fluid mechanics and heat transfer*. Hemisphere Publishing Corporation, New York, second edition, 1984.

- [107] U. Trottenberg, C. Oosterlee, and A. Schuller. *Multigrid*. Academic-Press, London, 2001.
- [108] S. Ul-Islam and C.Y. Zhou. Characteristics of flow past a square cylinder using lattice Boltzmann method. *Information Technology Journal*, 8:1094–1114, 2009.
- [109] K. Vafai and C.L. Tien. Boundary and inertia effects on flow and heat transfer in porous media. *International Journal of Heat and Mass Transfer*, 24:195–203, 1981.
- [110] S.P. Vanka. Block-implicit multigrid calculation of two-dimensional recirculating flows,. *Computer Methods in Applied Mechanics and Engineering*, 59:29–48, 1986.
- [111] S.P. Vanka. Block-implicit multigrid solution of navierstokes equations in primitive variables,. *Journal of Computational Physics*, 65:138–158, 1986.
- [112] E. Vishnuvaradhanarao and M.K. Das. Laminar mixed convection in a parallel two-sided lid driven differentially heated square cavity filled with a fluid-saturated porous medium. *Numerical Heat Transfer: Part A*, 53:88–110, 2008.
- [113] E.M. Wahba. Multiplicity of states for two-sided and four-sided lid-driven cavity flows. *Computers and Fluids*, 38:247–253, 2009.
- [114] D.C. Wan, B.S.V. Patnaik, and B. S. V. Wei. Discrete singular convolutionfinite subdomain method for the solution of incompressible viscous flows. *Journal of Computational Physics*, 180:229255, 2002.
- [115] P. Wesseling. A robust and efficient multigrid method. *Lecture Notes in Mathematics*, 960, Springer-Verlag, New York:614–630, 1981.
- [116] X. Wu, K.D. Squires, and Q. Wang. Extension of the fractional-step method to general curvilinear coordinate systems. *Numerical Heat Transfer: Part A*, 27(part B):175–194, 1995.

- [117] J. Yan and F. Thiele. Performance and accuracy of a modified full multigrid algorithm for fluid flow and heat transfer. *Numerical Heat Transfer, Part B*, 34:323–338, 1998.
- [118] H.C. Yee, J.R. Torczynski, S.A. Morton, M.R. Visbal, and P.K. Sweby. On spurious behaviour of CFD simulations. *International Journal for Numerical Methods in Fluids*, 30:675–711, 1999.
- [119] O. Younis, J. Pallares, and F.X. Grau. Numerical study of transient laminar natural convection cooling of high Prandtl number fluids in a cubical cavity: Influence of the Prandtl number. *Proceedings of World Academy of Science, Engineering and Technology*, 22:ISSN 1307–6884, 2007.
- [120] J. Zhu. The second-order projection method for the backward-step facing step flow. *Journal of Computational Physics*, 117:318–331, 1995.
- [121] Z.J. Zhu and H.X. Yang. Numerical investigation of transient laminar natural convection of air in a tall cavity. *Heat Mass Transfer*, 39(7):579–587, 2003.

List of Publications from Present Thesis

Journals

1. D. Santhosh Kumar, Anoop K Dass and Anupam Dewan, Analysis of non-Darcy models for mixed convection in a cavity using multigrid approach, *Numerical Heat Transfer: Part A*, 56:685-708, 2009.
2. D. Santhosh Kumar, Anoop K Dass and Anupam Dewan, Multigrid Accelerated Computation of 2D Time-Dependent Incompressible Viscous Flows, *Engineering Applications of Computational Fluid Mechanics*, 4:71-90, 2010.
3. D. Santhosh Kumar, Anoop K Dass and Anupam Dewan, A multigrid-accelerated three-dimensional transient-flow code and its application to a new test problem, *Journal of Hydrodynamics* (in press).
4. D. Santhosh Kumar, Anoop K Dass and Anupam Dewan, Multiple of stable solutions for a two-sided and four-sided lid-driven cavity flow using FAS multigrid method, in *Applied Mathematical Modelling* (submitted).

Refereed Conferences

1. D. Santhosh Kumar., Anoop K Dass and Anupam Dewan, Performance of multigrid method in a mixed convection porous cavity using Brinkmann-Forchheimer model, Proceeding of 2nd National Conference on CFD applications in Power and Industry Sectors, BHEL Hyderabad, Jan-2009.
2. D. Santhosh Kumar., Anoop K Dass and Anupam Dewan, Numerical Simulation Of Viscous Flow Over A Square Cylinder On Graded Cartesian Meshes Using Multigrid

Method , Proceedings of 37th National and 4th International Conference on Fluid Mechanics and Fluid Power, Indian Institute of Technology Madras, Chennai, Dec-2010 (accepted).

