

Characterization of Fluid Flow and Heat Transfer in Rotating Convection Systems

by

Deepak Saini

ORCID:0000-0003-4499-824X

A thesis submitted in total fulfillment for the
degree of Doctor of Philosophy

in the

Department of Mechanical Engineering
Melbourne School of Engineering

THE UNIVERSITY OF MELBOURNE

April 2021

Abstract

Rotating flows and induced convection by centrifugal forces are prevalent in nature and also encountered in several engineering applications. This study is inspired by an engineering problem with relevance to the modern gas-turbine engine. In an internal air system of a high-pressure compressor, cavities form between the co-rotating compressor disks. The fluid flow due to the rotation of these cavities and the induced heat transfer rate from the disks directly affects the operational efficiency and safety of a gas turbine engine.

The fluid flow inside these cavities exhibits complex phenomena as it involves a multitude of control parameters such as Rayleigh number, which defines the strength of the buoyancy-induced force to viscous force, the Rossby number, which characterizes the significance of the Coriolis forces induced due to the rotation and the flow Mach number. Direct numerical simulations for a closed cavity configuration are conducted with a systematic variation of parameters to scrutinize their effects on the flow quantities and the heat transfer rate. An interplay between these parameters is further investigated by conducting the linear-stability analyses to find the critical values of these parameters at the onset of convection. The investigations show that compressibility effects lead to a reduction of the growth rate of the dominant mode, and it modifies the overall formation of convection cells in the cavity. A stability criterion is proposed that showed that both Rossby and Mach number affect the stability of flow in the rotating cavities. A parametric study of Rossby number suggests that a small change in the magnitude of Coriolis forces directly influences the behaviour of convection plumes and induced heat transfer rate. The near-wall temperature profiles in the plume-ejection region exhibit the logarithmic behaviour analogous to turbulent shear flows and reconcile the transition to the ultimate regime of convection with an increased power exponent of $Nu-Ra$ scaling law.

Further, first-ever high-fidelity simulations are carried out for a cavity with an axial-through flow, which closely represents the internal air cooling system of a gas turbine engine, to shed light into the flow structure that previously was poorly understood. The results show a flow structure dominated by a toroidal vortex in the inner region of the cavity. In the remaining outer region of the cavity, radial flares emerge from the toroidal vortex which forms cyclonic and anti-cyclonic vortices and are found to be responsible for thermal mixing and heat transfer

from the side disks. In the outer region, the flow is observed to move radially outwards by Ekman layers formed on the side disks. The near shroud region is mostly dominated by the centrifugal buoyancy-induced flow and the Nusselt number values, which measure the effectiveness of convection heat transfer inside the cavity, highlights the inefficacy of axial-through flow to enhance the overall heat transfer rate.

Declaration of Authorship

I, DEEPAK SAINI, declare that this thesis titled, ‘CHARACTERIZATION OF FLUID FLOW AND HEAT TRANSFER IN ROTATING CONVECTION SYSTEMS’ and the work presented in it are my own. I confirm that:

- The thesis comprises only my original work towards the PhD at the University of Melbourne;
- due acknowledgement has been made in the text to all other material used; and
- the thesis is fewer than 100,000 words limit in length, exclusive of tables, maps, bibliographies and appendices as approved by the Research Higher Degrees Committee.

Deepak Saini

Preface

The publication status of all chapters presented in this thesis is presented below

- Chapters 1-3: Unpublished material not submitted for publication.
- Chapter 4: part of the chapter published in the International Journal of Heat and Fluid Flow on 2nd July 2020 and remainder in the conference proceedings of the Australasian Fluid Mechanics Conference 2018 on 13th December 2018 and in
- Chapter 5: part of the published in the International Journal of Heat and Fluid Flow on 2nd July 2020 and remainder in the conference proceedings of the Australasian Fluid Mechanics Conference 2020 on 10th December 2020.
- Chapter 6: Unpublished material not submitted for publication.
- Chapter 7: Published in ASME Turbomachinery Technical Conference and Exposition on 21st September 2020 and under production for publication in the Journal of Turbomachinery.

As the first author, I was responsible for setting up and running the simulations, analyzing and interpreting the data. The material contained in the publications is the original research work conducted by me during my PhD. I was primarily responsible for planning and preparation of the initial draft of the publications and editing in response to co-authors and reviewers feedback. The publications are not subject to any agreement or contract with the third party, which prohibit to include the material into this thesis.

I would like to acknowledge funding from the Melbourne Research Scholarship to support my PhD studies and computing resources provided by the Pawsey Supercomputing Centre with funding from the Australian Government and the Government of Western Australia. This research was also undertaken using the resources and services from the National Computational Infrastructure (NCI), which is supported by the Australian Government and LIEF HPC-GPGPU Facility hosted at the University of Melbourne which was established with the assistance of LIEF Grant LE170100200.

Acknowledgements

Foremost, I would like to thank my supervisor, Professor Richard Sandberg, for his continuous support, guidance and motivation throughout my PhD journey. His encouragement, technical expertise and lively discussions made my research tenure fascinating and enjoyable. I truly appreciate his support over the years, which allowed me to think independently and finish my thesis successfully. I am grateful to my co-supervisor, A/Professor Daniel Chung, for his valuable advice, suggestions and various stimulating discussions on the formulation of my research problem. I am incredibly grateful and honoured to work with them.

A special thanks to my PhD committee members, Professor Ivan Marusic and Dr. Simon Illingworth, for their invaluable suggestions and recommendations during my review meetings. I also acknowledge the worthy support from other academics and administration staff of the Department of Mechanical Engineering.

I am very grateful to Dr. Chitrarth Lav for spending hours on reading my thesis and providing useful suggestions. I would like to acknowledge the help I received from the Turbulence Simulation and Modelling Group throughout my PhD journey. In particular, I thank Dr. Richard Pichler, Dr. Dominic Ma and Dr. Harshal Akolekar for their support during my initial phase of PhD. Many thanks to my colleagues and friends in the fluids group and their contribution to my thesis in one way or another.

Finally, I am grateful to my family for helping me in all possible ways in my life. Without the blessings and invaluable support of my parents, Mohan Lal Saini and Ram Lubhai Saini, my sister Monika Saini and my brother Narotam Saini nothing would have been possible.

Contents

Abstract	i
Declaration of Authorship	iii
Preface	iv
Acknowledgements	v
List of Figures	ix
List of Tables	xvi
Abbreviations	xvii
Dimensionless Numbers	xviii
Symbols	xix
1 Introduction	1
1.1 Background	2
1.2 Performance of a gas turbine	3
1.3 Internal air cooling system	4
1.4 Organization of the thesis	6
2 Literature review	8
2.1 Buoyancy-Induced Flow	8
2.2 Closed rotating cavity	10
2.2.1 Effects of compressibility	14
2.2.2 Effects of the Coriolis forces	15
2.3 Rotating cavity with an axial throughflow	16
2.4 Research Questions	21
3 Governing equations and Numerical techniques	22
3.1 Navier-Stokes Equations	22
3.2 Numerical method	25

3.2.1	Multi-dimensional stability analysis	25
3.2.2	Large-Eddy Simulations	26
4	Centrifugal buoyancy-induced flow in a closed cavity	28
4.1	Numerical Setup	28
4.2	Results and Discussion	31
4.2.1	Grid quality	31
4.2.2	Instantaneous Flow Fields	34
4.2.3	Time averaged statistics	38
4.3	Effect of azimuthal domain size	44
4.4	Conclusions	46
5	Effects of compressibility in centrifugal buoyancy-induced flow	48
5.1	Introduction	48
5.2	Direct Numerical Simulation Results	49
5.3	Linear stability analysis	54
5.3.1	Multi-dimensional stability analysis	56
5.3.2	Results and Discussion	57
5.3.3	Effect of compressibility	60
5.4	Conclusions	64
6	Effects of Rossby number in centrifugal buoyancy-induced flow	66
6.1	Problem Setup	66
6.1.1	Definition of Nusselt number	67
6.2	Results and Discussions	69
6.2.1	Grid quality	69
6.3	Effect of Rossby number on the flow structure and the heat transfer	71
6.4	Effect of viscosity variation with temperature on the heat transfer rate	77
6.5	Scaling of Nusselt number with Rayleigh number	81
6.5.1	Comparison of Nu – Ra scaling with other studies	83
6.6	The role of plumes in the heat transfer	85
6.6.1	Conditional averaging	86
6.7	Conclusions	93
7	Buoyancy-induced flow in a cavity with axial-bore flow	95
7.1	Problem Description and Numerical Setup	95
7.2	Results and Discussion	98
7.2.1	Grid quality	100
7.3	Flow Structure	103
7.3.1	Instantaneous Flow Field	103
7.3.2	Mean Statistics	107
7.4	Heat Transfer	116
7.5	Conclusions	120

8	Conclusions and Scope for future work	122
8.1	Scope for future work	125
A	Derivation for stability criteria	127
B	Cavity with axial-bore flow- Temperature distribution on side disks	129
C	Grid generation - Cavity with axial-bore flow	131
D	List of all the DNS cases for a closed cavity	133
E	Azimuthal Fourier Transforms	134
F	Variation of Nusselt number with time for a closed cavity	136
	Bibliography	138

List of Figures

1.1	(a) Schematic of global atmospheric circulation patterns [7], (b) a model of heat-sink-impeller to enhance the heat transfer rate due to the boundary layer thinning effect of centrifugal forces [6].	2
1.2	Schematic Representation of gas turbine engine	4
1.3	Schematic of internal air system for H.P compressor from Owen et al. [13]	6
2.1	The Rayleigh–Bénard convection cells formed in the system with uniform heating of bottom plate from [17].	9
2.2	Schematic of closed rotating cavity	9
2.3	Schematic of the internal air cooling path of the high pressure compressor of a gas turbine engine showing the inter-disc cavities [4]. . .	17
2.4	Schematic of flow strucutre proposed by Farthing et al.[42] from [4] (Left figure). Schematic of assumed flow strucutre by [43] (Right figure)	17
4.1	Schematic of closed annular cavity. The walls of the cavity undergo solid-body rotation about the z -axis.	29
4.2	Grid-spacing in wall units for the top hot wall at different Rayleigh numbers	33
4.3	Grid-spacing in wall units for the side insulated wall at different Rayleigh numbers, where $r^* = [r - (r_i/L_\infty)]/[(r_o - r_i)/L_\infty]$	33
4.4	(a) Wavenumber spectra of numerical probe located at the middle of the cavity at different Rayleigh numbers and instantaneous contours of azimuthal vorticity for (b) $Ra = 10^6$, (c) $Ra = 10^7$ and (d) $Ra = 10^8$	34
4.5	Instantaneous radial velocity contours at mid-axial position $z = (1/2)(H/L_\infty)$ of the cavity rotating in the clockwise direction for (a) $Ra = 10^5$, (b) $Ra = 10^6$, (c) $Ra = 10^7$, and $Ra = 10^8$	35
4.6	Instantaneous azimuthal velocity contours in the frame of reference of rotation at mid-axial position $z = (1/2)(H/L_\infty)$ of the cavity rotating in the clockwise direction for (a) $Ra = 10^5$, (b) $Ra = 10^6$, (c) $Ra = 10^7$, and $Ra = 10^8$	35
4.7	Instantaneous temperature contours at mid-axial position $z = (1/2)(H/L_\infty)$ of the cavity rotating in the clockwise direction for (a) $Ra = 10^5$, (b) $Ra = 10^6$, (c) $Ra = 10^7$, and $Ra = 10^8$	35

4.8	Time evolution of temperature at the mid-axial position $z = (1/2)(H/L_\infty)$ of the cavity rotating in the clockwise direction for $Ra = 10^5$ and $Ro = 0.25$ at non-dimensional time units (a-h) $t = 22.6, 26.4, 28.3, 30.2, 45.2, 67.9, 113.1$ and 448.6	37
4.9	Time variation of Nusselt number at the outer wall $r = r_o/L_\infty$ of the cavity for $Ra = 10^5$ and $Ro = 0.25$. Refer to Appendix F for more details.	38
4.10	Comparison of average temperature profiles at different Rayleigh numbers. The temperature profiles are averaged over $(1/4)(H/L_\infty) \leq z/L_\infty \leq (3/4)(H/L_\infty)$ to neglect the effect of insulated walls	39
4.11	Comparison of Nusselt number Scaling with Rayleigh number with the scaling reported by Bohn et al.[23], Pitz et al.[27] and Gao et al.[29]. It is important to highlight that the range of Ra numbers considered in the present study as well as in other numerical studies [27] [29] are in the lower end of Bohn et al.[23] experimental conditions and data is sparse in that lower range with some deviations for the $Nu \propto Ra^{0.211}$ scaling law.	40
4.12	Comparison of average temperature profiles with the results reported by Pitz et al.[65]. The temperature profiles are averaged over $(1/4)(H/L_\infty) \leq z/L_\infty \leq (3/4)(H/L_\infty)$ to be consistent with [65].	41
4.13	Instantaneous density contours at mid-axial position $z = (1/2)(H/r_o)$ of the cavity for (a) $Ra = 10^6$ obtained compressible code results, (b) $Ra = 10^6$ from linear approximation, (c) $Ra = 10^7$ obtained compressible code results and (d) $Ra = 10^7$ from linear approximation.	43
4.14	Comparison of instantaneous temperature contours at mid-axial position $z = (1/2)(H/r_o)$ of the cavity at $Ra = 10^6$ for different azimuthal domain size of (a) $2\pi/12$, (b) $2\pi/6$, (c) $2\pi/5$ with the reference domain size of 2π	45
4.15	Comparison of average temperature profiles for $Ra = 10^6$ (—) and $Ra = 10^8$ (----) for different azimuthal domain sizes of 2π (red), $2\pi/5$ (black), $2\pi/6$ (green) and $2\pi/12$ (blue).	45
5.1	Comparison of average (a) radial pressure gradient and (b) density profiles at different Mach numbers for $Ra = 10^5$ and $Ro = 0.25$. The profiles are averaged over $(1/4)(H/L_\infty) \leq z/L_\infty \leq (3/4)(H/L_\infty)$ to neglect the effect of insulated walls.	49
5.2	The comparison of average radial pressure gradient for $Ra = 10^5$ at Ma number of 0.2 and 0.05 with the profiles for the same prediction by equation 5.1.	51
5.3	Instantaneous density contours at mid-axial position $z = (1/2)(H/L_\infty)$ of the cavity for $Ra = 10^5$, $Ro = 0.25$ and (a) $Ma_\infty = 0.05$, (b) $Ma_\infty = 0.1$, (c) $Ma_\infty = 0.15$, (d) $Ma_\infty = 0.2$	52

5.4	Instantaneous temperature contours at mid-axial position $z = (1/2)(H/L_\infty)$ of the cavity for $Ra = 10^5$, $Ro = 0.25$ and (a) $Ma = 0.05$, (b) $Ma = 0.1$, (c) $Ma = 0.15$, (d) $Ma = 0.2$.	52
5.5	Comparison of average temperature profiles at different Mach numbers for $Ra = 10^5$ and $Ro = 0.25$. The temperature profiles are averaged over $(1/4)(H/L_\infty) \leq z/L_\infty \leq (3/4)(H/L_\infty)$ to neglect the effect of insulated walls.	53
5.6	The variation of Nusselt number with Mach number for $Ra = 10^5$ and comparisons for $Nu-Ra$ scaling for other Rayleigh numbers with the scaling reported by Bohn et al.[23], Pitz et al.[27].	54
5.7	Schematic of closed annular cavity. The walls of the cavity undergo solid-body rotation about the z -axis.	55
5.8	Time evolution of density at the mid-axial position $z = (1/2)(H/L_\infty)$ of the cavity for $Ra = 10^5$, $Ro = 0.199$ and $Ma = 0.01$ at non-dimensional time units (a-e) $t = 10.0, 22.0, 25.0, 40.0$ and 55.0 , respectively, and for $Ra = 10^7$, $Ro = 0.199$ and $Ma = 0.05$ at non-dimensional time units (f-j) $t = 12.5, 18.0, 25.0, 32.5$ and 38.0 , respectively. (k) Wavenumber spectra of the monitor point located at the middle of the cavity at different Rayleigh numbers. All results from DNS.	58
5.9	(a)Temporal developement of first 16 most amplified Fourier modes in the LSA at $(z, r) = (0.527, 1.2)$ for $Ra = 10^5$ at $Ma = 0.01$ and $Ro = 0.199$. Instantaneous contour for radial velocity fluctuations at (b) $t = 5$ (contour levels $[-1.34e-05, +1.34e-05]$), (c) $t = 10$ (contour levels $[-2.34e-05, +2.34e-05]$) and (d) $t = 15$ (contour levels $[-9.34e-05, +9.34e-05]$).	59
5.10	Temporal developement of first 24 most amplified Fourier modes in the LSA at $(z, r) = (0.527, 1.2)$ for $Ra = 10^7$ at $Ma = 0.05$ and $Ro = 0.199$.	60
5.11	Time evolution of dominant mode in the azimuthal direction at $(z, r) = (0.527, 1.2)$ for — $Ra = 10^5$, --- $Ra = 10^7$ for $Ro = 0.199$ at different Ma numbers.	61
5.12	Comparison of growth rate of most unstable mode for different Ra numbers at $Ro = 0.199$.	62
5.13	A schematic to illustrate the stability condition given in equation 5.3, assuming the fluid parcel follows the adiabatic temperature profile. The temperature profiles shown here for different possible scenarios are just the representation to explain the condition given in equation 5.3. The actual profiles might have different slopes.	63
5.14	Stability curve as given in equation 5.4 and showing the conditions at different Ro numbers for $Ra = 10^7$. The black curve is the stability criterion defined by equation 5.3	64
6.1	Schematic of closed cavity rotating about the z -axis.	67
6.2	Grid-spacing in wall units for the top hot wall at $Ra = 10^7, 10^8$ and 10^9 for $Ro = 0.75$.	70

6.3	Grid-spacing in wall units for the side insulated wall at $Ra = 10^7$, 10^8 and 10^9 for $Ro = 0.75$, where $r^* = [r(r_i/L_\infty)]/[(r_o r_i)/L_\infty]$	70
6.4	Instantaneous temperature and radial velocity contours at the mid-axial position for $Ra = 10^8$ at (a,d) $Ro = 0.25$, (b,e) $Ro = 0.5$ (c,f) $Ro = 0.75$, respectively, and ; for $Ra = 10^9$ at (g,j) $Ro = 0.25$, (h,k) $Ro = 0.5$ (i,l) $Ro = 0.75$, respectively. The contour levels for the temperature and the radial velocity are $[-0.5, 0.5]$ and $[-0.25, 0.25]$, respectively.	72
6.5	Instantaneous temperature and radial velocity contours in r - z plane at a fixed circumferential location for $Ra = 10^8$ at (a,d) $Ro = 0.25$, (b,e) $Ro = 0.5$ (c,f) $Ro = 0.75$, respectively.	74
6.6	(a) Instantaneous radial velocity contours in the z - θ plane at radial location $(r - r_i)/(r_o - r_i) = 0.966$ for Ro (a) 0.25, (b) 0.5 and (c) 0.75 at contour levels $[-0.05, 0.18]$	75
6.7	(a) The variation of quantity given in equation 6.10 with axial coordinate for $Ra = 10^9$, and (b) the variation of integrated values over axial direction of the same term with Ro number for different Ra numbers.	76
6.8	The variation of Nusselt number at the hot wall with Rossby number and the data from other studies by Saini and Sandberg [58] at $Ro = 0.23$ and $Ra = 10^8$ and by Gao et al. [29] at $Ro = 0.202$ and $Ra = 10^9$	77
6.9	The comparison of mean temperature profiles at low Ra numbers for different Ro numbers.	78
6.10	The comparison of mean temperature profiles at high Ra numbers for different Ro numbers.	78
6.11	A schematic of mean temperature profile highlighting the assymetricity and nomenclature in the analysis.	79
6.12	The comparison of (a) χ_θ , which represents the core temperature rise and (b) the ratio of thermal boundary layer thickness for different Ra numbers. Both these quantities estimate the contribution of NOB effects towards the asymmetry of the temperature profiles.	80
6.13	The comparison of mean temperature profiles for different Ro numbers.	82
6.14	Comparison of Nu number scaling with Ra number for different Ro number.	82
6.15	The variation of Nusselt number at the hot wall with Rossby number with the data from other studies by Saini and Sandberg [58] for $Ra = 10^8$ at $Ro = 0.23$, Gao et al. [29] for $Ra = 10^9$ at $Ro = 0.202$ and Bohn et al. [23] for $Ra = 10^7$, 10^8 and 10^9 at $Ro = 0.109$, 0.084 and 0.065 , respectively.	84
6.16	Comparison of Nu number scaling with Ra number with results reported by Bohn et al.[23], Pitz et al.[27], Gao et al.[29], Saini and Sandberg [58] and Jiang et al.[41].	85
6.17	Instantaneous temperature contour of $Ra = 10^9$ for (a) $Ro = 0.25$ and (b) $Ro = 0.75$. The hot thermal plumes detected from the instantantaneous data shown in (a) and (b)	87

6.18	Conditionally-averaged contours of temperature for $Ra = 10^8$ at (a) $Ro = 0.25$ (contour levels $[1.0, 1.1575]$), (b) $Ro = 0.5$ (contour levels $[1.0, 1.630]$) and (c) $Ro = 0.75$ (contour levels $[1.0, 2.4178]$) and for $Ra = 10^9$ at (d) $Ro = 0.25$ (contour levels $[1.0, 1.1575]$), (e) $Ro = 0.5$ (contour levels $[1.0, 1.630]$) and (f) $Ro = 0.75$ (contour levels $[1.0, 2.4178]$).	87
6.19	(a) A schematic of different regions identified in the flow which include Region I, where hot plume rises from the outer wall, Region II, where cold plume rises from the inner wall and Region II for the wind of the convection, (b) instantaneous contour of temperature in the Region I in the $r - z$ plane shows the near wall hot plumes rising and (c) shows the instantaneous contour of temperature in the Region III in the $r - z$ plane shows the large scale circulation or convection wind.	88
6.20	The averaged plumes temperature for $Ra = 10^9$ at (a) $Ro = 0.5$ and (b) $Ro = 0.75$. The comparison of plume-dominated regions at the edge of thermal boundary layer thickness predicted by the plume temperature and Ri number values.	89
6.21	The comparison of time averaged profiles for $[\langle T \rangle_t - T_m]/[\Delta T]$ as a function of wall normal distance r/λ_{th} for different Ra number at (a) $Ro = 0.75$, (b) $Ro = 0.5$ and (c) $Ro = 0.25$. Solid circles show the DNS temperature profile from Ahlers et al. [80] at $Ra = 2 \times 10^{12}$	90
6.22	The variation of root-mean-square of temperature fluctuations as a function of wall normal distance r/λ_{th} for different Ra number at (a) $Ro = 0.75$, (b) $Ro = 0.5$ and (c) $Ro = 0.25$	92
6.23	Comparison of Nu number scaling with Ra number for plume-ejection region with the full 2π averaged values	93
7.1	Schematic of inter-disk cavity. The walls of the cavity undergo solid-body rotation about the z -axis except the shaft, which is held stationary to match the conditions of [49].	96
7.2	(a) Temperature profiles for the side disks from [49], Comparison of fully developed (b) mean axial velocity profile and (c) turbulent kinetic energy (TKE) in the bore region obtained by digital filter inflow at the downstream location $z/r_o = -0.0625$ from the inlet with the profiles from precursor periodic simulation.	100
7.3	Comparison of sub-grid scale viscosity with molecular viscosity for (a) $Ra = 0.76 \times 10^8$ and (b) $Ra = 1.54 \times 10^8$	101
7.4	Grid-spacing in wall units for the shroud (a – b), upstream disk (d – f) and downstream disk (g – i) at different Rayleigh numbers.	102
7.5	Wavenumber spectra of different monitor points located in the cavity at different Rayleigh numbers (— $Ra = 0.76 \times 10^8$, --- $Ra = 1.54 \times 10^8$).	102
7.6	Instantaneous axial velocity contour for the cavity showing the turbulent bore flow interaction with the cavity.	104

7.7	Instantaneous contour for circumferential vorticity in the cavity for $Ra = 1.54 \times 10^8$	104
7.8	Comparison of instantaneous contours of temperature at (a) $Ra = 0.76 \times 10^8$ and (b) $Ra = 1.54 \times 10^8$, radial velocity scaled with local disk speed at (c) $Ra = 0.76 \times 10^8$ and (d) $Ra = 1.54 \times 10^8$ and azimuthal velocity scaled with local disk speed at (e) $Ra = 0.76 \times 10^8$ and (f) $Ra = 1.54 \times 10^8$ at mid-axial position of the cavity rotating in the clockwise direction.	105
7.9	Comparison of time and circumferentially averaged velocity profiles at the mid-axial position of the cavity at different Rayleigh numbers.	107
7.10	Comparison of time and circumferentially averaged velocity profiles at the different radial locations of the cavity in Region II (a-c), Region III (d-f) and Region IV (g-i) for different Rayleigh numbers (— $Ra = 0.76 \times 10^8$, --- $Ra = 1.54 \times 10^8$).	108
7.11	(a) Ekman spiral formed at different radial locations on upstream disk and comparison with the solution of linear Ekman layer equations, (b) Comparison of boundary layer thickness for different Ra numbers on both the disks scaled with disk gap H	112
7.12	(a) Schematic of flow structure for the heated shroud and side disks with radially increasing temperature profile (I) interpretation from the time and circumferentially averaged statistics, (II) instantaneous temperature contours near the shroud shows the local convection cells (III) visualisation of toroidal vortex by iso-surfaces of λ -ci colored by local temperature, (b) Flow structure in the radial-azimuthal plane interpreted from the instantaneous flow field, where C refers to cyclonic and A to anti-cyclonic vortices.	114
7.13	Schematic of flow structure proposed by Farthing et al.[42] from [4] (Left figure). Schematic of assumed flow structure by [43] (Right figure)	115
7.14	Comparison of time and circumferentially averaged density $\langle \bar{\rho} \rangle_\theta$ and temperature $\langle \bar{T} \rangle_\theta$ profiles at the different radial locations of the cavity in Region II (a,b), Region III (c,d) and Region IV (e,f) for different Rayleigh numbers (— $Ra = 0.76 \times 10^8$, --- $Ra = 1.54 \times 10^8$).	116
7.15	Comparison of time averaged Nusselt number ($Nu_r^\infty = [r/(T_w - T_\infty)][-d\bar{T}/dz]_w$) calculated based on the shaft temperature (T_∞) for $Ra = 0.76 \times 10^8$ at (a) upstream disk, (b) downstream disk and for $Ra = 1.54 \times 10^8$ at (c) upstream disk, (d) downstream disk.	118
7.16	Comparison of the variation of local Nusselt number defined based on the shaft temperature (T_∞) (— $Nu_r^\infty = [r/(T_w - T_\infty)][-d\bar{T}/dz]_w$) and based on the core cavity temperature (--- $Nu_r^c = [r/(T_w - T_c)][-d\bar{T}/dz]_w$) on the side disks for different Rayleigh numbers.	118

7.17	Comparison of shroud Nusselt number with the results of closed cavity without bore flow from [57]. It is important to highlight that the radius ratio (r_i/r_o) for the closed cavity is 0.52 as compared to the cavity with axial bore flow for which the radius ratio is 0.318. Thus in this comparison, the effect of this difference in the radius ratio is neglected.	119
C.1	Grid for inter-disk cavity.	132
F.1	Time variation of Nusselt number at the outer wall $r = r_o/L_\infty$ of the cavity for $Ra = 10^5$ and 10^7 . The initial time interval for $Ra = 10^7$ is shown here as the full dataset was not available. However, it is important to highlight that the flow statistics for $Ra = 10^7$ were calculated over the time interval from 383 to 540 (which is not shown here).	136
F.2	Time variation of Nusselt number at the outer wall $r = r_o/L_\infty$ of the cavity for $Ra = 10^5$ and $Ro = 0.25$	137
F.3	Instantaneous temperature contours at mid-axial position $z = (1/2)(H/L_\infty)$ of the cavity at the corresponding minimum and maximum values of Nu number observed at the outer wall.	137

List of Tables

4.1	DNS cases: N_r , N_z and N_θ are number of grid points in respective directions.	31
4.2	Grid sensitivity for $Ra = 10^7$	32
4.3	The comparison of grid resolution in the thermal boundary layer against the guidelines provided by Shishkina et al. [61]	32
4.4	DNS cases of different sector sizes for $Ra = 10^6$ and 10^8	44
6.1	DNS cases to study the effect of Rossby number.	69
6.2	The comparison of grid resolution in the thermal boundary layer against the guidelines provided by Shishkina et al. [61]	71
6.3	The comparison of grid resolution in the thermal boundary layer against the guidelines provided by Shishkina et al. [61]	71
7.1	Simulation Parameters	99
B.1	Non-dimensional side disk temperature with respect to T_∞ for case I and II	129
D.1	DNS conditions replicating the experimental conditions of [23] from chapter 4.	133
D.2	DNS cases from chapter 6 to study the effect of Rossby number. . .	133

Abbreviations

CFL	Courant– F riedrich–Lewy condition
DNS	Direct N umerical S imulations
HPC	H igh P ressure C ompressor
LES	Large E ddy S imulations
LSA	Linear S tability A nalysis
RANS	Reynolds A veraged N avier– S tokes
TKE	Turbulent K inetic E nergy
URANS	Unsteady R eynolds A veraged N avier– S tokes
WALE	Wall– A dapting L ocal E ddy

Dimensionless Numbers

Ma	Mach number
Nu	Nusselt number
Pr	Prandtl number
Ra	Rayleigh number
Re	Reynolds number
Ri	Richardson number
Ro	Rossby number

Symbols

H	axial gap between the disks
z	axial direction or axial length
U_{bulk}	bulk velocity of axial bore flow
θ	circumferential direction
T_{core}	core temperature
F_{cor}	Coriolis force
ρ	density
μ	dynamic viscosity
δ	Ekman depth
Δr^+	grid-spacing in wall-units in the radial direction
Δz^+	grid-spacing in wall-units in the axial direction
$r\Delta\theta^+$	grid-spacing in wall-units in the azimuthal direction
q	heat flux
ν	kinematic viscosity
N	number of grid points
t	non-dimensional time
Ω	non-dimensional angular velocity
p	pressure
γ	ratio of specific heats
r	radial direction or radius
T_{rms}	root-mean-square of temperature fluctuations
ρ_∞	reference density
Ω_∞	reference angular velocity

T_∞	reference temperature
U_∞	reference velocity
L_∞	reference length scale
μ_∞	reference viscosity
κ_∞	reference thermal conductivity
S	Sutherland temperature
μ_t	sub-grid scale viscosity
c_p	specific heat at constant pressure
τ	shear stress
λ^θ	thermal boundary layer tickness
β	thermal expansion coefficient
α	thermal diffusivity
κ	thermal conductivity
E	total energy
T	temperature
u, v, w	velocity in z , r and θ direction

Chapter 1

Introduction

Rotating flows and the induced convection by rotation is a ubiquitous phenomenon that arises in many natural and engineering systems. In geophysical flows, it controls numerous observed features of the planets such as geomagnetism [1] and thermal emissions [2] and is also of interest in the atmospheric sciences to study the global circulations [3]. The rotating convection systems are also cardinal across a wide range of engineering applications such as gas turbines [4], pumps and other turbomachinery systems or sub-systems [5]. An example of rotating flows, shown in figure 1.1 (a), highlights the scientific significance of earth's rotation to control the atmospheric circulations, which influences the weather cycle. The atmospheric circulation pattern has many dynamic features; one of them is the deflection of low-level wind from the poles to the equator. The reason for this deflection is the Coriolis forces induced due to the earth's rotation. This deflection of the wind leads to the development of three different circulations or convection cells, as shown in figure 1.1 (a). The second example in figure 1.1 (b) illustrates the effect of centrifugal forces on the heat transfer enhancement in a heat sink [6] for electronic cooling. The reason behind the high efficiency of this heat sink design compared to a conventional design is the centrifugal forces due to the rotation of the heat sink itself. The centrifugal forces cause a boundary layer thinning effect around the impeller fins, and it enhances the wall heat transfer rate. Thus, it is of interest to engineers as well as to scientists to understand the fundamentals of

rotating flows and convection induced in rotating systems to develop new theories, accurate models and to improve their engineering designs further. This thesis work is primarily motivated by an engineering application to a gas turbine, as outlined next.

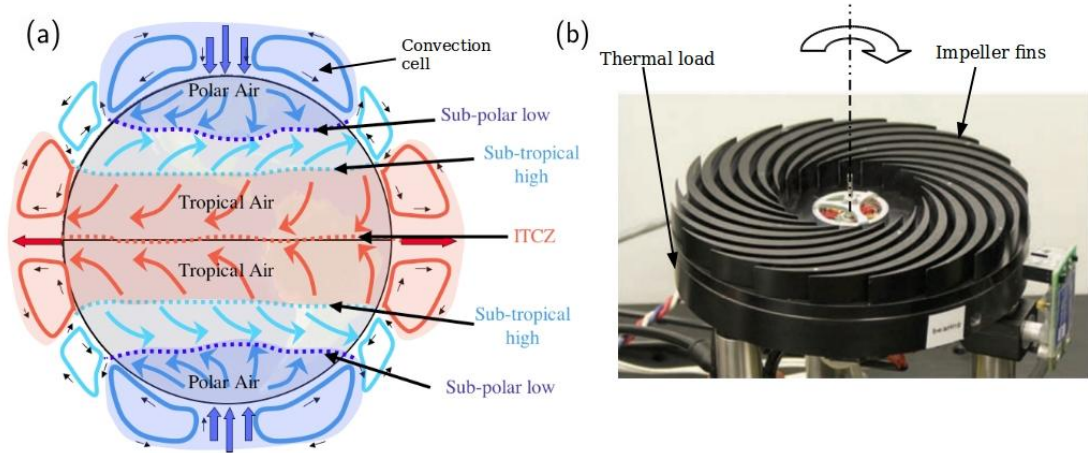


FIGURE 1.1: (a) Schematic of global atmospheric circulation patterns [7], (b) a model of heat-sink-impeller to enhance the heat transfer rate due to the boundary layer thinning effect of centrifugal forces [6].

1.1 Background

The ever increasing demand for energy around the world makes it imperative to develop more efficient technologies for the future. According to International Energy Agency[8], the overriding need to transform the energy sector is to limit the rise in the global mean temperature to well below 2°C. It can be achieved by designing energy-efficient buildings, improving industrial processes and transforming the transportation energy needs, which could reduce the world's projected energy requirements in 2050 by one third and reduce the global emission of greenhouse gases. Within the transportation industry, the energy demands for aviation has grown by 1.4% per year since 2000 and is projected to increase dramatically in the near future due to its skyrocketing growth. A slight improvement in the performance of a gas turbine engine would lead to significant savings in terms of fuel

consumption and its detrimental effects on our climate. Thus, it is unequivocally important for engineers to seek further design improvements of future gas turbines.

1.2 Performance of a gas turbine

A gas turbine is an internal combustion open-cycle engine based on the Brayton cycle [9]. A schematic representation of a gas turbine engine is shown in figure 1.2. The air enters the engine via a fan, and more than 90% of the air passes through the bypass duct to generate the majority of engine thrust. The rest of the air enters the core of the engine and is then compressed to a higher pressure at different compressor stages (low pressure and high-pressure compressor, as shown in figure 1.2). After the compressor, pressurised air enters the combustion chamber where combustion takes place and further increases the temperature of the compressed air. This hot pressurised air then expands into the turbine to produce work required to power the compressors, fan and other auxiliary devices of an engine. After releasing its energy, the air exits through the exhaust system and generates thrust.

The evolution of gas turbine technology for future commercial aviation is mainly focused on improving the thermodynamic efficiency, reducing weight and enhancing the safety of an engine. The thermodynamic efficiency based on the ideal cycle is defined as

$$\eta = 1 - \frac{T_1}{T_2} = 1 - \left(\frac{P_1}{P_2} \right)^{\frac{(\gamma-1)}{\gamma}}, \quad (1.1)$$

where T_1 , P_1 and T_2 , P_2 represent the temperature and pressure at the compressor inlet and outlet, respectively. The thermodynamic efficiency has been improved from 10% to 50% since the invention of the first working engine [10], with the theoretical limit of the idealized gas turbine cycle estimated to be 80%. Thus based on equation 1.1, the thermodynamic efficiency of a gas turbine can be further improved by increasing the overall pressure ratio. However, such change in the operating conditions leads to further design challenges for the high-pressure compressor. First, the exit temperature of the compressor rises with an increase

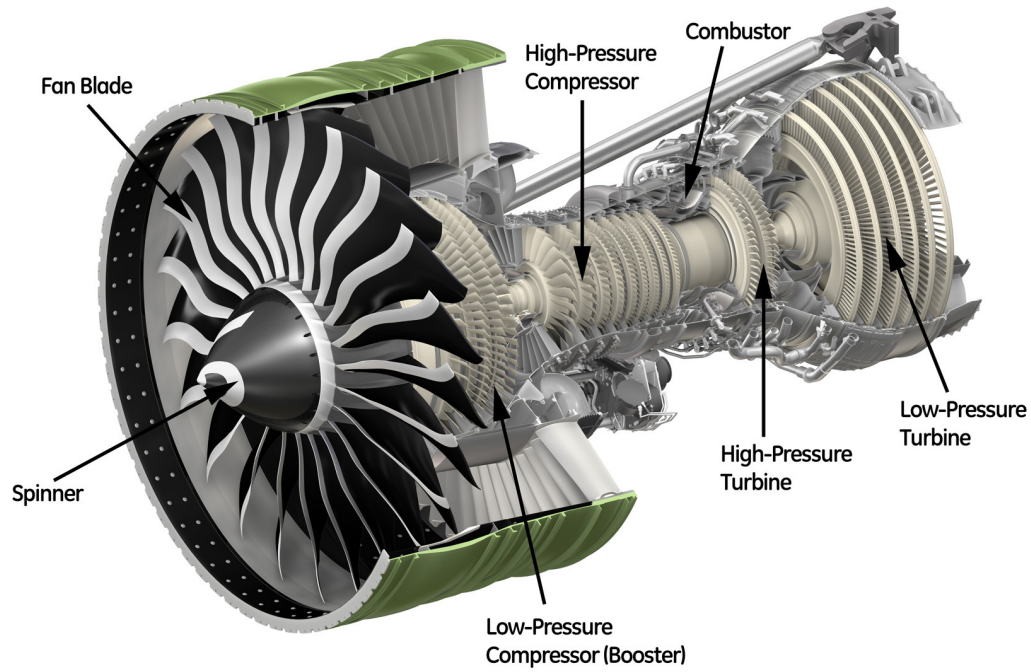


FIGURE 1.2: Schematic Representation of gas turbine engine

in pressure ratio and causes local heating of the compressor blades beyond their material withstanding limits. Second, it increases the average temperature of air bled from the respective stage of the compressor, which is intended to be used for the cooling of turbine blades. Third, with a rise in the pressure ratio, the clearance between the last stage of compressor blades and the casing reduces, and it requires precise prediction of thermal expansion of the compressor disks internal air cooling system to avoid any damage to the blades. Thus, to circumvent above-mentioned design challenges which arise when increasing the overall pressure ratio, it is essential to design an effective and efficient internal air cooling system of a gas turbine engine.

1.3 Internal air cooling system

In a modern gas turbine engine, a secondary internal air path, also known as internal air cooling system, is responsible for removing excess heat from the various

components, which can be detrimental for their safe operation. The focus of this study is on the internal air cooling section of the high-pressure compressor, as shown in figure 1.3. The cavities formed between the rotating disks of the compressor and some of them are completely closed (mainly in industrial gas turbines) or fed by axial throughflow of cold air [11]. The fluid flow and heat transfer inside these annular cavities directly influences the cooling of the compressor blades and the blade-tip clearance. The accurate prediction of the clearance between the blade-tip and the casing is not only crucial for operational efficiency, but it also affects the stage efficiency. For every 1% change in tip clearance (expressed in percentage of mean blade height) the stage efficiency is affected by 1.5% [12]. Thus, for the designer, it is necessary to predict the steady as well as transient temperature distribution on the compressor disk to calculate their thermal expansion. This temperature distribution depends upon the buoyancy-induced flow and induced wall heat-transfer rate inside these cavities. Moreover, it is essential to minimize the mass flow rate of axial throughflow, as the cold air bled from the primary engine cycle at different stages of compression is parasitic to the engine performance as it does not contribute to generating any thrust. In the present-day gas turbine engines, up to one-fifth of the total core mass flow is used by the secondary air system for numerous functions and thus it directly affects the specific fuel consumption of an engine. Therefore, it is crucial to predict the flow physics and the induced wall heat-transfer rate accurately inside these cavities to optimize the overall design and enhance the performance and the safety of an engine.

The centrifugal buoyancy-induced flow inside these cavities is highly unsteady, three dimensional, unstable and encompasses many dynamic flow features and has therefore also been termed as the “Everest” of rotating flow problems in the initial experimental studies of Owen et al.[14] and Farthing et al.[15]. Due to the complexity of the problem, only the measurement of the resulting wall heat-transfer rate was the focus of most of the experimental studies, which will be reviewed in chapter 2.

High fidelity simulations, on the other hand, provide the opportunity to understand the detailed flow physics and consequently propose newer theories or models.

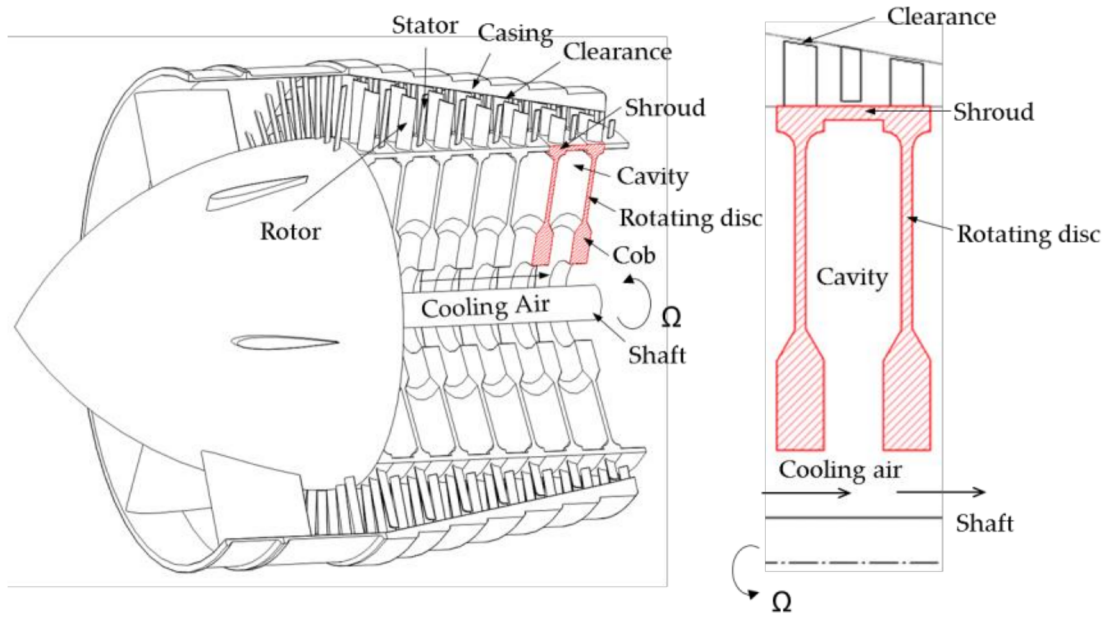


FIGURE 1.3: Schematic of internal air system for H.P compressor from Owen et al. [13]

In the first part of this study, we will investigate the closed cavity configuration by carrying out direct numerical simulations (DNS) for systematic variation of a set of parameters. It will provide deeper insights into the influence of different governing parameters on the flow physics and the induced wall heat-transfer rate. In the second part, we will characterize the flow structure and heat transfer mechanisms for the cavity with an axial bore flow by conducting highly resolved large-eddy simulations (LES).

1.4 Organization of the thesis

The thesis is organized into the following chapters with a brief description of the contents of each chapter as follows.

Chapter 2 reviews the literature on centrifugal buoyancy-induced flows covering experimental, numerical and theoretical studies. The research gaps in the literature are identified to define the research questions of this study.

Chapter 3 presents the governing equations and numerical techniques used in this study.

Chapter 4 covers the problem setup for the canonical closed cavity configuration. The results of DNSs are discussed with a focus on comparison with the experiments and with other numerical studies to highlight the differences.

Chapter 5 presents the results of the effects of compressibility in the closed cavity problem. The discussions are centered initially on the DNS results, followed by linear-stability analysis to provide further insights.

Chapter 6 discusses the results of the effect of Rossby number on flow dynamics and the wall heat transfer rate. It also presents the results on the conditional averaging of the data to understand the roles of plumes in the heat-transfer enhancement.

Chapter 7 covers the LES results of the cavity with an axial throughflow. It presents the mean flow quantities to explain the flow structure, boundary layer characteristics formed on the side disks and heat-transfer results.

Chapter 8 summarizes the conclusions from the present study. It also includes an outline of future work.

Chapter 2

Literature review

2.1 Buoyancy-Induced Flow

Buoyancy-induced flows are all around us, from the atmospheric flows to the boiling of water to a multitude of industrial applications. For example, in our atmosphere, the solar radiation heats the fluid parcels near the surface, and those lighter fluid parcels rise and leave behind low-pressure regions. The cold air from the surrounding occupies that low-pressure region, forming the atmospheric circulation which leads to the jet currents. Effectively, an external physical process that causes a density difference in the flow system leads to a buoyancy-induced flow in the presence of body forces such as gravity or centrifugal forces. An idealized case for the demonstration of this type of flow is the Rayleigh–Bénard convection, where convection currents in the flow develop between two walls at different temperatures in the presence of gravity. The seminal work on this problem was carried out by Bénard [16], where a pattern of cells (Bénard cells) was observed as shown in figure 2.1. Subsequently, Rayleigh [18] extended the mathematical analysis of the same problem and reproduced the results of Bénard [16]. Many other fundamental aspects of this problem were explored thereafter to understand the onset of convection instability [19], pattern formation [20], heat transfer [21] and transition to the ultimate regime of convection [22].

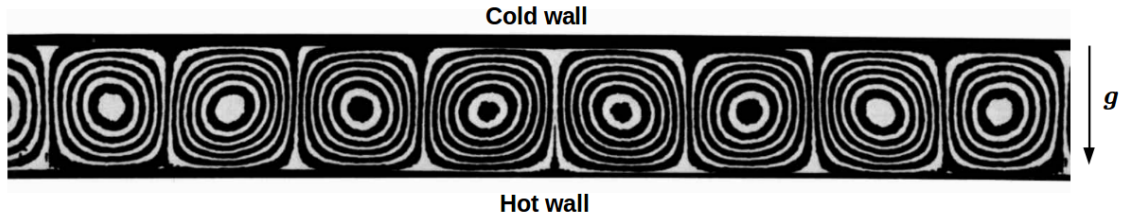


FIGURE 2.1: The Rayleigh–Bénard convection cells formed in the system with uniform heating of bottom plate from [17].

The internal air cooling system in a gas turbine engine also has a configuration similar to the classical Rayleigh–Bénard problem where gravity is replaced by centrifugal forces. Figure 2.2 shows a canonical model of the annular rotating cavity that rotates at angular velocity Ω about z axis. It is bounded between two insulated side disks, and the outer cylindrical surface is maintained at a high temperature and the inner cylindrical surface at a low temperature. This imposed radial temperature difference creates a radial density difference across the cavity. This density difference combined with the centrifugal acceleration due to the rotation induces fluid flow in the cavity relative to the solid-body rotation.

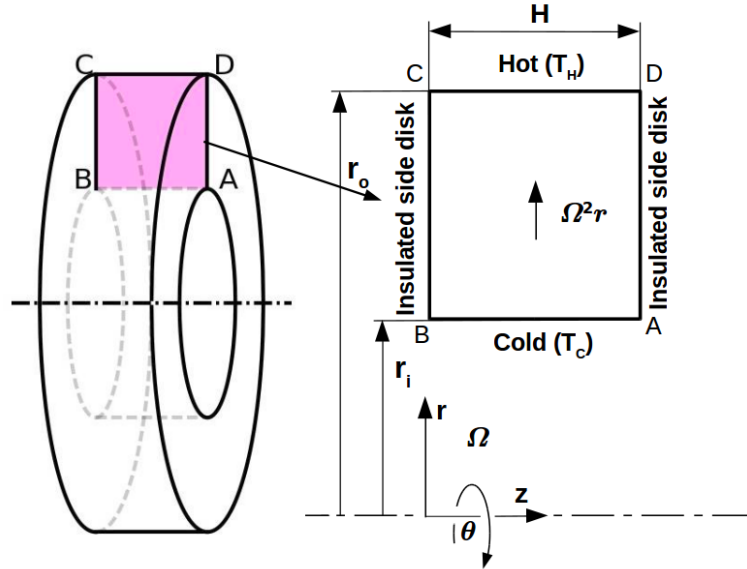


FIGURE 2.2: Schematic of closed rotating cavity

The aim of this chapter is to review the existing literature on centrifugal buoyancy-induced flows with the relevance to the internal air cooling system of a gas turbine. The discussion of literature is divided into two major categories, based on the problem under consideration. The first configuration involves the closed rotating cavity without any axial throughflow, which is a canonical setup shown in figure 2.2. In the second configuration, the literature pertinent to the cavity with an axial throughflow is discussed.

2.2 Closed rotating cavity

The closed cavity problem under consideration has a temperature gradient aligned with the direction of the centrifugal force vector, as shown in figure 2.2. Initial experiments for the configuration of the closed rotating cavity relevant to the internal air cooling system were carried out by Bohn et al. [23]. They tested three different cavity configurations of different dimensions in their experiments, amongst which the second configuration resembles the schematic in figure 2.2 and thus is used for comparison in this study. For a given set of geometric parameters, the authors [23] proposed the wall Nusselt number (Nu) as a function of rotational Reynolds number (Re) and Rayleigh number (Ra) as

$$Nu = f(Ra, Re) \quad (2.1)$$

and provided the correlation for Nusselt number from the experimental measurements as

$$Nu = 0.317Ra^{0.211}, \quad Re = 1.441Ra^{0.557}. \quad (2.2)$$

In equations 2.1 and 2.2, Reynolds and Rayleigh numbers are defined as $Re = (\rho\Omega r_m L)/\mu$ and $Ra = [\rho^2 c_p (\Omega^2 r_m) (\Delta T/T_\infty) L^3]/(\mu\kappa)$, respectively. The Nusselt number (Nu) is defined as the ratio of the heat flux from the outer to the inner wall by convection to the heat transfer by conduction. Due to the temperature limits of their experimental setup, the wall temperature difference varied over only a small range and reduced with the increase in the value of Rayleigh number.

They pressurized the cavity to increase the Rayleigh number at a small temperature difference. Thus, at high Rayleigh number, the thermodynamic state of the fluid was different as compared to the low Rayleigh number cases. Moreover, in the experiments, all the control parameters of the problem, viz. rotational speed, temperature difference and the density of the working fluid were varied to achieve a particular combination of rotational Reynolds number and Rayleigh number. Thus, it is difficult to isolate the effects of all these parameters on the flow and the heat transfer rate. Also, the experiments were designed to measure the heat transfer only, and consequently, no information about the flow structure was reported. However, they extended their study by using CFD to solve the steady-state as well as unsteady equations assuming the flow to be laminar. The comparison of steady calculation results with experiments showed an over-prediction of Nu number values and the authors speculated the reason being the poor insulation of the side walls in their experimental setup. The results of unsteady simulations were inconclusive due to convergence issues and observed large deviations from the experiments. A subsequent numerical study by Bohn and Gier [24] solved three-dimensional Navier-Stokes equations with a $k-\epsilon$ turbulence model. A correction to the experimental data of Bohn et al. [23] was also proposed in this study to the mean values of Nu number to compensate for the heat losses from the side insulated walls given as

$$\begin{aligned} Nu_{corr} &= Nu(Ra_{corr}), \\ Ra_{corr} &= (1/1500)Ra^{1.406}, \end{aligned} \tag{2.3}$$

where Ra_{corr} is the corrected Rayleigh number calculated as a function of original Ra number and Nu_{corr} is the corrected value of Nusselt number at a corrected Rayleigh number. The comparison with experimental data after corrected for heat loss from the side walls, showed again an over-prediction of Nu number values. Thus there was no clear explanation for the reduced $Nu-Ra$ scaling coefficient of 0.211 compared to the scaling coefficient of the Rayleigh-Bénard convection [22] and no depiction of the flow structure inside the cavity.

To understand the flow physics and the reduced scaling coefficient of Nusselt number with Rayleigh number, several numerical studies were carried out to reproduce the experimental results of Bohn et al. [23]. A pioneering study by Sun et al. [25] analyzed this problem by solving compressible Navier–Stokes equations for the same conditions as [23] for the Rayleigh number of the order of 10^{10} . They observed large scale structures in the cavity for all the Rayleigh numbers, which lead to a significant variation of the average heat transfer rate. The Nusselt number values at the outer wall were in good agreement with the experimental results. The authors hypothesized that the Coriolis effects in the core suppressed the turbulence. It was proposed to be the reason behind the reduced heat transfer rate observed in the experiments as well as in their simulations. Furthermore, to reduce the computational cost for high Rayleigh number cases, they simulated a 45° sector of the cavity with periodicity in the azimuthal direction. They found little variations in the mean heat transfer rate and discernible changes in the core of the flow structure. The study thus provided strong evidence about the ability of a compressible solver to reproduce the experimental results of Bohn et al. [23].

However, a widely used approach for simulating buoyant flows is solving the incompressible equations with the Boussinesq approximation. A study by King et al. [26] used the vorticity-streamline function, and velocity-vorticity formulations to analyze this problem in two and three-dimensional spaces. The equations for the formulations, as mentioned earlier, were derived from the Navier–Stokes equations using the Boussinesq approximation. The results of the 2D computations for Rayleigh number of the order of 10^9 showed flow structures analogous to Rayleigh–Bénard convection with enhanced thermal mixing at higher Rayleigh number. The number of vortex pairs in the domain was a function of time and varied between 4 to 6, and this produced unsteadiness in the wall heat transfer rate. Further, the Nusselt number computed at the hot wall was in good agreement with the $1/3$ scaling law, which was significantly higher than the measurements of Bohn et al. [23]. The reason for this disagreement was not clarified by the authors.

Another study that used an incompressible solver with Boussinesq approximation is that of Pitz et al. [27], who conducted direct numerical simulations. Large-scale

mushroom-shaped structures were observed in the domain, identical to those found by King et al. [26]. The prediction of the Nusselt number at the outer wall was not in agreement with the experimental results but rather in consensus with the $1/3$ scaling law. They also tested the extended version of the Boussinesq approximation in their study proposed by Lopez et al. [28] for rotating flows. The Nusselt number scaling for the extended Boussinesq approximation was in close agreement with experimental results [23], but it was reported to be imperfect for this problem as smearing of the flow structures was observed in the azimuthal direction. Clearly, using incompressible Navier-Stokes equations with the Boussinesq approximation is not able to reproduce the $Nu-Ra$ scaling for the experimental conditions of Bohn et al. [23].

A recent study by Gao et al. [29] also highlighted these differences between the compressible and incompressible solvers for the physical conditions of Bohn et al. [23]. They compared the DNS results of an incompressible solver with LES of a compressible solver. The results of the compressible solver were reported to be in close agreement with the corrected experimental scaling Bohn and Gier [24] of Nusselt number. Viscous heating in the case of the compressible solver was claimed to be the reason behind the differences observed between the two solvers.

The discussion thus far shows that, there is no consensus in the literature on the physical reason behind the scaling coefficient of Nusselt number with Rayleigh number observed in the experiments [23]. The main reason behind the disagreement in the findings of different studies is most likely rooted in the fact that the governing parameters in the experiments by Bohn et al. [23] were not varied in a controlled manner. Thus, it is difficult to isolate their effects in the simulations as well. Therefore, there is an unequivocal requirement to formulate this problem in a more systematic way to separate different parameters and understand their effects on the flow physics and heat transfer. The two predominant phenomena which were neglected in the above-discussed studies were the effects of (a) the compressibility and, (b) the Coriolis forces. In the next two sub-sections, studies focussed onto understanding the two effects mentioned above in buoyancy-induced flows, which will serve to define the research questions.

2.2.1 Effects of compressibility

The compressibility effects in rotating flows become significant when rotational speed increases to a value beyond a critical number. In other words, the rotational Mach number's significance necessitates the solution of the compressible Navier–Stokes to represent the flow physics accurately. The initial results on the effect of compressibility with a condition for flow stability was provided by Eckhoff and Storesletten [30] and subsequently by Kilfoil and Chew [31] for a closed rotating system. The effect of high rotational speed on the wall heat transfer rate for an open cavity was reported by Farthing et al. [32] for a cavity with the axial throughflow for engine representative conditions and recently observed by Jackson et al. [33] in their experimental measurements. They observed a decrease in the value of wall Nusselt number with an increase in the value of the rotational Reynolds number. They attributed this to the compressibility effects in the core of the cavity, as it led to an increase in the core temperature and reduced the wall heat transfer rate. Tang and Owen [34] in their theoretical model considered the compressibility effects in the core to predict the wall Nusselt number. They proposed a scaling of the Nusselt number as

$$Nu \propto Ra^{1/4}(1 - \zeta Re^2)^{5/4}, \quad (2.4)$$

where ζ was the compressibility parameter. They concluded that the compressibility effects were indeed responsible for the reduced scaling coefficient observed in the experiments of Bohn et al. [23]. However, they sought more experimental or computational evidence to support their hypothesis about the effects of compressibility effects in centrifugal buoyancy-induced flows. A very recent numerical study by Gao and Chew [35] also highlighted the compressibility effects in the closed cavity configuration of Bohn et al. [23]. The authors conjectured that at high Ra number the compressibility effects can lead to reduce the driving temperature gradient for convection and reduce the likelihood of turbulent transition. Thus to provide more evidence, the effects of compressibility in the closed cavity problem will be the focus of this study.

Eckhoff and Storesletten [30] Kilfoil and Chew [31]

2.2.2 Effects of the Coriolis forces

The Coriolis force is a fictitious force, which acts on a body in rotation in the frame of reference of rotation, with respect to the inertial frame of reference. These forces can be characterized by a non-dimensional parameter known as Rossby number. It is defined as ratio of inertial to the Coriolis forces as

$$Ro = \frac{U}{2\Omega L}, \quad (2.5)$$

where U and L are the reference velocity and the length scales of the flow and Ω is the angular velocity of rotation. A small value of Rossby number signifies strong influence of Coriolis force and vice-versa. The initial studies were mainly focussed on investigating the effects of rotation in Rayleigh–Bénard convection problem [36] [37] [38][39] with gravity as a body force. For high Rossby number cases, Kunnen et al. [36] observed large-scale circulations in the turbulent convection in a cylindrical domain. However, under the influence of high rotation, the large-scale circulations were suppressed, which did not affect the heat transfer rate as it attained the maximum value at $Ro = 0.18$. A study by Stevens et al.[37] investigated the Rayleigh–Bénard convection with rotation about the vertical axis aligned with the direction gravity. They observed an increase in the value of Nusselt number compared to non-rotating cases with a decrease in the value of Rossby number. The formation of Ekman vortices, aligned from the hot wall to the cold wall, were found to be the reason for the heat transfer enhancement at low Rossby number. A similar conclusion was drawn by Chong et al. [38], where the stabilization effect of rotation led to the formation of coherent plume-like structures which resulted in heat transfer enhancement. However, there are few recent studies, where the effects of the Rossby number in centrifugal buoyancy-induced flows have been investigated. A study by Rouhi et al. [40] carried out an incompressible DNS with the Boussinesq approximation using the thin shell approximation $((r_o - r_i) \ll r_o)$, refer to figure 2.2). They observed a trend

of decrease in the heat transfer rate with the decrease in the Ro number with a strong bidirectional wind at $Ro = 1.25$ at which the heat transfer from the hot wall was minimum. More recently, a study by Jiang et al. [41] for a cylindrical annulus showed the effect of Rossby number on the wall heat transfer rate as well as on the flow structure. They conducted direct numerical simulations and experiments in the Boussinesq limits. At a low Rossby number, the flow was observed to reach a quasi-two-dimensional state with a reduction in the heat transfer rate. However, in the intermediate range of Rossby number, the three-dimensionality of the flow appeared to increase the heat transfer rate. The scaling of Nu number with Ra number followed the exponent value of 0.27. The power exponent of Nu – Ra scaling law showed an increased value compared to 0.211 of Bohn et al. [23]. The major difference in the physical conditions of Jiang et al. [41] as compared to Bohn et al. [23] lied in the values of Ro number which had a higher values in the latter study. A sudden increase in the value of Nu – Ra power exponent was observed at $Ra \geq 4 \times 10^{10}$ in the experiments of Jiang et al. [41]. The authors speculated the reason for this increase in the scaling coefficient was due to the transition to the ultimate regime [22]. The comparison of Nu – Ra scaling law with the the results of Bohn et al. [23] showed that the scaling exponent is higher than the

As discussed above, most of the studies on the effects of the Coriolis forces were mainly limited to the Boussinesq limits and for the problem configuration of little relevance to the internal air cooling system of a gas turbine. Thus, in this study, we are interested in compressible centrifugal buoyancy-induced flows, which represent the conditions of an internal air cooling system and therefore will be the approach adopted to understand the effects of the Rossby number.

2.3 Rotating cavity with an axial throughflow

While the previous the literature survey covered closed rotating cavities, rotating cavities with an axial throughflow are better representing the internal cooling system of a gas turbine engine. Figure 2.3 shows a schematic of the internal

air cooling section of the high-pressure compressor. The cavities form between the compressor disks of an engine, which is fed by axial throughflow of cold air. The fluid flow and heat transfer inside these annular cavities directly influences the cooling of the compressor blades and thermal expansion of the disks, which controls the blade-tip clearance. It is crucial to predict the flow physics and induced wall heat transfer rate inside these cavities accurately to optimize the overall design and enhance the safety of an engine.

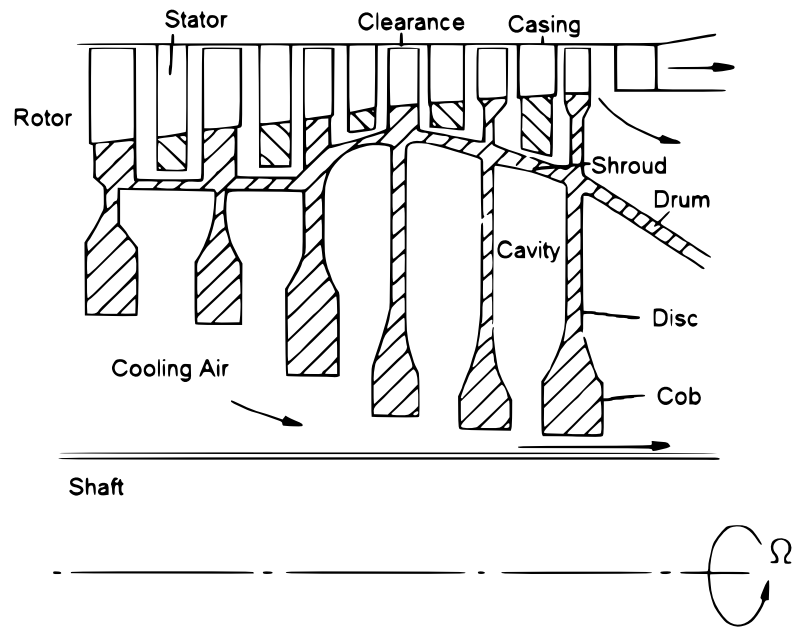


FIGURE 2.3: Schematic of the internal air cooling path of the high pressure compressor of a gas turbine engine showing the inter-disc cavities [4].

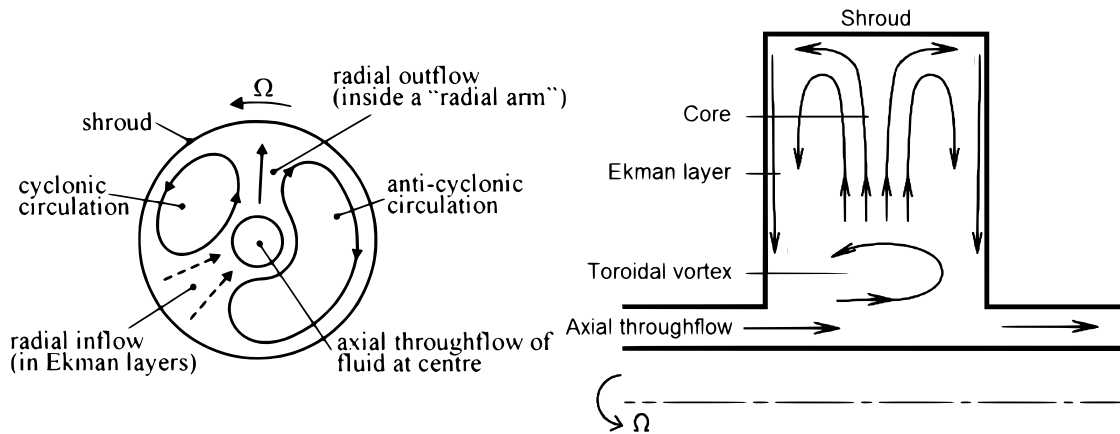


FIGURE 2.4: Schematic of flow structure proposed by Farthing et al. [42] from [4] (Left figure). Schematic of assumed flow structure by [43] (Right figure)

An initial experimental study on heated side disks was conducted by Farthing et al. [42][32] to characterize the flow structure and heat transfer rate. The experiments were carried out for a cavity with an unheated shroud (top surface of the cavity shown in figure 2.3) and imposed a radially decreasing temperature distribution on the side disks. The schematic of the proposed flow structure is presented in figure 2.4, showing a radial arm that is formed inside the cavity, bifurcating into a cyclonic and anti-cyclonic pair of vortices. They speculated that radial inflow occurs inside the boundary layers formed on the side disks. Additional experiments were carried out with insulated cobs (bottom section of the disk shown in figure 2.3), and two or three radial arms were observed inside the cavity, unlike the one radial arm observed for the cavity without any cobs. In their companion study [32], heat transfer results were presented. A radially increasing Nusselt number profile was reported for the cavity with radially increasing temperature distribution on the side disk. Another experimental study was carried out by Bohn et al. [44] to characterize the flow structure for radially increasing temperature distribution on the side disks. In their experiments, the fluid flow inside the cavity, without cobs and unheated shroud surface, was visualized by using a laser light sheet technique. A flow structure similar to Farthing et al. [42] was reported in their study for the radially increasing temperature profiles on the side disk. However, in both the studies [42] [44], the shroud surface was unheated, and the side disks were designed without any cobs. Both these conditions are necessary for the experimental results to be applicable to a real engine cavity.

Thus, to design an experiment more closely representing real engine conditions, an experimental rig for a single cavity with cobs was set up by Owen and Powell [45]. The velocity and heat transfer measurements were carried out for one heated disk, and all other surfaces were insulated. The measurement of circumferential velocity showed that the fluid in the core of the cavity rotates at a speed slower than the local disk speed and it further reduced with an increase in the temperature difference. To provide more insights on the experimental results of Owen and Powell [45], a subsequent numerical study was carried out by Owen et al. [46] by solving the compressible unsteady RANS equations with a κ - ϵ turbulence

model. A flow structure analogous to previous studies [42] [44] was reported with qualitative agreement with the experiments on the radial distribution of Nusselt number.

Further, to replicate the design architecture of the internal air path shown in figure 2.3, an experimental study on a rig with multiple cavities was conducted by Long et al. [47]. The velocity measurements were carried out for radially increasing temperature distribution on the side disks with a heated shroud. The variation of the circumferential velocity component showed a peak in the inner part of the cavity and decreased to solid-body rotation in the outer part. Based on these measurements, the radial extent of the toroidal vortex was reported to be $r \approx 0.6$. No measurements were carried out inside the disk boundary layers due to the flow unsteadiness. Thus, it was challenging to interpret the overall flow structure, as concluded by the authors. A numerical study was carried out by Sun et al. [48] for a cavity configuration matching the experiments of Long et al. [47] to overcome the limitations of the experimental studies. Both LES and URANS calculations were carried out and compared against the experimental data. A strong unsteadiness in the flow structure was observed, and LES results were reported to be in better agreement with experiments. However, no detailed information about the flow structure was provided in their study.

To isolate the effects of axial Reynolds number, rotational Reynolds number and Rayleigh number on the disk temperature profiles, an experimental study was conducted by Atkins and Kanjirakkad [49] on the multi cavity rig of Long et al. [47]. The experiments were performed for high Rossby number and low Rossby number conditions. In both situations, the Rayleigh number was varied to study its effect on the disk temperature distribution. A negligible effect of Rayleigh number on temperature distribution in the inner region of the cavity was reported, with a strong dependence on axial Reynolds number. At low rotational Reynolds number, the mixing level between the axial bore flow and the outer cavity flow was increased as compared to the high rotational Reynolds number, which suppressed the mixing. Further, a study was carried out by Puttock-Brown et al. [50] to understand the dependence of shroud heat transfer rate on the axial Reynolds number. The shroud

Nusselt number was reported to be dependent only on the shroud Rayleigh number with no discernible effect of axial Reynolds number. The Nusselt number values were in accord with the correlation for turbulent free convection from a flat plate. Subsequently, 3D unsteady RANS calculations were conducted to support the experimental results and showed the existence of Rayleigh-Bénard streaks along with the shroud. A pioneering study by Owen and Tang [43] proposed a theoretical model based on the solution of Ekman-layer equations. They assumed a flow structure as shown in figure 2.4 and assumed a laminar, inviscid and adiabatic core to find the approximate solution for the equations. The final solution showed that the local Nusselt number based on the core temperature is proportional to the core Grashof number, which is the ratio of the Rayleigh number to the Prandtl number, to the power of one fourth. Very recent large-eddy simulation results were reported by Pitz et al. [51] to study the effect of axial throughflow on the centrifugal-buoyancy induced flow. The study was conducted on a closed cavity configuration extended to study the effect of bore flow. Thus, the flow features were also compared with the closed cavity results to quantify the effect of bore flow. Pitz et al. [51] carried out the simulations for two axial Reynolds numbers of 10^3 and 2×10^3 at a fixed Rayleigh number of 10^8 . The thickness of the disk boundary layer was found to be consistent with a laminar Ekman layer, and flow in the core region was observed to be turbulent. The thickness of the kinetic boundary layer formed on the side disks was observed to be varying in the radial direction. However, the authors did not provide any information about the flow structure in the cavity with heated disks and a shroud.

Clearly, there is no general interpretation of the flow structure that develops in a cavity with heated disks, radially increasing temperature profiles and a heated shroud. This is mainly due to the difficulty of the experiments to measure and visualize the fluid flow inside the cavity due to the flow unsteadiness [47]. To the author's knowledge, there is no highly resolved numerical study which has addressed this problem. Hence, in this study, highly resolved large eddy simulations were carried out for the high Rossby number experimental conditions of [49] to understand the overall flow structure and study the effect of Rayleigh number.

2.4 Research Questions

Based on the literature review, it can be summarized that there is a gap in understanding the effect of compressibility and the Coriolis forces in the closed cavity problem. A parametric study conducted here might explain the reason behind the reduced scaling coefficient observed in the experiments by Bohn et al. [23] on which there is a lack of consensus in the current literature. Further, the cavity with an axial throughflow problem there exists no clear depiction of flow structure for real engine conditions due to the challenges in the experiments and lack of high-fidelity numerical studies on this problem. Thus, the research statement of this work is as follows.

To characterize the effects of compressibility, the Coriolis force and the Rayleigh number and consequently the heat transfer for centrifugal buoyancy-induced flow in closed cavities and in cavities with axial throughflow.

The research statement further leads to the following research questions that this thesis attempts to answer:

- *What is the physical reason behind the reduced scaling exponent of $Nu-Ra$ power law observed in the experiments?*
- *How the compressibility and the Coriolis forces influence the flow structure as well as the wall heat transfer rate?*
- *What is the generalized flow depiction in a cavity with axial throughflow for the conditions of heated side disks and shroud? How the variation in Ra number affects flow structure and induced wall heat transfer rate?*

Chapter 3

Governing equations and Numerical techniques

3.1 Navier-Stokes Equations

The compressible Navier-Stokes equations in the conservative form are solved for the problem under consideration. The flow is assumed to be an ideal gas with the ratio of specific heat γ , and the dynamic viscosity of μ_∞ and the thermal conductivity is κ_∞ defined at reference temperature of T_∞ . With u , v and w representing the non-dimensional velocity components in the z , r and θ directions, respectively; the non-dimensional compressible Navier-Stokes equations in cylindrical coordinates mapped to the computational space ξ , η and z are

$$\frac{\partial \hat{\mathbf{Q}}}{\partial t} + \frac{\partial \hat{\mathbf{E}}}{\partial z} + \frac{\partial \hat{\mathbf{F}}}{\partial r} + \frac{1}{r} \frac{\partial \mathbf{G}}{\partial \theta} = \hat{\mathbf{S}}_{\mathbf{v}}, \quad (3.1)$$

where the inviscid flux vectors in generalized coordinates are given as

$$\hat{\mathbf{Q}} = J\mathbf{Q}, \quad \hat{\mathbf{E}} = r_\eta \mathbf{E} - z_\eta \mathbf{F}, \quad \hat{\mathbf{F}} = -r_\xi \mathbf{E} + z_\xi \mathbf{F}, \quad (3.2)$$

and the source term $\hat{\mathbf{S}}_{\mathbf{v}}$ is computed as

$$\hat{\mathbf{S}}_{\mathbf{v}} = \frac{\partial \hat{\mathbf{E}}_{\mathbf{v}}}{\partial \xi} + \frac{\partial \hat{\mathbf{F}}_{\mathbf{v}}}{\partial \eta} + \frac{J}{r} \frac{\partial \hat{\mathbf{G}}_{\mathbf{v}}}{\partial \theta} - \frac{J}{r} \mathbf{H}, \quad (3.3)$$

where

$$\hat{\mathbf{E}}_{\mathbf{v}} = r_{\eta} \mathbf{E}_{\mathbf{v}} - z_{\eta} \mathbf{F}_{\mathbf{v}}, \quad \hat{\mathbf{F}}_{\mathbf{v}} = -r_{\xi} \mathbf{E}_{\mathbf{v}} + z_{\xi} \mathbf{F}_{\mathbf{v}}. \quad (3.4)$$

In equation 3.2, the inviscid flux vectors are

$$\begin{aligned} \mathbf{Q} &= [\rho, \rho u, \rho v, \rho w, \rho E]^T \\ \mathbf{E} &= [\rho u, \rho u u + p, \rho u v, \rho u w, u(\rho E + p)]^T \\ \mathbf{F} &= [\rho v, \rho u v, \rho v v + p, \rho v w, v(\rho E + p)]^T \\ \mathbf{G} &= [\rho w, \rho u w, \rho v w, \rho w w + p, w(\rho E + p)]^T, \end{aligned} \quad (3.5)$$

where the total energy, $E = T/[\gamma(\gamma - 1)Ma_{\infty}^2] + (1/2)u_i u_i$. The viscous flux derivatives in equation 3.4 and the additional terms (H) in cylindrical coordinates in equation 3.3 are given as

$$\begin{aligned} \mathbf{E}_{\mathbf{v}} &= [0, \tau_{zz}, \tau_{zr}, \tau_{z\theta}, -q_z + u\tau_{zz} + v\tau_{zr} + w\tau_{z\theta}]^T \\ \mathbf{F}_{\mathbf{v}} &= [0, \tau_{rz}, \tau_{rr}, \tau_{r\theta}, -q_r + u\tau_{rz} + v\tau_{rr} + w\tau_{r\theta}]^T \\ \mathbf{G}_{\mathbf{v}} &= [0, \tau_{\theta z}, \tau_{\theta r}, \tau_{\theta\theta}, -q_{\theta} + u\tau_{\theta z} + v\tau_{\theta r} + w\tau_{\theta\theta}]^T \\ \mathbf{H} &= [\rho v, \rho u v - \tau_{zr}, \rho v v - \rho w w - \tau_{rr} + \tau_{\theta\theta}, 2\rho v w - 2\tau_{\theta r}, \\ &\quad v(\rho E + p) + q_r - u\tau_{rz} - v\tau_{rr} - w\tau_{r\theta}]^T. \end{aligned} \quad (3.6)$$

In above equations, $J = z_{\xi} r_{\eta} - z_{\eta} r_{\xi}$ is the determinant of coordinate tranformation matrix from computational sapce to physical space and vice-versa.

The viscous stress tensor and the heat-flux vector are computed as

$$\begin{aligned}
\tau_{zz} &= \frac{2\mu}{3Re_\infty} \left[2 \frac{\partial u}{\partial z} - \frac{\partial v}{\partial r} - \frac{1}{r} \left(\frac{\partial w}{\partial \theta} + v \right) \right] \\
\tau_{rr} &= \frac{2\mu}{3Re_\infty} \left[- \frac{\partial u}{\partial z} + 2 \frac{\partial v}{\partial r} - \frac{1}{r} \left(\frac{\partial w}{\partial \theta} + v \right) \right] \\
\tau_{\theta\theta} &= \frac{2\mu}{3Re_\infty} \left[- \frac{\partial u}{\partial z} - \frac{\partial v}{\partial r} + \frac{2}{r} \left(\frac{\partial w}{\partial \theta} + v \right) \right] \\
\tau_{rz} &= \frac{\mu}{Re_\infty} \left[\frac{\partial u}{\partial r} + \frac{\partial v}{\partial z} \right] \\
\tau_{\theta z} &= \frac{\mu}{Re_\infty} \left[\frac{\partial w}{\partial z} + \frac{1}{r} \frac{\partial u}{\partial \theta} \right] \\
\tau_{\theta r} &= \frac{\mu}{Re_\infty} \left[\frac{1}{r} \left(\frac{\partial v}{\partial \theta} - w \right) + \frac{\partial w}{\partial r} \right] \\
q_z &= \frac{-\mu}{Pr(\gamma - 1)Ma_\infty^2 Re_\infty} \frac{\partial T}{\partial z} \\
q_r &= \frac{-\mu}{Pr(\gamma - 1)Ma_\infty^2 Re_\infty} \frac{\partial T}{\partial r} \\
q_\theta &= \frac{-\mu}{Pr(\gamma - 1)Ma_\infty^2 Re_\infty} \frac{1}{r} \frac{\partial T}{\partial \theta}
\end{aligned} \tag{3.7}$$

The molecular viscosity is computed using Sutherland's law [52], $\mu = T^{3/2}(1 + S/T_\infty)/(T + S/T_\infty)$, where S is the Sutherland temperature. The equation of state is $p = \rho T/(\gamma Ma_\infty^2)$ is used to close the above system of equations. The above system of equations for ideal fluid are non-dimensionalized with length L_∞ , velocity scale U_∞ , density ρ_∞ , temperature T_∞ and pressure $p_\infty = \rho_\infty U_\infty^2$ yield the following non-dimensional parameters

$$Re_\infty = \frac{\rho_\infty U_\infty L_\infty}{\mu_\infty}, \quad Pr_\infty \equiv \frac{\mu_\infty c_p}{\kappa_\infty}, \quad Ma_\infty \equiv \frac{U_\infty}{(c_p(\gamma - 1)T_\infty)^{1/2}}, \tag{3.8}$$

where Re_∞ , Pr_∞ and Ma_∞ are the reference Reynolds, Prandtl and Mach numbers, respectively.

3.2 Numerical method

The simulations (DNSs in chapters 3, 4, 5 and LESs in 6) have been conducted by using the in-house compressible Navier–Stokes solver HiPSTAR (High Performance Solver for Turbulence and Aeroacoustics Research) [53]. A fourth-order accurate standard finite-difference scheme is used for spatial discretization in the axial (z) and the radial (r) directions. In the azimuthal (θ) direction, a Fourier pseudo-spectral method is used for discretization (refer to Appendix E for more details). Time integration is achieved using a low-storage fourth-order accurate Runge–Kutta scheme. To enhance the stability of the code skew-symmetric splitting of the non-linear terms and an 11 point wave-number-optimized filter is applied to the solution after each full-time step with low weighting to reduce grid-to-grid point oscillations.

The code has been used and validated for a range of turbomachinery applications ([54] [55] [56]) and more recently for simulating the centrifugal buoyancy-induced flow in a closed cavity [57] [58].

3.2.1 Multi-dimensional stability analysis

To conduct the stability analyses in chapter 5, a forced Navier–Stokes simulation approach is introduced in this section. For simplicity, the governing equations are presented in tensor notation. A base state at time $t = 0$ is defined for the conservative variables $\rho^0 u_i^0$, $\rho^0 E^0$ and density ρ^0 . At the beginning of the simulation, all the spatial derivatives of the base state for the continuity, momentum and energy equations are calculated as

$$\left. \frac{\partial \rho}{\partial t} \right|_{t=0} = - \frac{\partial}{\partial x_k} (\rho^0 u_k^0), \quad (3.9)$$

$$\left. \frac{\partial (\rho u_i)}{\partial t} \right|_{t=0} = - \frac{\partial}{\partial x_k} [\rho^0 u_i^0 u_k^0 + p^0 \delta_{ik} - \tau_{ik}^0], \quad (3.10)$$

$$\left. \frac{\partial (\rho E)}{\partial t} \right|_{t=0} = - \frac{\partial}{\partial x_k} [u_k^0 (\rho^0 E^0 + p^0) + q_k^0 - u_i^0 \tau_{ik}^0], \quad (3.11)$$

and stored. The definition of viscous stress tensor and the heat-flux vector are given in equation 3.7. The temporal derivatives of the base state defined in equations 3.9, 3.10 and 3.11 are called forcing terms and maintained as a reference base state. The behaviour of small perturbations is then investigated by subtracting the forcing terms from the solution of the Navier–Stokes equations at each Runge–Kutta substep as the solution progresses in time as follows

$$\frac{\partial \rho}{\partial t} = -\frac{\partial}{\partial x_k}(\rho u_k) - \frac{\partial \rho}{\partial t}\Big|_{t=0}, \quad (3.12)$$

$$\frac{\partial}{\partial t}(\rho u_i) = -\frac{\partial}{\partial x_k}[\rho u_i u_k + p \delta_{ik} - \tau_{ik}] - \frac{\partial}{\partial t}(\rho u_i)\Big|_{t=0}, \quad (3.13)$$

$$\frac{\partial}{\partial t}(\rho E) = -\frac{\partial}{\partial x_k}[u_k(\rho E + p) + q_k - u_i \tau_{ik}] - \frac{\partial}{\partial t}(\rho E)\Big|_{t=0}. \quad (3.14)$$

3.2.2 Large-Eddy Simulations

Large-Eddy simulations (LES) are conducted in chapter 7 for a cavity with an axial-throughflow. The implicit filtering is used in LES with the filter size same as the grid-spacing. Thus the turbulence scales smaller than the grid size are modelled with a cut-off in the physical space, and the total molecular stress tensor (τ_{ij}^t) is obtained as a sum of molecular stress tensor (τ_{ij}) and sub-grid scale tensor (τ_{ij}^{SGS})

$$\begin{aligned} \tau_{ij}^t &= \tau_{ij} + \tau_{ij}^{SGS}, \\ &= \frac{\mu + \mu_t}{Re_\infty} \left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} - \frac{2}{3} \frac{\partial u_k}{\partial x_k} \delta_{ij} \right), \end{aligned} \quad (3.15)$$

where μ_t is the sub-grid scale viscosity calculated by using wall-adapting local eddy viscosity (WALE) model proposed by Nicoud and Ducros [59]. This model relies on the square of the velocity gradient tensor and provides a cubic decay of eddy viscosity in the vicinity of walls and thus suitable for wall-bounded flows.

The eddy viscosity in the WALE model is computed as

$$\mu_t = \bar{\rho}(C_w\Delta)^2 \frac{(\mathcal{S}_{ij}^d \mathcal{S}_{ij}^d)^{3/2}}{(S_{ij}^d S_{ij}^d)^{5/2} + (\mathcal{S}_{ij}^d \mathcal{S}_{ij}^d)^{5/4}}, \quad (3.16)$$

where $\mathcal{S}_{ij}^d$ is the traceless, symmetric tensor of the square of the velocity gradient tensor and C_w is the model coefficient.

Chapter 4

Centrifugal buoyancy-induced flow in a closed cavity

In this chapter, a closed cavity configuration is investigated by carrying out the direct numerical simulations for the physical conditions replicating the experiments of Bohn et al. [23]. The first part of the chapter characterizes the flow structure within the cavity, following which comparisons of the heat-transfer results with the experiments [23] are made. In the latter part, the differences in the results with an incompressible formulation using the Boussinesq approximation are highlighted.

The results of this chapter have been published in the conference proceedings of the Australasian Fluid Mechanics Conference 2018 (Saini et al. [57]) and in the International Journal of Heat and Fluid Flow (Saini and Sandberg [58])

4.1 Numerical Setup

The computational domain is a closed annular cavity that rotates at an angular velocity Ω_∞ about its axis, as shown in figure 4.1. The inner and outer radii are r_i and r_o , respectively, and the mean radius is $r_m = (r_o + r_i)/2$. The axial extent of the cavity is H . The outer wall is hot at a temperature T_H and the inner wall

is cold at a temperature T_C , i.e. $\Delta T = T_H - T_C > 0$. The compressible Navier–Stokes equations as discussed in chapter 3 for the conservative variables are solved in an inertial frame for the cavity that is filled with a perfect gas with specific heat at constant pressure c_p and the ratio of specific heats γ .

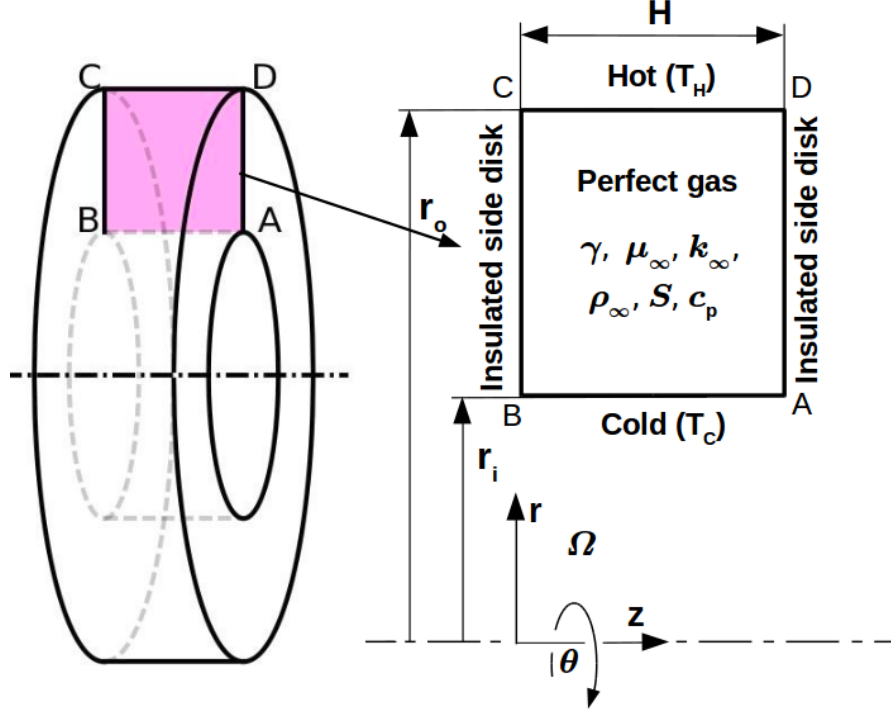


FIGURE 4.1: Schematic of closed annular cavity. The walls of the cavity undergo solid-body rotation about the z -axis.

At $T_\infty \equiv (T_H + T_C)/2$, the dynamic viscosity of the gas is μ_∞ and the thermal conductivity is κ_∞ . The governing equations are non-dimensionalised with length $L_\infty = (r_o - r_i)$, free-fall velocity $U_\infty = [\Omega_\infty^2 r_m (\Delta T / T_\infty) L_\infty]^{1/2}$, density ρ_∞ , temperature T_∞ and pressure $p_\infty = \rho_\infty U_\infty^2$ yield the following non-dimensional numbers that completely specify this problem as

$$\begin{aligned}
 Ra_\infty &= \frac{\rho_\infty^2 c_p (\Omega_\infty^2 r_m) (\Delta T / T_\infty) (r_o - r_i)^3}{\mu_\infty \kappa_\infty} = \left(\frac{\rho_\infty U_\infty L_\infty}{\mu_\infty} \right)^2 \frac{\mu_\infty c_p}{\kappa_\infty} = Re_\infty^2 Pr_\infty, \\
 Pr_\infty &\equiv \frac{\mu_\infty c_p}{\kappa_\infty} = 0.71, \quad \gamma = 1.4, \quad \frac{S}{T_\infty} = 0.368, \quad Ma_\infty \equiv \frac{U_\infty}{(c_p (\gamma - 1) T_\infty)^{1/2}}, \\
 \frac{\Delta T}{T_\infty}, \quad \frac{r_i}{L_\infty} &= 1.083, \quad \frac{r_o}{L_\infty} = 2.083, \quad \frac{H}{L_\infty} = 1.043,
 \end{aligned} \tag{4.1}$$

where Ra_∞ , Re_∞ , Pr_∞ and Ma_∞ are the reference Rayleigh, Reynolds, Prandtl and Mach numbers, respectively. To characterize the effect of Coriolis forces in the system, the Rossby number is defined as

$$\begin{aligned} Ro &\equiv \frac{[\Omega_\infty^2 r_m (\Delta T / T_\infty) (r_o - r_i)]^{1/2}}{2\Omega_\infty (r_o - r_i)} \\ &= \frac{1}{2^{3/2}} \left(\frac{1 + r_i/r_o}{1 - r_i/r_o} \right)^{1/2} \left(\frac{T_H - T_C}{T_m} \right)^{1/2}, \end{aligned} \quad (4.2)$$

which appears in the boundary conditions in the inertial frame of reference. The boundary conditions for this problem are thus $u = v = 0$ for all walls of the cavity and, since all walls are rotating at constant angular velocity of $\Omega = 1/(2Ro)$,

$$\begin{aligned} w(z, r = r_i/L_\infty, \theta) &= \Omega r_i/L_\infty, \\ w(z, r = r_o/L_\infty, \theta) &= \Omega r_o/L_\infty, \\ w(z = 0, r, \theta) &= w(z = H/L_\infty, r, \theta) = \Omega r. \end{aligned} \quad (4.3)$$

The outer and inner cylindrical walls of the cavity are isothermal with

$$\begin{aligned} T(z, r = r_o/L_\infty, \theta) &= T_H/T_\infty = 1 + 4Ro^2[(1 - r_i/r_o)/(1 + r_i/r_o)], \\ T(z, r = r_i/L_\infty, \theta) &= T_C/T_\infty = 1 - 4Ro^2[(1 - r_i/r_o)/(1 + r_i/r_o)]. \end{aligned} \quad (4.4)$$

The side disks of the cavity are insulated so that $q_z = 0 \Rightarrow \partial T / \partial z = 0$ on both sides. In this study, the effect of gravity is neglected as the centrifugal force is much larger than the gravitational force ($\Omega^2 r / g \gg 1$). In the natural convection problem the response of the system is defined by Nusselt number:

$$Nu \equiv \frac{q_{convection}}{q_{conduction}} = \frac{\ln(r_o/r_i)}{\Delta T / T_\infty} \left[\frac{d\bar{T}}{dr} \right]_{r=1}, \quad (4.5)$$

which is the non-dimensional wall heat transfer coefficient. Thus for a fixed geometrical parameters (r_i/L_∞ , r_o/L_∞ and H/L_∞) of the cavity, the flow physics and the induced heat transfer rate inside the cavity for an ideal gas of Pr_∞ are governed by Ra_∞ , Ro and Ma_{infty} , which are independent non-dimensional parameters.

Case	Ra	Re	$\Delta T/T_\infty$	Ro	$N_r \times N_z \times N_\theta$
1	10^5	375.3	0.157	0.25	$120 \times 120 \times 194$
2	10^6	1186.7	0.232	0.30	$160 \times 160 \times 194$
3	10^7	3752.9	0.179	0.26	$200 \times 200 \times 194$
4	10^8	11867.8	0.137	0.23	$432 \times 432 \times 258$

TABLE 4.1: DNS cases: N_r , N_z and N_θ are number of grid points in respective directions.

4.2 Results and Discussion

Direct numerical simulations are carried out for Rayleigh numbers in the range $10^5 < Ra < 10^8$ as shown in table 4.1. The corresponding grid size for each of the Ra numbers is also provided. The conditions specified in table 4.1 satisfy the experimental conditions of [23] for Rayleigh number in the range $10^6 \leq Ra \leq 10^8$. The value of Ma number in all the cases is 0.05, very close to the range of Ma numbers computed from the experimental data of [23]. The Rossby number for the case of $Ra = 10^5$ is 0.25, close to the conditions of the other Ra numbers for further comparison.

4.2.1 Grid quality

The grid convergence results for the $Ra = 10^7$ case are shown in table 4.2 for the three different levels of grid. A minimal variation in the values of Nu number and core temperature from grid-level 2 to 3 shows the grid convergence. Thus, grid-level 2 is found to be adequate for the case of $Ra = 10^7$. The grid for the other Ra numbers was designed based on the resolution of $Ra = 10^7$ by using the scaling law derived from Kolmogorov's theory of isotropic turbulence for which the total number of grid points scale with $Re^{9/4} \equiv (Ra/Pr)^{9/8}$ [60].

Thus, the current grid resolution for $Ra = 10^5$, 10^6 and 10^8 is found to be of same order of magnitude as the resolution required for the isotropic turbulence. However, the near-wall resolution in the thermal boundary layer formed in the radial direction, as well as the grid resolution in the core, is estimated by following

Grid for $Ra = 10^7$	$N_r \times N_z \times N_\theta$	Nu_h	% change in Nu_h	T_{core}	% change in T_{core}
Level 1	$160 \times 160 \times 194$	10.041	-3.55	1.02154	0.0205
Level 2	$200 \times 200 \times 194$	10.375	-0.35	1.02132	0.00098
Level 3	$300 \times 300 \times 194$	10.4120	-	1.02133	-

TABLE 4.2: Grid sensitivity for $Ra = 10^7$.

Ra	Hot wall N_{BL}^{th}	Cold wall N_{BL}^{th}	min N_{BL}^{th} required [61]	max Δr in core	max Δr in core required [61]
10^5	27	24	4	0.016	0.036
10^6	32	30	5	0.006	0.017
10^7	33	31	6	0.005	0.0083
10^8	61	57	9	0.0023	0.0038

TABLE 4.3: The comparison of grid resolution in the thermal boundary layer against the guidelines provided by Shishkina et al. [61]

the guidelines of Shishkina et al. [61], and compared against the lower bounds for the same provided in their study as shown in table 4.3. The thermal boundary layer thickness for comparison in table 4.3 is calculated based on the maximum value of T_{rms} profiles averaged over time, the azimuthal and the axial direction, and N_{BL}^{th} are the numbers of grid points in the thermal boundary layer. As per the comparison shown in table 4.3, the resolution appears sufficient for all Ra numbers in the radial direction.

In addition, the grid-spacings in wall units for the top wall (hot) and the side insulated wall are shown in figures 4.2 and 4.3, respectively. For all the cases, the wall-normal spacing Δr^+ is less than 0.1 at the top wall and in the axial direction, the grid is stretched with maximum grid-spacing $\Delta z^+ < 0.5$ for all Ra numbers. In the azimuthal direction, $r\Delta\theta^+$ is close to 4 for the $Ra = 10^8$ and less than that for all other cases. Similarly, for the insulated wall, the wall-normal spacing Δz^+ is less than 0.05 for all the Ra numbers and spacing in the radial direction Δr^+ is also less than 1. The maximum value of $r\Delta\theta^+ \approx 8$ lies near the top wall for $Ra = 10^8$.

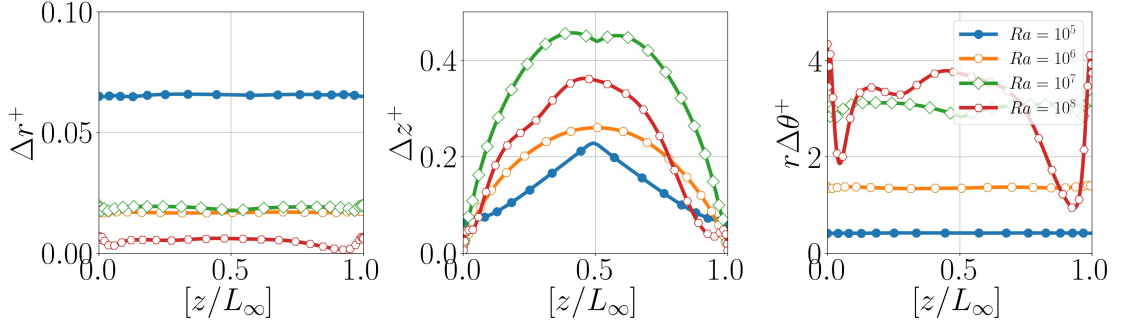


FIGURE 4.2: Grid-spacing in wall units for the top hot wall at different Rayleigh numbers

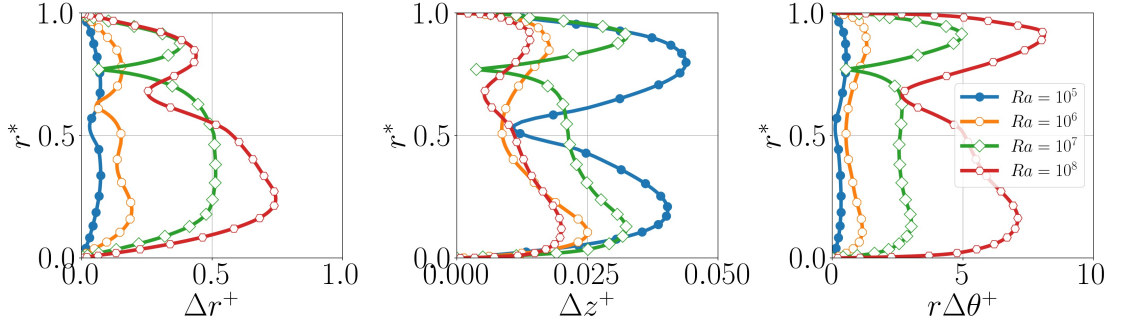


FIGURE 4.3: Grid-spacing in wall units for the side insulated wall at different Rayleigh numbers, where $r^* = [r - (r_i/L_\infty)]/[(r_o - r_i)/L_\infty]$

To further ascertain whether the turbulent scales in the core are adequately resolved in the azimuthal direction, azimuthal wavenumber spectra of radial velocity in the middle of the cavity are plotted for different Ra numbers as shown in figure 4.4(a). The spectra follow the $-5/3$ power law in the inertial range and decay of about four orders of magnitude is observed in the amplitude of energy for all the Ra numbers. These comparisons and values indicate that the simulations are well resolved and thus reliable.

For all the cases a non-dimensional time step was used to satisfy the Courant-Friedrich-Lewy (CFL) stability condition with maximum CFL number less than one. The simulations were run for 30 rotations for each case, and the flow statistics were calculated by averaging over the last 12 rotations.

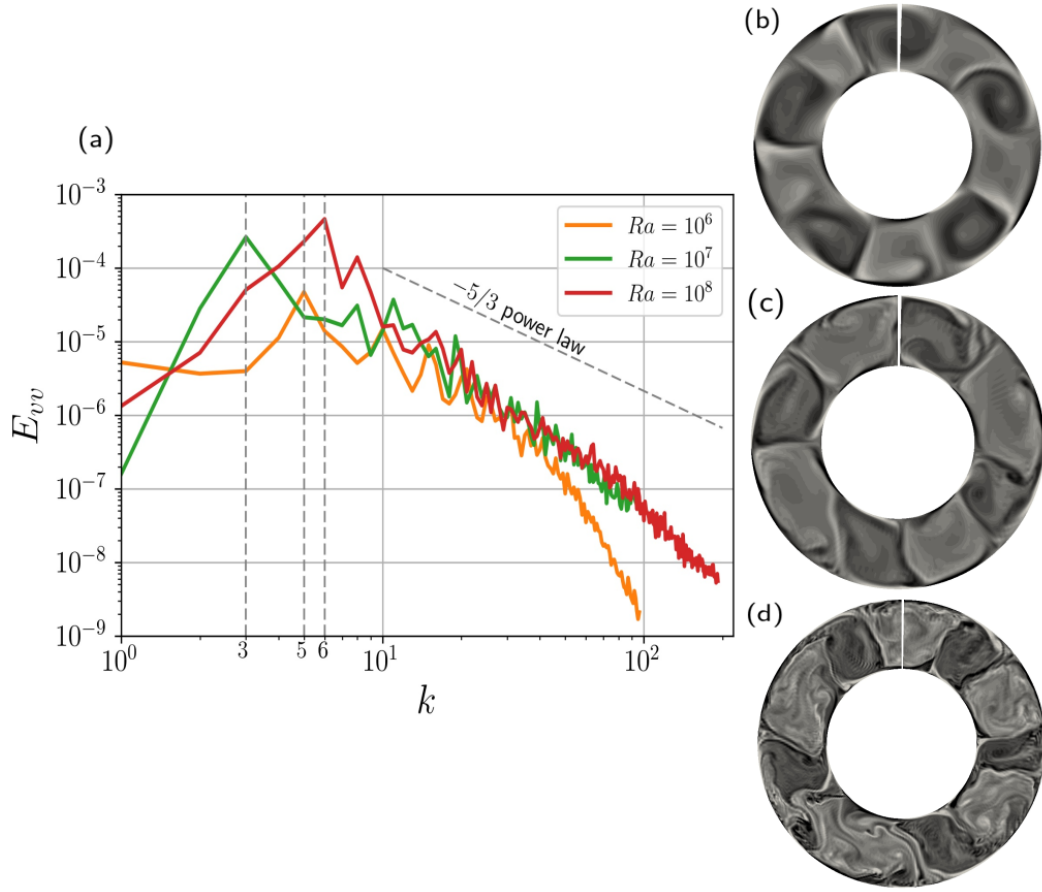


FIGURE 4.4: (a) Wavenumber spectra of numerical probe located at the middle of the cavity at different Rayleigh numbers and instantaneous contours of azimuthal vorticity for (b) $Ra = 10^6$, (c) $Ra = 10^7$ and (d) $Ra = 10^8$.

4.2.2 Instantaneous Flow Fields

Figures 4.5, 4.6 and 4.7, show the instantaneous radial velocity, azimuthal velocity and temperature contours, respectively, in the $r - z$ plane at mid-axial position of the cavity. The consecutive inward and outward radial flow produces a pair of vortices in the cavity, as shown in figure 4.5. Further, the radial outflow and inflow bifurcate to form a cyclonic arm, which rotates in the direction of rotation and an anti-cyclonic arm, which rotates in the opposite direction, as depicted by the positive and negative azimuthal velocity, respectively, in figure 4.6. This leads to a pair of cyclonic and anti-cyclonic vortices, as shown in figure 4.7. These vortices are more discernible at a low Ra number and become chaotic as the Ra number is increased. Figure 4.7 shows the core of the cavity has more temperature

uniformity at high Rayleigh number as turbulence leads to the breakdown of the convection rollers which also leads to a decrease in the thermal boundary layer thickness. The number of vortex pairs in the cavity is also different for different Ra number.

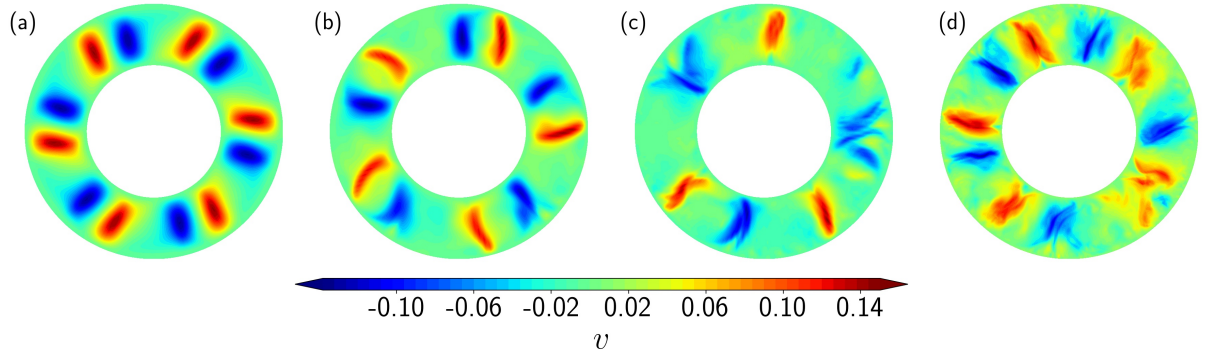


FIGURE 4.5: Instantaneous radial velocity contours at mid-axial position $z = (1/2)(H/L_\infty)$ of the cavity rotating in the clockwise direction for (a) $Ra = 10^5$, (b) $Ra = 10^6$, (c) $Ra = 10^7$, and $Ra = 10^8$.

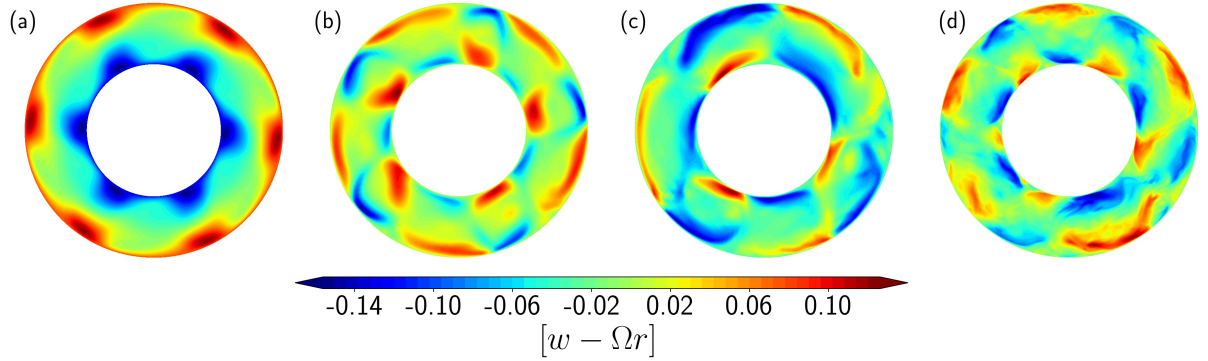


FIGURE 4.6: Instantaneous azimuthal velocity contours in the frame of reference of rotation at mid-axial position $z = (1/2)(H/L_\infty)$ of the cavity rotating in the clockwise direction for (a) $Ra = 10^5$, (b) $Ra = 10^6$, (c) $Ra = 10^7$, and $Ra = 10^8$.

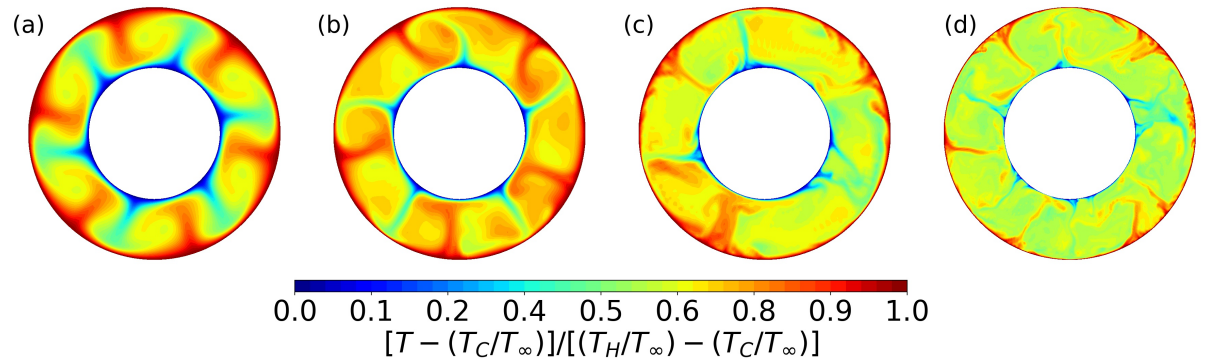


FIGURE 4.7: Instantaneous temperature contours at mid-axial position $z = (1/2)(H/L_\infty)$ of the cavity rotating in the clockwise direction for (a) $Ra = 10^5$, (b) $Ra = 10^6$, (c) $Ra = 10^7$, and $Ra = 10^8$.

The number of convection rollers is six for $Ra = 10^5$ and is reduced to five for $Ra = 10^6$. For higher Ra , the number of rollers varies between instantaneous snapshots. To quantify the number of large scale structures, we can refer to wavenumber spectra, as shown in figure 4.4(a). The spectra show a peak in the energy at a wavenumber corresponding to the number of rollers in the domain. For instance, for $Ra = 10^6$, the peak energy level is at $k = 5$; thus there are five convection rollers in the domain as shown in the vorticity contour in figure 4.4(b). Similarly, for $Ra = 10^7$, the peak energy is at wavenumber $k = 3$, which means there are three vortex pairs in the domain. For $Ra = 10^8$, there are six convection rollers as shown by peak energy level at wavenumber $k = 6$ in figures 4.4(a) and 4.4(d). The lack of a consistent trend in the variation of the number of rollers with the Ra number is because we have also varied Ro number, which influences the number of rollers in the domain.

In addition, these convection rollers and corresponding radial plumes merge with each other to form a larger roller in the domain. It can be observed for the case of $Ra = 10^7$, as shown in figure 4.7(c), where at approximately the 4'o clock position, two plumes of cold fluid are rising from the inner wall, forming a large convection roller, comparable to the size of the other two dominant rollers. A similar phenomenon of the merger of convection rollers can be seen for $Ra = 10^8$ as well at approximately 3'o clock position in figure 4.7(d).

The occurrence of merging convection rollers is more frequent in the transient phase of the flow, which can be distinctly observed for the low Ra number case as shown in figure 4.8, which shows the time evolution of instability for $Ra = 10^5$. The initial state of the system is conduction with a linear temperature profile from the inner (cold) to the outer (hot) wall. This state is clearly shown in figure 4.8(a), where the instability is in an embryonic stage at the interface between the hot and the cold fluid. As the flow evolves, the plumes of cold fluid (high density) rise in an asymmetric manner from the inner wall due to the centrifugal force as shown in figure 4.8(b). Initially, there are nine plumes in the domain, which grow to form “mushroom” shaped structures, as shown in figure 4.8(c). As time goes by, these fully grown convection plumes interact with each other

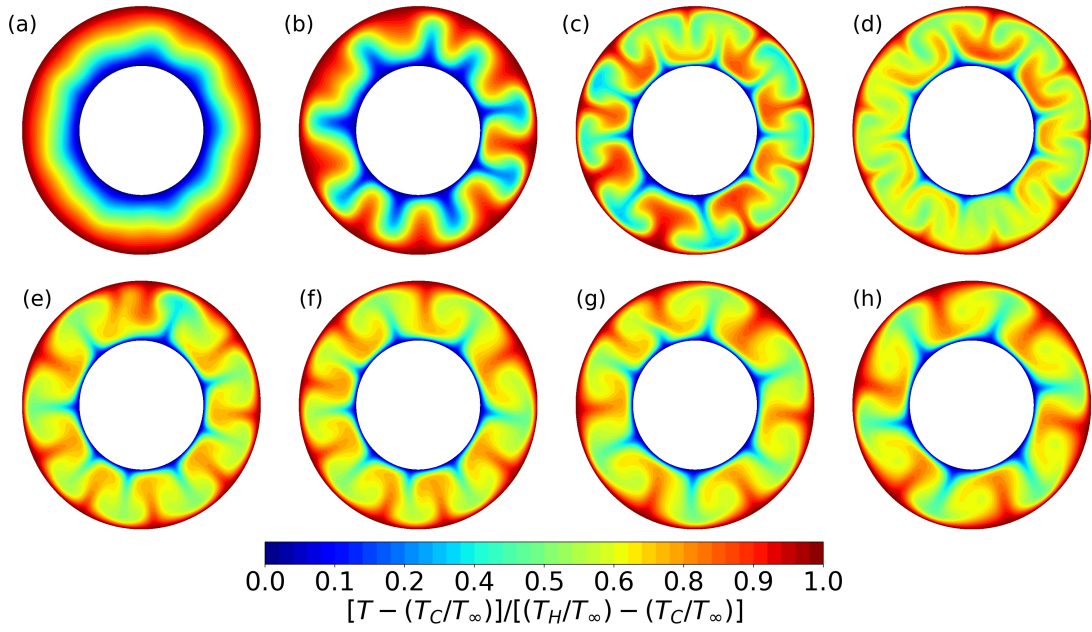


FIGURE 4.8: Time evolution of temperature at the mid-axial position $z = (1/2)(H/L_\infty)$ of the cavity rotating in the clockwise direction for $Ra = 10^5$ and $Ro = 0.25$ at non-dimensional time units (a-h) $t = 22.6, 26.4, 28.3, 30.2, 45.2, 67.9, 113.1$ and 448.6 .

[figures 4.8(d-g)] and finally reach fully developed state as shown in figure 4.8(h). The time evolution of instability in the present study is quite different from the results of Pitz et al. [62]. In particular, we observe modes with higher azimuthal wavenumbers than the dominant mode in the saturated state to grow faster in the linear regime. A detailed investigation into this will be discussed in chapter 5. The time dependence of the number of rollers in the domain was also reported by King et al. [26] as the unsteadiness in the flow structure. This unsteadiness of the flow structure can be observed in the wall heat transfer rate or Nu number also. Figure 4.9 shows the variation of Nu number, averaged in the azimuthal and axial direction at the outer wall, with time. After the initial transient phase up to $t = 300$ units, periodic oscillations are observed in the value of Nu , and the system reaches a fully developed state. Thus beyond $t = 300$ time units, the flow appears statistically converged, allowing us to calculate the time-averaged statistics for further analysis.

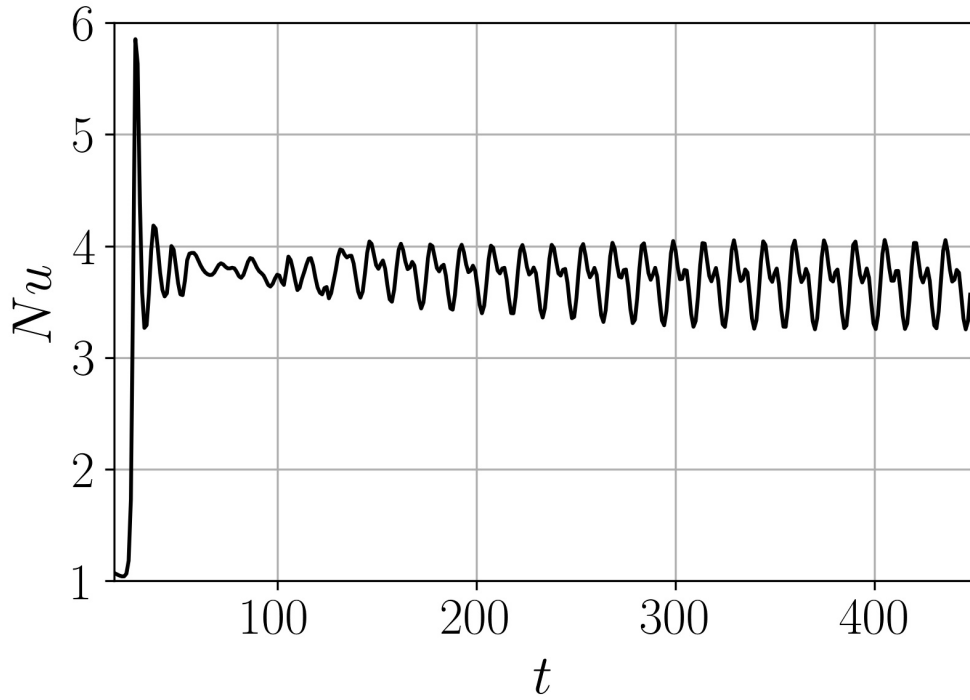


FIGURE 4.9: Time variation of Nusselt number at the outer wall $r = r_o/L_\infty$ of the cavity for $Ra = 10^5$ and $Ro = 0.25$. Refer to Appendix F for more details.

4.2.3 Time averaged statistics

Figure 4.10 shows the temperature profile averaged over time, the azimuthal and the axial directions for different Ra number. The bulk of the cavity has almost isothermal conditions with large gradients near the boundaries. The temperature profiles are not symmetric about the mean of the two wall temperatures. One reason for this is the inherent asymmetry in the geometry due to the inner radius being smaller than the outer radius ($r_i/r_o = 0.52$). The small radius ratio leads to a large difference in the centrifugal force ($\Omega^2 r$) from the inner radius to the outer radius, which results in temperature profiles being asymmetric irrespective of other physical conditions. Another reason is the difference in the value of $\Delta T/T_\infty$ for different cases.

Figure 4.10 shows that the bulk temperature profiles for $Ra = 10^5$ and 10^7 are very close to each other as they have almost the same value of $\Delta T/T_\infty$. However, the case with $Ra = 10^6$ has a maximum value of $\Delta T/T_\infty$, and thus it has a maximum bulk temperature. Conversely, the case with $Ra = 10^8$ has a minimum

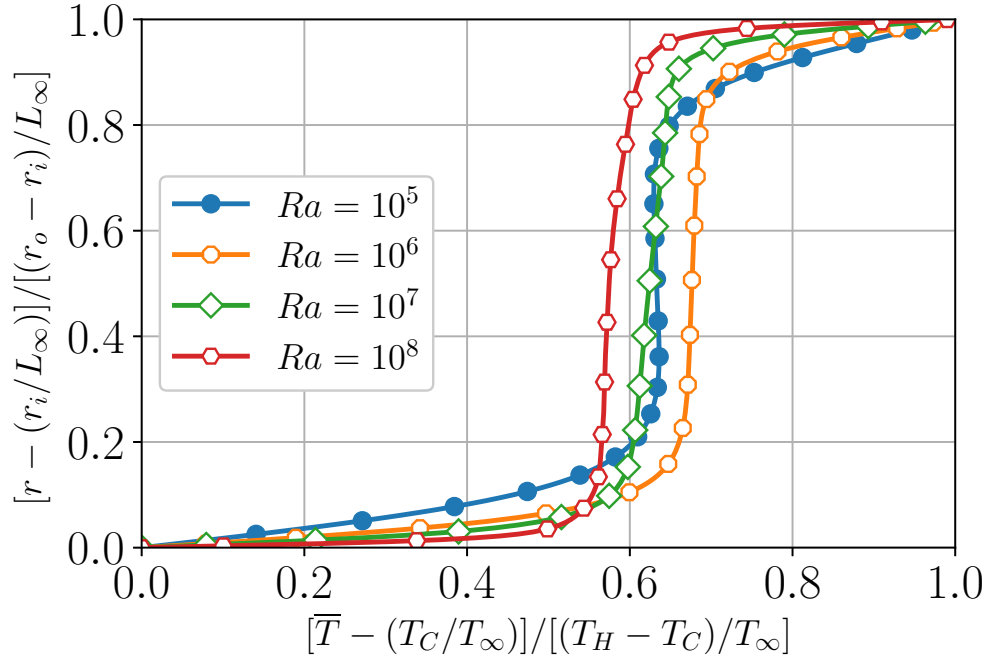


FIGURE 4.10: Comparison of average temperature profiles at different Rayleigh numbers. The temperature profiles are averaged over $(1/4)(H/L_\infty) \leq z/L_\infty \leq (3/4)(H/L_\infty)$ to neglect the effect of insulated walls

bulk temperature. Thus, with the increase in the value of $\Delta T/T_\infty$, the symmetry of the convective flow breaks down. It leads to a rise in the core temperature away from the arithmetic mean of the hot and the cold wall temperature $(T_H + T_C)/2$. The physical reason for this asymmetry is the variation of viscosity with the temperature. The higher viscosity near the outer wall makes the fluid more sluggish as compared to the inner wall and enhances its tendency to mix with the core on its way towards the inner wall. This mixing leads to a rise in the average temperature in the core region, as shown in figure 4.10. This is one of the notable non-Oberbeck-Boussinesq (NOB) effects [63]. This effect is predominant in the case of $Ra = 10^6$, for which $\Delta T/T_\infty$ has the maximum value. To further understand the effects mentioned above, the model of Wu and Libchaber [64] is extended to calculate the ratio of temperature drop in the top thermal boundary layer (ΔT_H) with respect to the temperature drop in the bottom thermal boundary layer (ΔT_C) as

$$\frac{\Delta T_H}{\Delta T_C} = \left(\frac{r_i}{r_o} \right) \left(\frac{\mu_C}{\mu_H} \right)^{1/4} \left(\frac{\rho_C}{\rho_H} \right)^{1/2} \left(\frac{\beta_C}{\beta_H} \right)^{1/4}. \quad (4.6)$$

In equation 4.6, μ , ρ and β are the values of molecular viscosity, density and thermal expansion coefficient, respectively, computed for the respective thermal boundary layers. The ratio r_i/r_o in the equation 4.6 represents the component of geometrical asymmetry. The other terms in the equation 4.6 are functions of temperature difference between the two walls and have different values for different Ra numbers. The comparison of the values of $(\Delta T_H/\Delta T_C)$ predicted by equation 4.6 with respect to the results of the DNS shows good agreement for Ra number up to the order of 10^7 , with maximum error of 10%. This error increases to 20% for $Ra = 10^8$. However, equation 4.6 provides a reasonably accurate prediction and explanation of the asymmetry observed in the temperature profiles. Further, the radial extent of the bulk region increases with Ra numbers and leads to a decrease in the thermal boundary layer thickness.

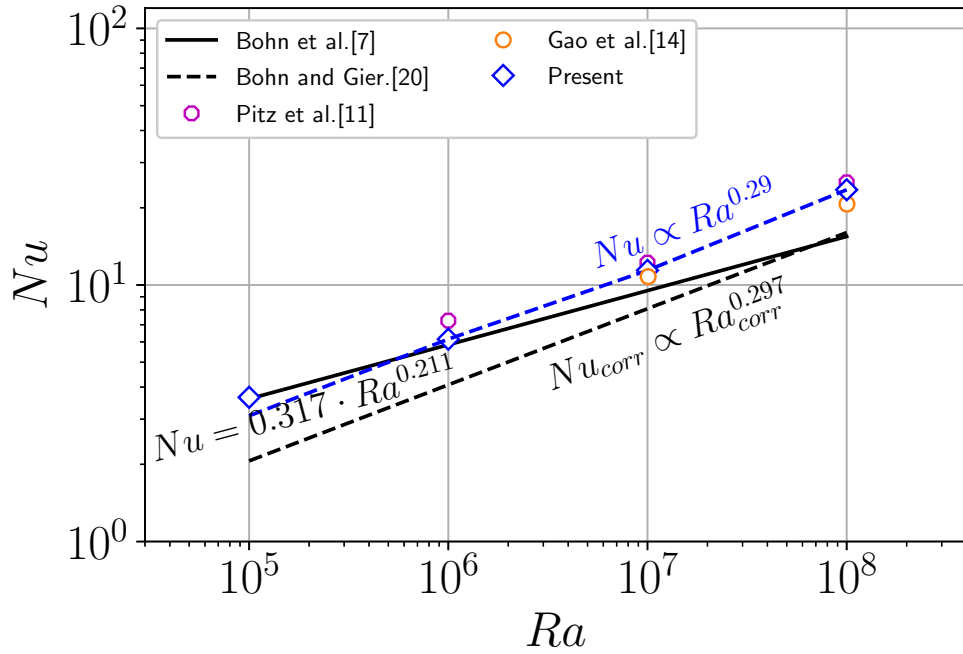


FIGURE 4.11: Comparison of Nusselt number Scaling with Rayleigh number with the scaling reported by Bohn et al.[23], Pitz et al.[27] and Gao et al.[29]. It is important to highlight that the range of Ra numbers considered in the present study as well as in other numerical studies [27] [29] are in the lower end of Bohn et al.[23] experimental conditions and data is sparse in that lower range with some deviations for the $Nu \propto Ra^{0.211}$ scaling law.

The thermal boundary layer thickness also characterizes the heat transfer efficiency defined in the non-dimensional form as Nusselt number. Figure 4.11 presents the scaling of Nu with Ra . The comparison with the experimentally obtained scaling of Bohn et al. [23] shows good agreement for the low Ra number cases. The values of Nu start to deviate from the experiment as we increase the Ra number. In the range $10^6 \leq Ra \leq 10^8$, the power exponent of the Nu – Ra scaling law for the current simulations approaches a value of 0.29. This is consistent with the corrected scaling law for high Ra number cases by Bohn and Gier [24]. Further, the comparison with the results of Gao et al. [29] in figure 4.11, who examined the same cavity configuration by solving the compressible Navier–Stokes equations, also shows a good agreement between both the compressible solvers.

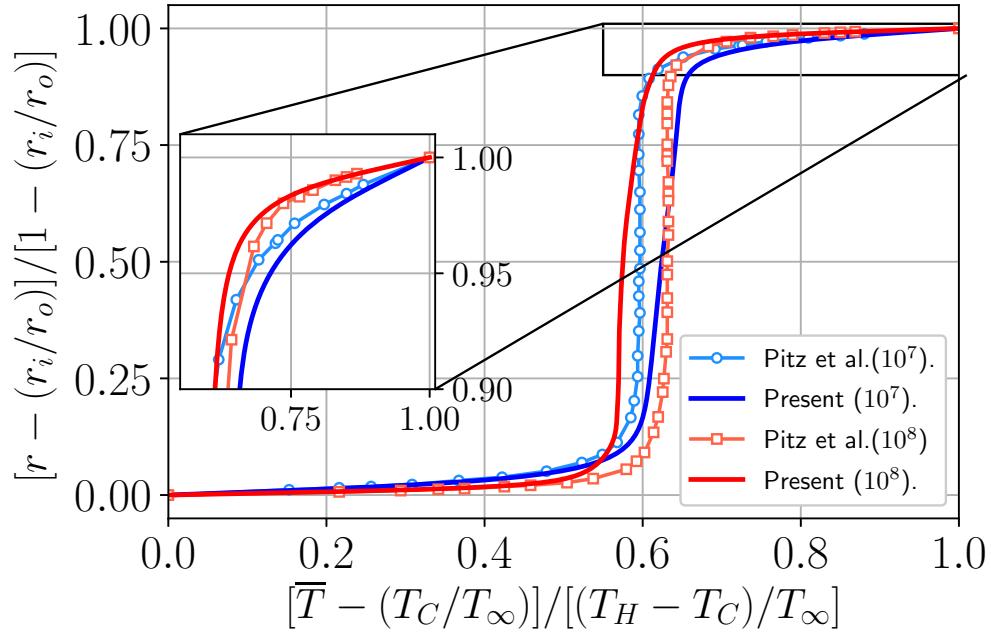


FIGURE 4.12: Comparison of average temperature profiles with the results reported by Pitz et al. [65]. The temperature profiles are averaged over $(1/4)(H/L_\infty) \leq z/L_\infty \leq (3/4)(H/L_\infty)$ to be consistent with [65].

The comparison with the results of the incompressible solver by Pitz et al. [27] shows a close agreement of Nu number values at higher Ra numbers 10^7 and 10^8 . Still, substantial differences in bulk temperature profiles have been observed. Figure 4.12 presents the comparison of the averaged temperature profile with LES results of Pitz et al. [65].

The comparison shows the prediction of the higher temperature gradient in the radial direction for the $Ra = 10^7$ near the outer wall compare to the results of the present study. This can be directly related to the overprediction of Nu at the outer wall as shown in Fig. 4.11.

In the case of $Ra = 10^8$, the near wall temperature profile looks identical, which is consistent with the prediction of Nu as shown in Fig. 4.11. A large deviation is observed in the temperature of the core region of the cavity for both the Ra . Further, as reported by Pitz et al.[65] and shown in Fig. 4.12, the core temperature increases with an increase in the Ra . This trend is in contradiction to the results of the current study as shown in Fig. 4.12, where the core temperature decreases with the increase in the Rayleigh number.

Thus, the discrepancy observed between the values of Nu number at low Ra number could be due to the high value of $\Delta T/T_\infty$, beyond the limit of the Boussinesq approximation used in the incompressible simulations. At such a large temperature difference, the Boussinesq approximation is no longer a valid assumption; the imposed temperature difference must be small enough to cause only negligible density changes [66]. This can be assessed qualitatively by comparing the density distribution in the domain calculated from the same instantaneous temperature field. Figure 4.13 shows the instantaneous density distribution obtained directly from the compressible formulation and calculated by using the algebraic expression of the Boussinesq approximation $[\rho = \rho_\infty(1 - (T - 1))]$, applied to the same instantaneous temperature field of the compressible DNS. The results show that the density contours obtained from compressible calculations as shown in Fig. 4.13a and 4.13c for the $Ra = 10^6$ and 10^7 , respectively, are quite different from the ones obtained from the linear approximation of density shown in Fig. 4.13b and 4.13d for the same Ra . As observed from the instantaneous density contours of the compressible formulation, most of the core region of the cavity is filled with the less dense or hot fluid from the outer wall with the narrow plumes of high-density fluid rising from the inner wall. However, in the case of the Boussinesq approximation, enhanced mixing can be observed in the core of the cavity with wider convection cells filling the region between the hot and the cold walls, which augment the

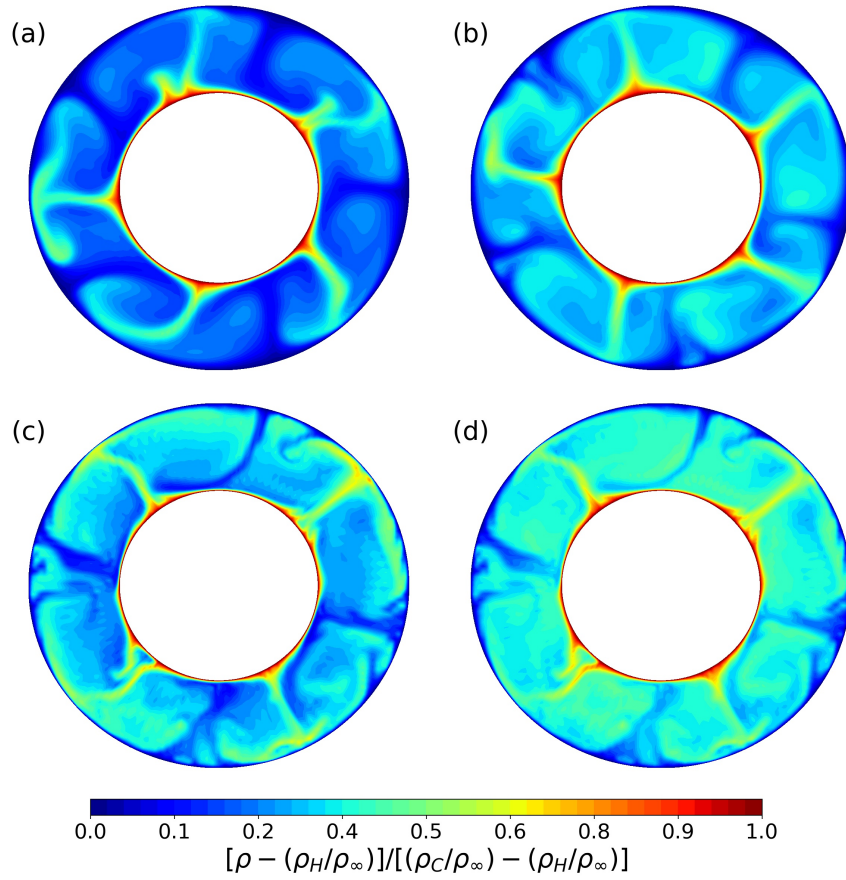


FIGURE 4.13: Instantaneous density contours at mid-axial position $z = (1/2)(H/r_o)$ of the cavity for (a) $Ra = 10^6$ obtained compressible code results, (b) $Ra = 10^6$ from linear approximation, (c) $Ra = 10^7$ obtained compressible code results and (d) $Ra = 10^7$ from linear approximation.

overall heat transfer rate between the outer and the inner walls of the cavity. This could be the possible reason for the overprediction of the Nusselt number in the case of incompressible calculations with the Boussinesq approximation.

In conclusion, the power exponent for the $Nu-Ra$ scaling law obtained from the current DNS agrees well with the corrected-scaling law from experiments[24]. Thus the reduced power exponent measured by Bohn et al. [23] in the experiments can be attributed to the heat loss from the side disks. Further, the comparison with results of an incompressible solver shows that the reason for the disagreement between the two approaches could be the use of Boussinesq approximation, which considers the density as a linear function of temperature for small temperature differences.

The next section is devoted to understanding the effect of azimuthal domain size of the annulus cavity.

4.3 Effect of azimuthal domain size

The basic objective of this analysis as already stated above is to devise a guideline to reduce the computational cost for high Ra number cases by simulating a sector instead of the full 2π domain. This idea was initially proposed by Sun et al.[25] who followed this approach by running a $2\pi/8$ degree sector for high Ra number cases and observed some differences in the flow structure as compared to a full 2π domain and unexpectedly no effect on the wall heat transfer rate. However, it is very important to scrutinize the small differences in the flow that arise due to different azimuthal domain sizes, as their impact will get amplified at high Rayleigh numbers and might lead to the wrong prediction of wall heat transfer rate and other flow quantities.

Ra	Azimuthal domain size	Nu	$N_r \times N_z \times N_\theta$
10^6	$2\pi/12$	6.1	$160 \times 160 \times 34$
	$2\pi/6$	5.54	$160 \times 160 \times 34$
	$2\pi/5$	6.15	$160 \times 160 \times 34$
	2π	6.15	$160 \times 160 \times 194$
10^8	$2\pi/12$	23.09	$432 \times 432 \times 34$
	$2\pi/6$	26.07	$432 \times 432 \times 66$
	2π	25.50	$432 \times 432 \times 258$

TABLE 4.4: DNS cases of different sector sizes for $Ra = 10^6$ and 10^8 .

To that end, additional simulations were carried out at $Ra = 10^6$ and 10^8 for several different azimuthal domain sizes, as shown in table 4.4. Figure 4.14 shows the instantaneous temperature contours in the $r - z$ plane at the mid-axial position of the cavity for $Ra = 10^6$. Qualitative differences in the flow structure can clearly be observed in the different sectors as compared to the full 2π domain. These differences translate to variations in the mean temperature profile, as shown in figure 4.15. The mean temperature profiles for cases with azimuthal extent $2\pi/5$ and 2π

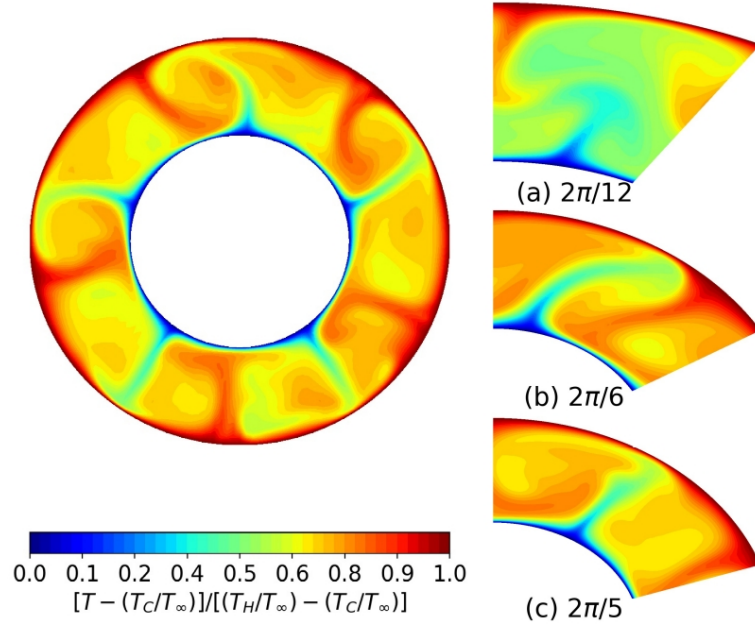


FIGURE 4.14: Comparison of instantaneous temperature contours at mid-axial position $z = (1/2)(H/r_o)$ of the cavity at $Ra = 10^6$ for different azimuthal domain size of (a) $2\pi/12$, (b) $2\pi/6$, (c) $2\pi/5$ with the reference domain size of 2π .

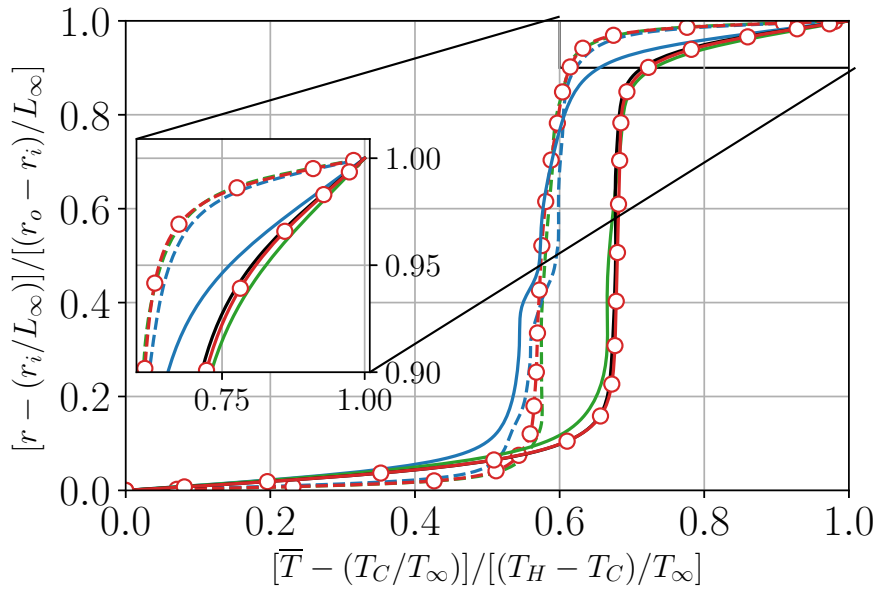


FIGURE 4.15: Comparison of average temperature profiles for $Ra = 10^6$ (—) and $Ra = 10^8$ (---) for different azimuthal domain sizes of 2π (red), $2\pi/5$ (black), $2\pi/6$ (green) and $2\pi/12$ (blue).

are in perfect agreement with each other for $Ra = 10^6$, while the other sector size shows different behaviour. Similarly, the mean temperature profile for sector size $2\pi/6$ agrees well with the profile of the full 2π domain size for $Ra = 10^8$ as shown in figure 4.15. A similar agreement can be observed in the value of Nu number for the $Ra = 10^6$ case, but with 2% overprediction at $Ra = 10^8$ for the sector size of $2\pi/6$, as shown in table 4.4. Thus it can be concluded that for $Ra = 10^6$ and 10^8 , the sector domain size of $2\pi/5$ and $2\pi/6$ are the required azimuthal domain sizes to capture the full physics with reasonable accuracy. The physical reason behind this perfect agreement can be deduced from the wavenumber spectra shown in figure 4.4(a). For $Ra = 10^6$, the wavenumber $k = 5$ is the most energetic mode and associated with a pair of convection rollers observed in the flow. The domain size of $2\pi/5$ supports $k = 5$ and thus can represent the complete flow physics of the problem. Similarly, the domain size of $2\pi/6$ supports the dominant mode, $k = 6$, for the $Ra = 10^8$ case. The reduced domain sizes have 82.5% and 74.4% fewer grid points compared to the full 2π size for $Ra = 10^6$ and 10^8 , respectively. Thus, a significant reduction in the computational cost can be achieved. Hence, in order to accurately capture the flow physics at reduced computational cost, one needs to include the most energetic wavenumber for a particular Ra number in the reduced azimuthal domain size. This dominant wavenumber can be obtained from relatively inexpensive precursor stability calculations.

4.4 Conclusions

In this study, numerical simulations of centrifugal buoyancy-induced flow in a closed rotating cavity for Ra numbers in the range of 10^5 to 10^8 were conducted, matching the experimental conditions of Bohn et al. [23]. The results of the present study showed a well-defined flow structure in the domain with counter-rotating vortices. A different number of convection rollers was observed in the domain for the different Ra numbers. The time evolution of flow for $Ra = 10^5$ showed that the number of convection rollers varied with time. The combination of geometrical parameters and NOB effects led to asymmetric temperature profiles in

the cavity. In the range $10^6 \leq Ra \leq 10^8$, the power exponent of the Nu – Ra scaling law approaches a value of 0.29, in agreement with the corrected experimental exponent [24].

A comparison of the current results with the predictions of an incompressible solver using the Boussinesq approximation shows that the latter overpredicts of the Nu number. Further, the comparison of the averaged temperature profiles show a deviation near the wall as well as in the core region of the cavity. The cause of this difference could be due to the Boussinesq approximation which considers the density as a linear function of temperature. Qualitative comparison of density contours obtained from the results of the compressible code with the ones obtained from the Boussinesq approximation is reported. More uniform prediction of density by using the Boussinesq approximation could be a reason behind the overprediction of the Nu in the latter case.

Chapter 5

Effects of compressibility in centrifugal buoyancy-induced flow

In chapter 4, the results were compared with the experimental data of [23] to understand the $Nu-Ra$ scaling. The maximum rotational speed achieved in the experiments by Bohn et al. [23] was approximately one-half of engine representative conditions [49]. At higher rotational speeds, compressibility effects, especially in the core region of the cavity, become increasingly important, affecting the heat transfer rate. In this chapter, the initial discussions are centred on the results from direct numerical simulations to study the effects of compressibility, followed by linear-stability analysis to provide further insights.

The contents of this chapter are from the article published in the International Journal of Heat and Fluid Flow (Saini and Sandberg. [58]) and the conference proceedings of Australasian Fluid Mechanics Conference 2020 (Saini and Sandberg [67])

5.1 Introduction

At high rotational speeds, the effects of compressibility play a significant role and affect the flow structure and heat transfer rate. The experimental evidence for this

was provided by Farthing et al. [32], where a decrease in the value of heat transfer rate with the increase in the rotational speed of the cavity was reported. Owen and Tang [43] also highlighted these effects of compressibility in the core region of the closed cavity configuration and sought more experimental or computational evidence to prove or disprove their predictions. The objective here is to simulate the same phenomenon by varying the Ma number for the case of $Ra = 10^5$.

5.2 Direct Numerical Simulation Results

The numerical setup and boundary conditions for this case can be found in chapter 4. To isolate the effect of Mach number, it is varied in the range $0.05 < Ma_\infty < 0.2$ for $Ra = 10^5$ and a fixed Rossby number of $Ro = 0.25$. A constant value of Rossby number implies a constant temperature difference between the hot and cold walls, as per the definition of Ro number given in chapter 4. The range of Ma_∞ number is based on the experimental conditions of Atkins and Kanjirakkad [49] for a cavity with throughflow replicating real engine conditions.

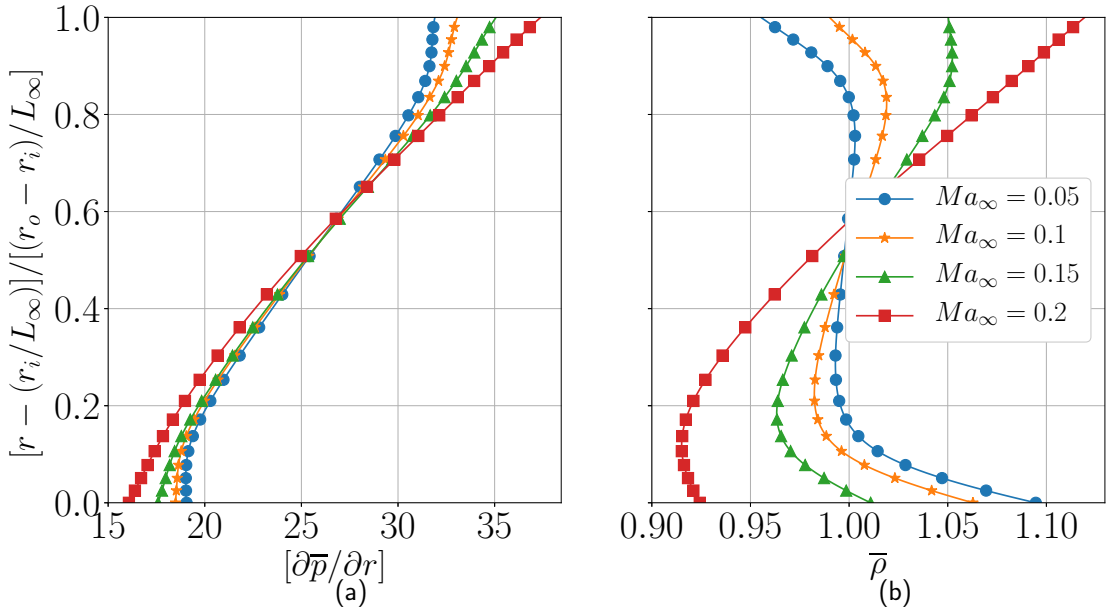


FIGURE 5.1: Comparison of average (a) radial pressure gradient and (b) density profiles at different Mach numbers for $Ra = 10^5$ and $Ro = 0.25$. The profiles are averaged over $(1/4)(H/L_\infty) \leq z/L_\infty \leq (3/4)(H/L_\infty)$ to neglect the effect of insulated walls.

Figure 5.1(a) shows the averaged radial pressure gradient profiles at different Ma_∞ number. The pressure gradient profile for $Ma_\infty = 0.05$ has a quasi-constant value near the outer and inner walls of the cavity and increases linearly with radius in the core. However, as the Ma_∞ number increases, the pressure gradient profile near the walls starts moving away from a constant value to linear variation with the radius as shown in figure 5.1(a). Thus, at higher Ma_∞ number, the pressure gradient increases near the hot outer wall and decreases near the cold inner wall more rapidly with the radius as compared to the low Ma_∞ number cases. That Ma_∞ number affects the pressure gradient in the core region of the flow can further be supported by assuming the expansion and contraction of fluid parcels in the centrifugal force field to be an adiabatic process. It implies that the conduction heat transfer between the fluid parcels is negligible, which is a valid assumption for air as it has low thermal conductivity. Thus, by using the pressure balance equation with the centrifugal force and the first law of thermodynamics, the closed-form equation for the pressure gradient in the core is shown in equation 5.1.

$$\frac{\partial p}{\partial r} = \frac{\gamma \overline{p_{c1}} Ma_\infty^2 r}{4 Ro^2 \overline{T_{c1}}} \left[1 + \frac{\gamma - 1}{8 Ro^2 \overline{T_{c1}}} Ma_\infty^2 (r^2 - r_{c1}^2) \right]. \quad (5.1)$$

In the above equation, $\overline{p_{c1}}$ and $\overline{T_{c1}}$ are the time, axially and azimuthally averaged values of pressure, temperature at a fixed radius r_{c1} , respectively, at which the core region of the convection starts from the inner wall. The comparison of pressure gradient at two different Ma_∞ numbers with the predictions using equation 5.1 is shown in figure 5.2. The agreement of pressure gradient with the above-derived equation confirms that the increase in the Ma_∞ number leads to adiabatic expansion and compression of fluid layers in the core. It further causes local expansion and compression near the inner cold and the hot outer wall, respectively, counter-acting the effects of temperature at the respective walls.

The effect of this pressure gradient on the density profiles can be seen in figure 5.1(b). The averaged density for $Ma_\infty = 0.05$ shows the natural variation observed in the buoyancy-induced flow; high-density fluid near the cold bottom wall and low-density fluid near the hot outer wall with quasi-constant density in the core

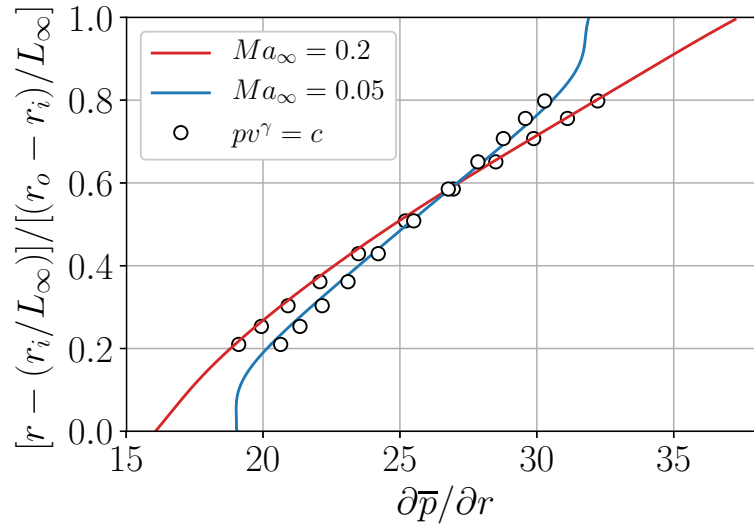


FIGURE 5.2: The comparison of average radial pressure gradient for $Ra = 10^5$ at Ma number of 0.2 and 0.05 with the profiles for the same prediction by equation 5.1.

region. However, as Ma_∞ number increases, the density at the outer wall starts to increase while decreasing at the inner wall due to the formation of high and low-pressure regions near the respective walls. At higher Ma_∞ number, the effect of strong pressure gradient and consequential local compression and expansion leads to a density distribution that is reminiscent of a stratified flow. This stratification leads to suppression of the instabilities that normally contribute to the formation of the convection rollers. This situation can be perceived from figures 5.3 and 5.4, which shows the instantaneous density and temperature contours for different Ma_∞ numbers, respectively.

Figure 5.3(a) shows the instantaneous density contour for $Ma_\infty = 0.05$, which is the baseline case to compare. As we increase the Ma_∞ number to 0.1, inside the convection rollers small regions of compressed or high-density fluid near the hot wall of the cavity can be observed. The conditions at $Ma_\infty = 0.15$ lie somewhere in between the non-stratified density profile at $Ma_\infty = 0.1$ and the strongly stratified conditions of density at $Ma_\infty = 0.2$. As it can be observed from the temperature contours in figure 5.4(c) at $Ma_\infty = 0.15$, the weak convection currents still exist near the inner wall. These convection cells are suppressed by the small pockets of low-density fluid near the inner wall and of high-density fluid near the outer wall,

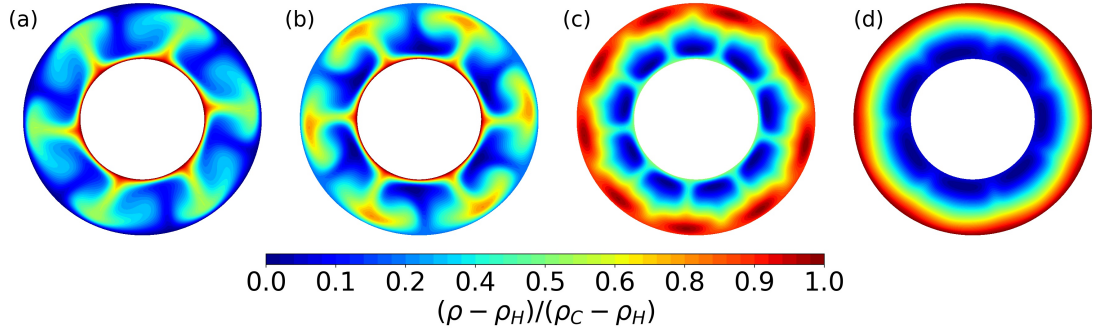


FIGURE 5.3: Instantaneous density contours at mid-axial position $z = (1/2)(H/L_\infty)$ of the cavity for $Ra = 10^5$, $Ro = 0.25$ and (a) $Ma_\infty = 0.05$, (b) $Ma_\infty = 0.1$, (c) $Ma_\infty = 0.15$, (d) $Ma_\infty = 0.2$.

as shown in figure 5.3(c). At $Ma_\infty = 0.2$, the condition is completely stratified due to the strong radial pressure gradient in the system.

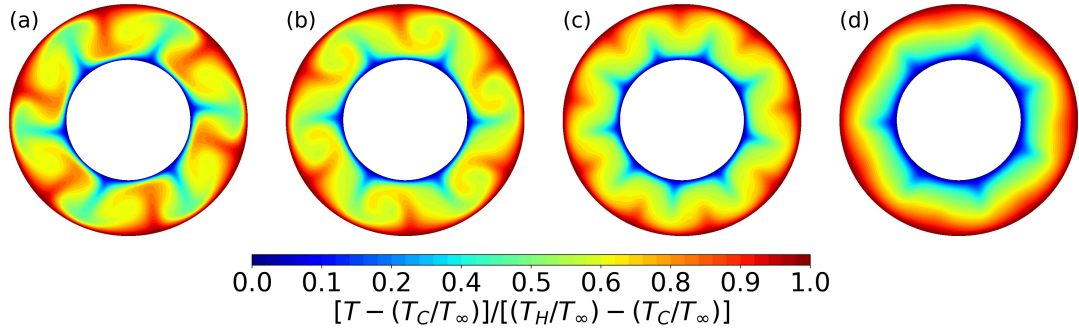


FIGURE 5.4: Instantaneous temperature contours at mid-axial position $z = (1/2)(H/L_\infty)$ of the cavity for $Ra = 10^5$, $Ro = 0.25$ and (a) $Ma = 0.05$, (b) $Ma = 0.1$, (c) $Ma = 0.15$, (d) $Ma = 0.2$.

Figure 5.5 shows the average temperature profiles at different Ma_∞ numbers. Here also we will consider the profile for $Ma_\infty = 0.05$ as a baseline for the comparison. In any buoyancy-driven flow, the maximum temperature gradient will always be observed in the thermal boundary layers, and any gradient in the core region signifies inefficient mixing and resistance to heat transfer. As we increase the Ma_∞ number, the boundary layer thickness increases at both the walls and the core temperature also starts to deviate from the isothermal conditions. Thus, the overall wall heat transfer rate reduces with the increase in the Ma_∞ number. At the maximum $Ma_\infty = 0.2$, a nearly linear temperature profile is observed, which

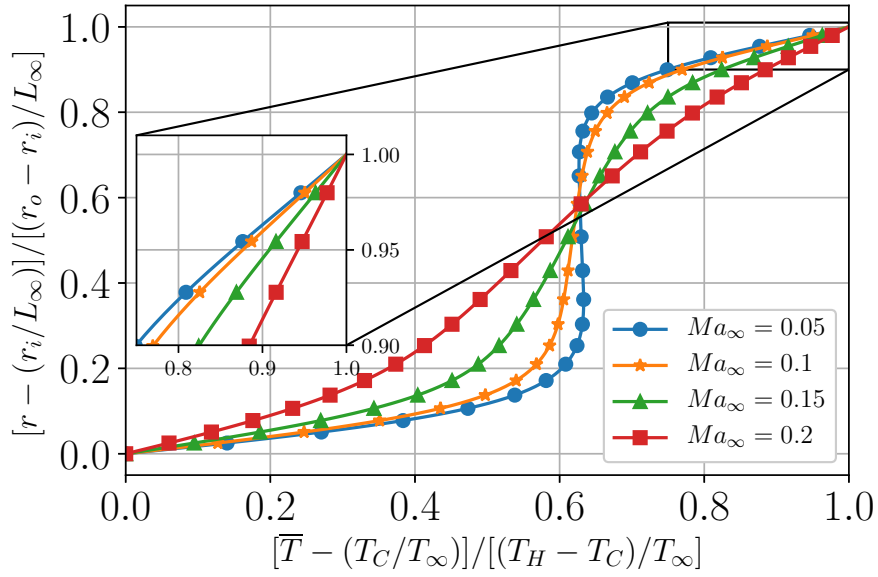


FIGURE 5.5: Comparison of average temperature profiles at different Mach numbers for $Ra = 10^5$ and $Ro = 0.25$. The temperature profiles are averaged over $(1/4)(H/L_\infty) \leq z/L_\infty \leq (3/4)(H/L_\infty)$ to neglect the effect of insulated walls

means conduction is the dominant mode of heat transfer with negligible convection occurring due to the stratification.

To further quantify the effectiveness of the wall heat transfer rate, the value of Nu number is also compared for the different Ma_∞ numbers in figure 5.6. The value of Nu reduces with the increase in Ma_∞ number and approaches to one at $Ma_\infty = 0.2$. This effect of the Ma_∞ number, although analyzed here only for the lowest Ra number, provides strong evidence for compressibility playing a considerable role in reducing the heat transfer rate. Thus, the compressibility leads to a heat transfer reduction by stabilizing the flow and suppressing the hydrodynamic instability leading to the formation of convection rollers. To elaborate on that, in the next section, we will investigate the effects of compressibility on the linear growth of flow instability in the closed cavity.

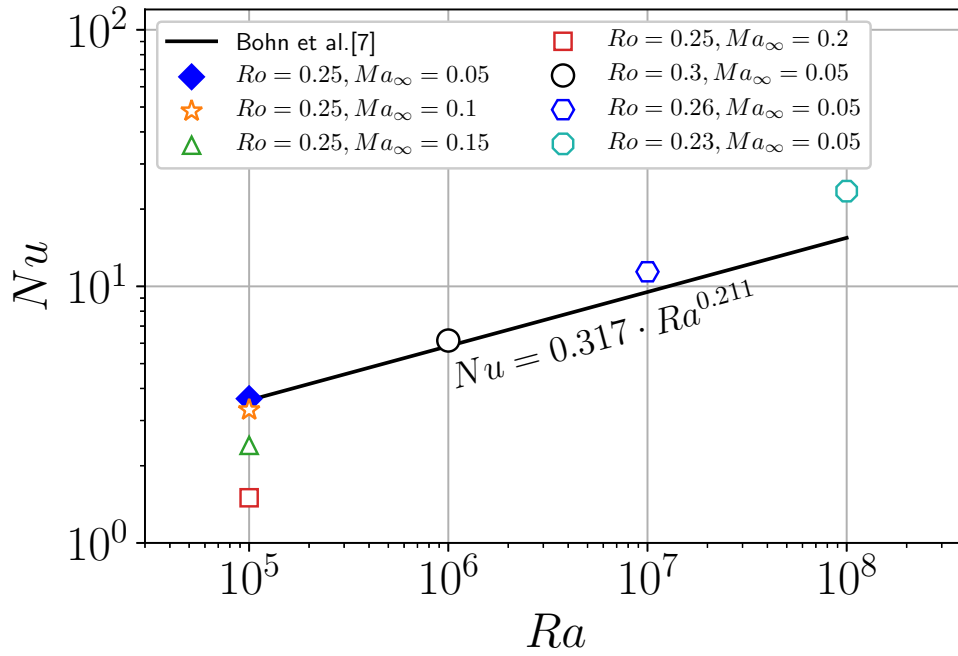


FIGURE 5.6: The variation of Nusselt number with Mach number for $Ra = 10^5$ and comparisons for Nu – Ra scaling for other Rayleigh numbers with the scaling reported by Bohn et al.[23], Pitz et al.[27].

5.3 Linear stability analysis

Linear stability theory revolves around understanding the behaviour of small disturbances in the flow. In other words, the objective is to estimate the minimum value of a critical parameter beyond which a disturbance of infinitesimally small amplitude in the initial conditions grows exponentially. The experiments for the centrifugal buoyancy-induced flows carried out by Busse and Carrigan [68] studied the onset of convection in a rotating annulus with a heated outer wall and a cold inner wall. They proposed an analytical expression based on the linear theory for the critical value of the buoyancy parameter as a function of the Ekman number and the Prandtl number. The agreement between the experimental observations and the analytical results suggested that linear theory is able to describe such types of flows. A recent numerical study for centrifugal buoyancy-induced flows in a closed cavity was carried out by Pitz et al.[62] for the onset of convection. They performed linear stability analysis based on the incompressible Navier–Stokes equations to obtain the critical Rayleigh number at different radius ratios. In the

asymptotic limit of radius ratio approaching unity, the critical Rayleigh number was found to be 1,708. This value also corresponds to the critical value for natural convection under gravity. The critical wavenumber number obtained from the linear stability analysis was observed to be dominant in the turbulent regime as well.

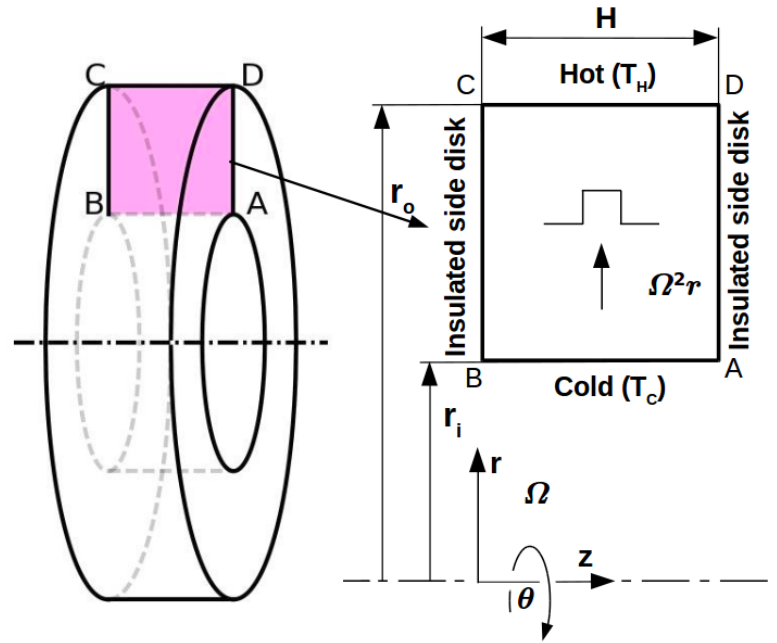


FIGURE 5.7: Schematic of closed annular cavity. The walls of the cavity undergo solid-body rotation about the z -axis.

Another study that carried out linear stability analysis by solving incompressible equations with the Boussinesq approximation is of Kang et al.[69]. They included an additional Coriolis buoyancy term in their equations and investigated the annular cavity problem for a Rayleigh number of the order of 10^5 . A decrease in the value of the critical Rayleigh number with the increase in the radius ratio was observed. They too found that the critical Rayleigh number approached the value of 1,708 in the limit of radius ratio approaching unity. A complex transition phenomenon from the onset of the convection to the chaotic motion was reported. The studies mentioned above were limited to the physical conditions where the Boussinesq approximation was valid and thus considered Rayleigh number as the only critical parameter. However, at high rotational speeds, the effects of compressibility play a significant role and affect the flow structure and heat transfer

rate. As discussed in section 5.2 the radial density distribution approaches the stratified conditions with an increase in the flow Mach number, and this prevents the convection rollers from growing.

Thus the objective of this section is to investigate the effects of compressibility on the linear growth of flow instability in the centrifugal buoyancy-induced flow by solving the compressible Navier–Stokes equations.

5.3.1 Multi-dimensional stability analysis

The multi-dimensional stability analyses are carried out by using a forced Navier–Stokes simulation approach as discussed in chapter 3. A base state of solid body rotation of the cavity with conduction heat transfer between the outer and the inner wall is considered given as

$$\begin{aligned} \rho^0 &= 1.0, \quad \rho^0 u_1^0 = 0.0, \quad \rho^0 u_2^0 = 0.0, \quad \rho^0 u_3^0 = \Omega r, \\ \rho^0 E^0 &= \frac{\rho^0 T^0}{\gamma(\gamma - 1)Ma_\infty^2} + (1/2)\rho^0 u_i^0 u_i^0, \end{aligned} \tag{5.2}$$

where $T^0 = (T_H - T_C)[\log(r/r_o)/\log(r_o/r_i)] + T_H$ is the heat conduction temperature profile. For the stability analysis, all the Fourier modes are perturbed by a spatial step function in the r – z plane of amplitude 10^{-6} at the center of the cavity, as shown in figure 5.7. The perturbation is introduced at the initial state and then its development is monitored as the simulation progresses. If the flow conditions are unstable then the initial perturbation would grow exponentially until it reaches a non-linear state. In contrast, for the stable flow conditions the response of an initial perturbation subsides with time until it disappears. This approach has been successfully used in previous studies to investigate the linear stability of flow over an aerofoil [70] and for axisymmetric wakes [71].

5.3.2 Results and Discussion

The linear stability analyses for two Ra numbers of 10^5 and 10^7 for the physical conditions given in chapter 4 have been carried out to understand the onset of convection. To further study the compressibility effects, the Ma number is varied in the range 0.01–0.2 for $Ra = 10^5$ and 0.05–0.2 for $Ra = 10^7$, respectively. The grid resolution in the $r - z$ directions is kept the same as for the DNS that were discussed in chapter 4. In the azimuthal direction, 32 Fourier modes are used for both the Ra numbers.

Figure 5.8 shows the time evolution of the instability from the DNS for $Ra = 10^5$ and $Ra = 10^7$ cases. The initial state of heat conduction in the DNS is perturbed by adding sinusoidal perturbations of the form $\sin(k\theta)$ to the temperature in the azimuthal direction. From the instantaneous flow field, the initial conduction state of the system can be observed in figure 5.8(a). As the flow evolves in time, the local disturbance added to the conduction state at the interface of hot and cold fluid can be seen (figure 5.8(b)). The instability in the flow, which is at a fetal stage at this time, further grows temporally and spatially to form “mushroom” shaped structures as shown in the figure 5.8(c). The plumes of the high-density fluid (low temperature) rise in an asymmetric manner in the radial direction (figure 5.8(c)). At the fully grown stage (figure 5.8(d)), there are eight plumes in the domain. Similar behaviour for the time evolution of the instability can be observed in the case of $Ra = 10^7$ in figures 5.8(f)–(j). However, at higher Ra number, the flow transitions to turbulence more rapidly and a more chaotic flow field emerges. At the fully developed state, the number of convection rollers is six for $Ra = 10^5$ and five for $Ra = 10^7$, which also corresponds to the peak energy level in the time-averaged wavenumber spectra at $k = 6$ and $k = 5$, respectively, as shown in figure 5.8(k). The multiple peaks are observed in the wavenumber spectra for the $Ra = 10^5$ case correspond to the harmonics of the dominant wavenumber, which can be distinctly observed in the spectra as the flow is not fully turbulent.

To understand this transition from the initial conduction state to an unsteady flow field, the linear stability analyses are conducted for the same conditions.

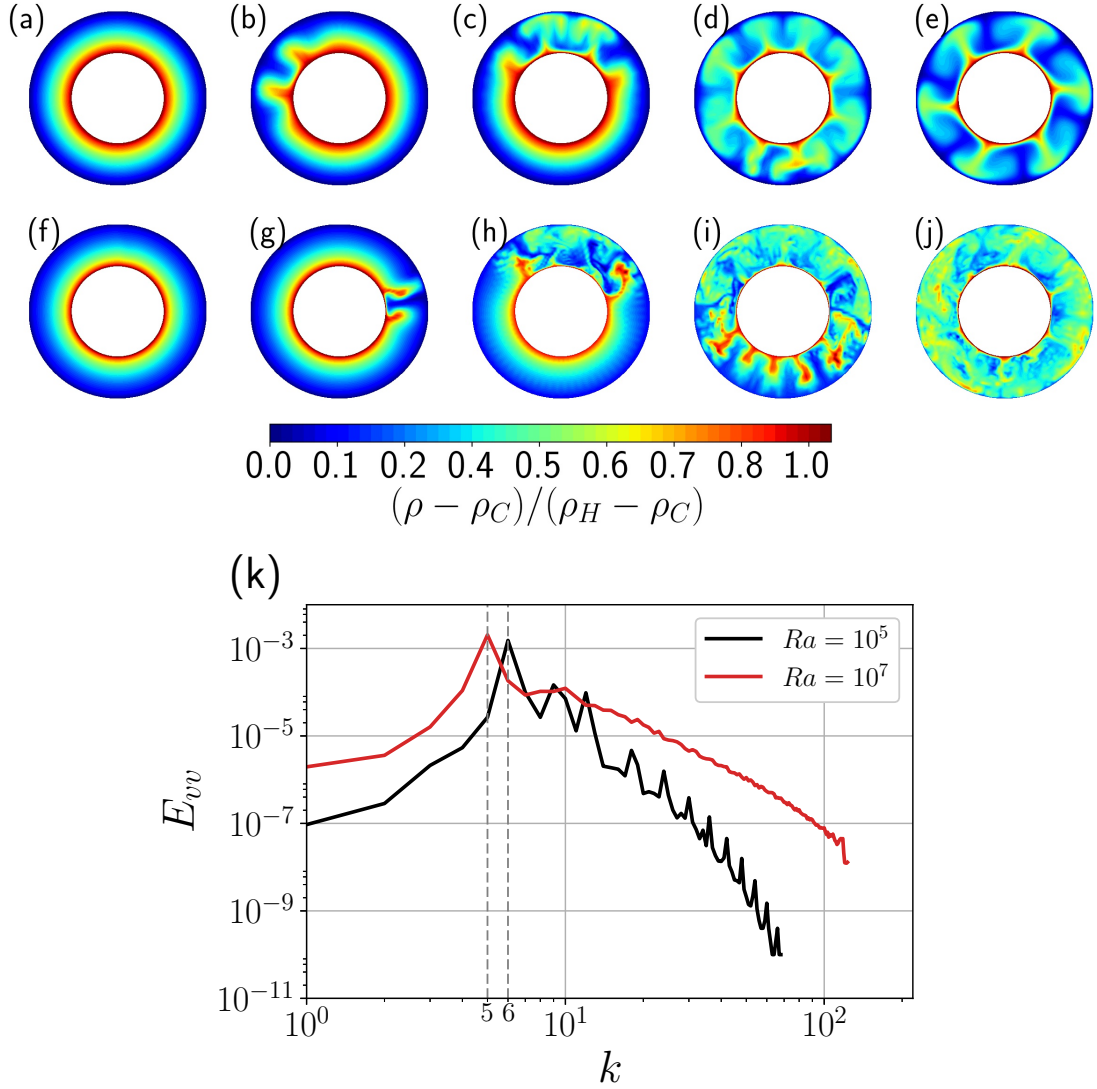


FIGURE 5.8: Time evolution of density at the mid-axial position $z = (1/2)(H/L_\infty)$ of the cavity for $Ra = 10^5$, $Ro = 0.199$ and $Ma = 0.01$ at non-dimensional time units (a-e) $t = 10.0, 22.0, 25.0, 40.0$ and 55.0 , respectively, and for $Ra = 10^7$, $Ro = 0.199$ and $Ma = 0.05$ at non-dimensional time units (f-j) $t = 12.5, 18.0, 25.0, 32.5$ and 38.0 , respectively. (k) Wavenumber spectra of the monitor point located at the middle of the cavity at different Rayleigh numbers. All results from DNS.

All the modes are perturbed with an initial pulse of amplitude 10^{-6} at the r - z location of $(1.586, 0.5217)$. The behaviour of all the azimuthal Fourier modes is then monitored. Figure 5.9(a) shows the temporal development of the first sixteen most-amplified azimuthal modes. The wavenumbers $k = 8, 9$ and 10 are the most unstable and have the highest growth rates. Further, as observed in figure 5.9(a), multiple modes get amplified in the linear regime, and the superposition of these modes leads to an asymmetric or localized growth of the instability. The localized

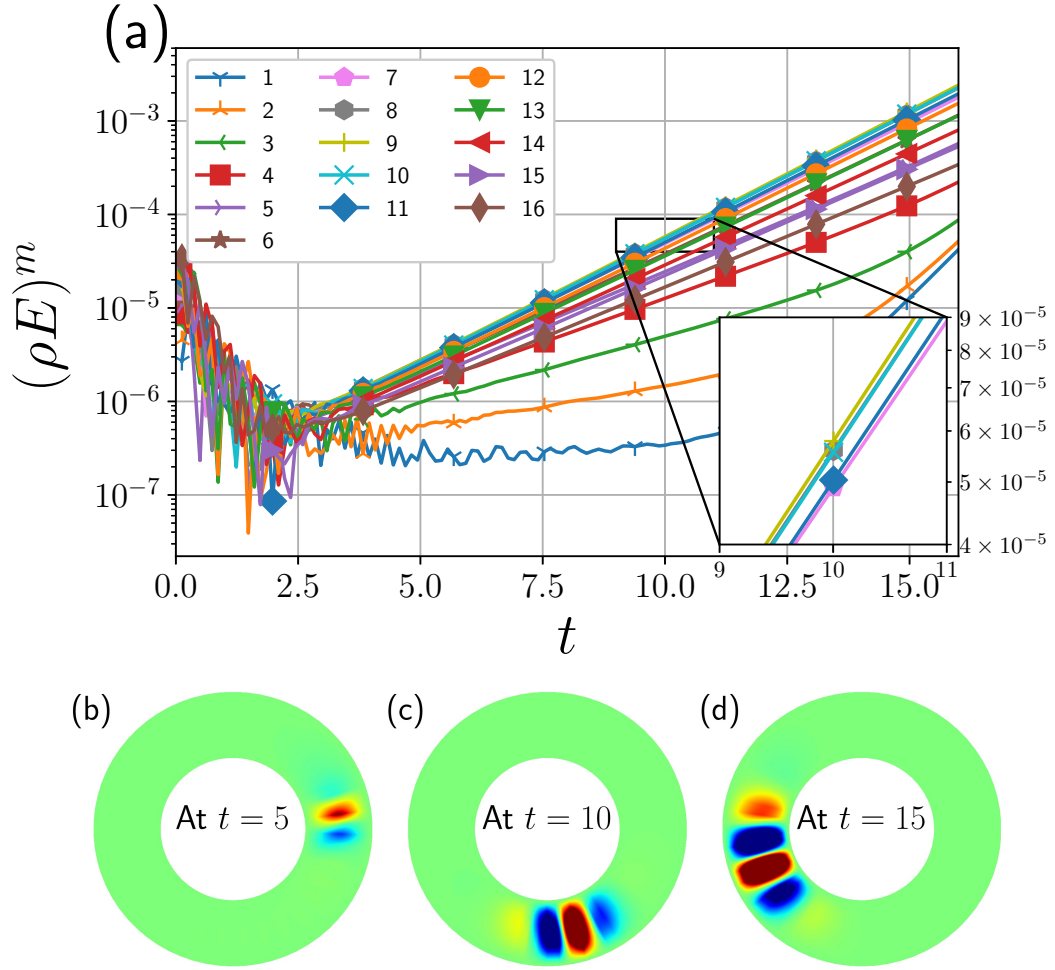


FIGURE 5.9: (a) Temporal development of first 16 most amplified Fourier modes in the LSA at $(z, r) = (0.527, 1.2)$ for $Ra = 10^5$ at $Ma = 0.01$ and $Ro = 0.199$. Instantaneous contour for radial velocity fluctuations at (b) $t = 5$ (contour levels $[-1.34e-05, +1.34e-05]$), (c) $t = 10$ (contour levels $[-2.34e-05, +2.34e-05]$) and (d) $t = 15$ (contour levels $[-9.34e-05, +9.34e-05]$).

spatial growth of the instability can be observed in the figures 5.9(b)-(d), which show the instantaneous contours of radial velocity fluctuations at different time instants. Similar behaviour is observed for the $Ra = 10^7$ case in figure 5.10. In this case, the wavenumbers $k = 15, 16$ and 17 are the most linearly amplified. In the linear regime, there are multiple unstable modes, and each one has a different growth rate. The superposition of these unstable modes leads to the asymmetric growth of perturbation in the transition phase of the flows, as shown in figure 5.8. Also, the most unstable wavenumbers in the linear regime have a much shorter wavelength than the one that eventually appears in the fully developed state in the DNS.

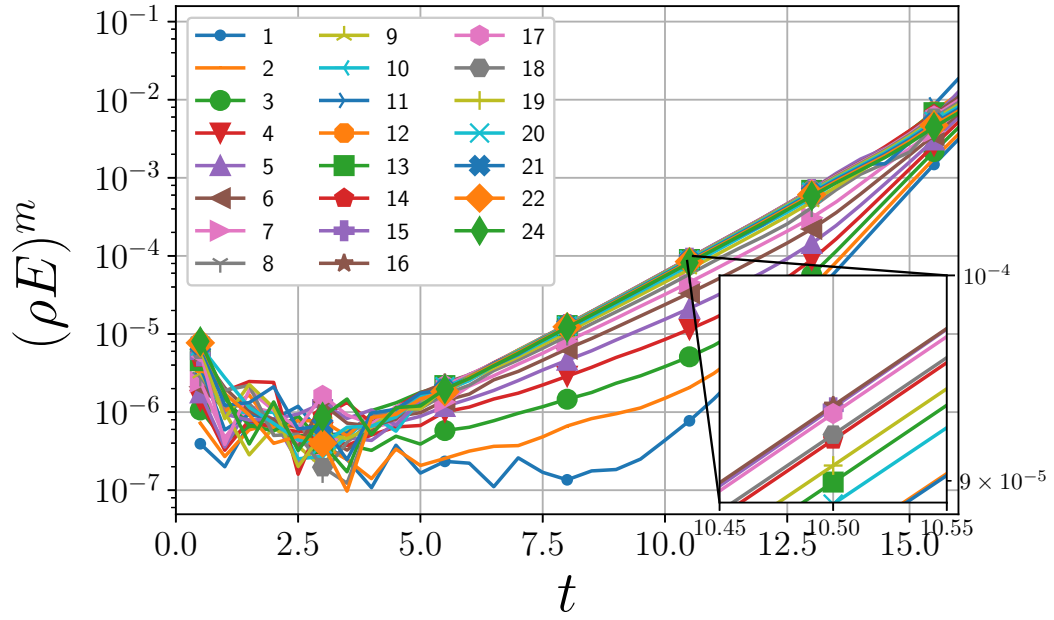


FIGURE 5.10: Temporal development of first 24 most amplified Fourier modes in the LSA at $(z, r) = (0.527, 1.2)$ for $Ra = 10^7$ at $Ma = 0.05$ and $Ro = 0.199$.

This implies that non-linear interaction between these linearly unstable modes occurs which could eventually lead to amplification of the lower wavenumber in the saturation state.

5.3.3 Effect of compressibility

To study the effect of compressibility on the growth of the flow instability, the Ma_∞ number is varied at a constant value of $Ro = 0.199$ for both the Ra numbers. Figure 5.11 shows the time evolution of the most unstable mode for the $Ra = 10^5$ and 10^7 cases for different Ma numbers. The growth rate of the most unstable mode reduces with the increase in the value of Ma number and the flow becomes linearly stable at $Ma = 0.2$ for both the Ra numbers. The growth rate of the most unstable mode is further plotted for both the Ra numbers in figure 5.12. At Ma number of 0.2, the growth rate of all the modes becomes negative for both the Ra numbers. Thus, the compressibility effects lead to the suppression of instability which normally leads to the formation of convection rollers in the domain. To further understand the stabilizing effect of Ma number, a stability

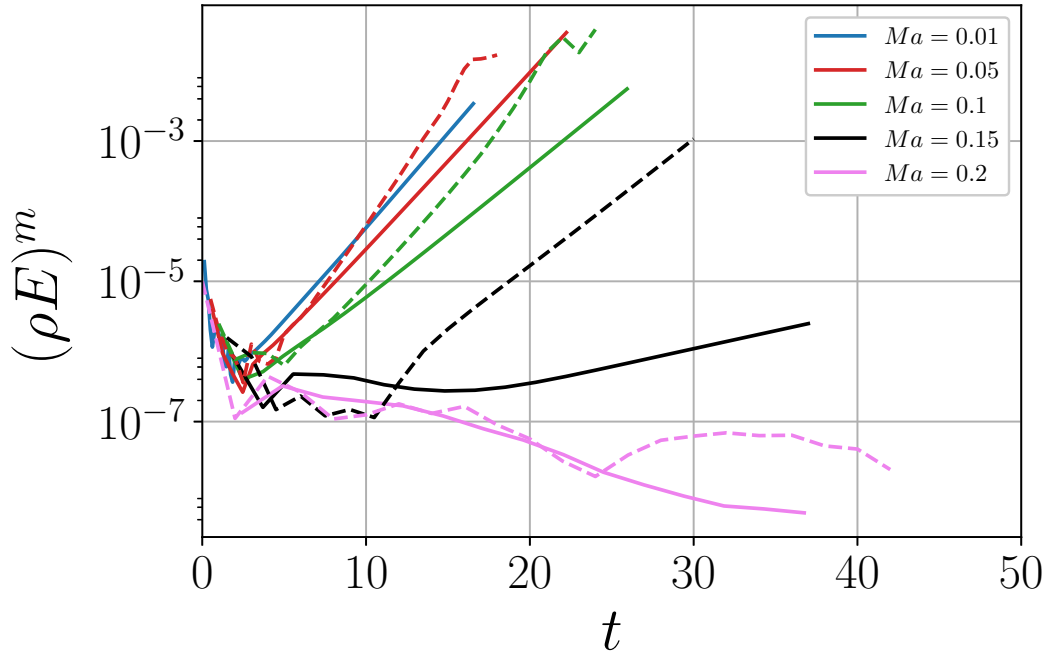


FIGURE 5.11: Time evolution of dominant mode in the azimuthal direction at $(z, r) = (0.527, 1.2)$ for — $Ra = 10^5$, --- $Ra = 10^7$ for $Ro = 0.199$ at different Ma numbers.

criterion for the onset of convection is derived. The flow stability is considered to be analogous to the static stability condition of the atmosphere [72]. For the convection instability to grow, the temperature gradient in the system has to be larger than the adiabatic temperature gradient. Thus, for a stable system the condition

$$\frac{dT}{dr} < \left(\frac{dT}{dr} \right)_{ad} \quad (5.3)$$

has to be satisfied. To elaborate on the stability condition given in equation 5.3, consider the upward and the downward motion of fluid parcels follows an adiabatic process, as shown in figure 5.13. It implies that the temperature of the fluid parcel can increase or decrease only by the work done during this motion and not by any heat transfer from the surroundings. To illustrate this, in figure 5.13, let us assume the initial position of the fluid parcel is at $r = r_0$ at temperature T_0 . Due to centrifugal forces, the fluid parcel moves to a new position at $r = r_1$ and attains the temperature value of T_1 by following the adiabatic temperature profile. If the temperature gradient $(dT/dr < (dT/dr)_{adia})$ is less than the adiabatic one, then the upward motion of the fluid parcel (adiabatically) leads to an increase

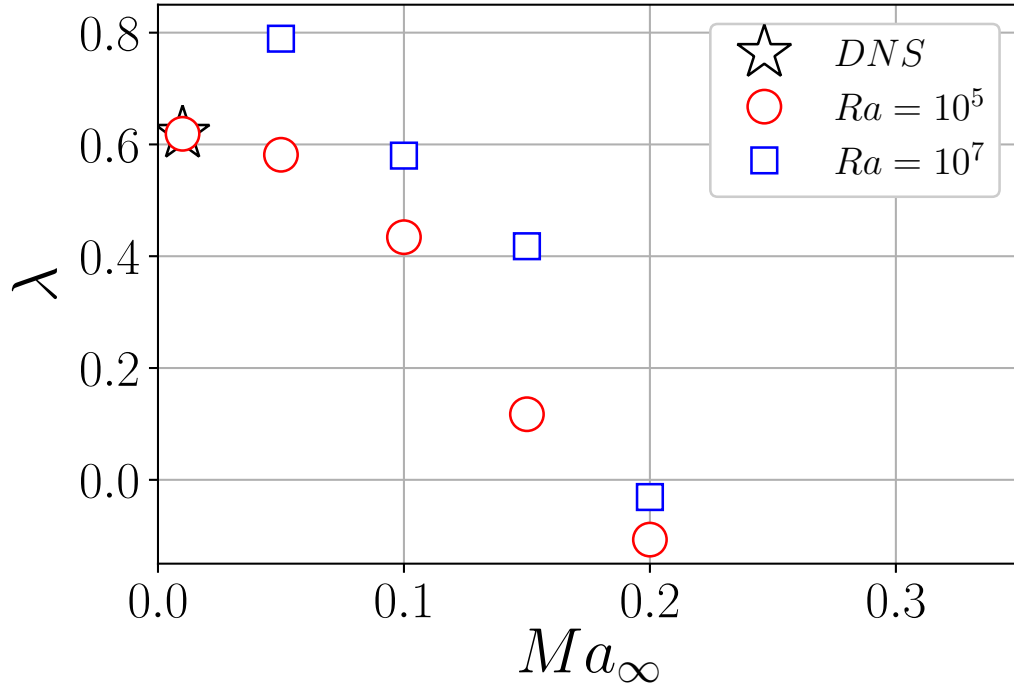


FIGURE 5.12: Comparison of growth rate of most unstable mode for different Ra numbers at $Ro = 0.199$.

in its temperature (decrease in the density) compared to its surroundings, i.e. $T_1 > T_{S1}$ as shown in figure 5.13. In the centrifugal force-field, a low-density fluid parcel in the high-density surroundings is pushed back to its original state, and thus the system is stabilized. On the other hand, a temperature gradient greater than the adiabatic temperature gradient ($dT/dr > (dT/dr)_{adia}$) leads to an unstable situation as in this case the fluid parcel will have low temperature value (or high density) compared to the surroundings ($T_1 < T_{S2}$). In this scenario, the higher centrifugal force on the heavier fluid parcel allows the instability to grow, making the flow unstable. The stability criterion given in equation 5.3 can also be expressed in terms of non-dimensional parameters by using the first law of thermodynamics and the pressure balance equation with the centrifugal force (refer to Appendix A for more details) in the system as

$$Ro < Ma_\infty^{1/2} \frac{(\gamma - 1)^{1/4}}{2^{3/2}} \left(\frac{1 + r_i/r_o}{1 - r_i/r_o} \right)^{1/2}. \quad (5.4)$$

Figure 5.14 shows the stability curve (black curve) as given by equation 5.4. In the plane of Ro – Ma_∞ , the stability curve takes the parabolic shape ($Ma_\infty \approx Ro^2$)

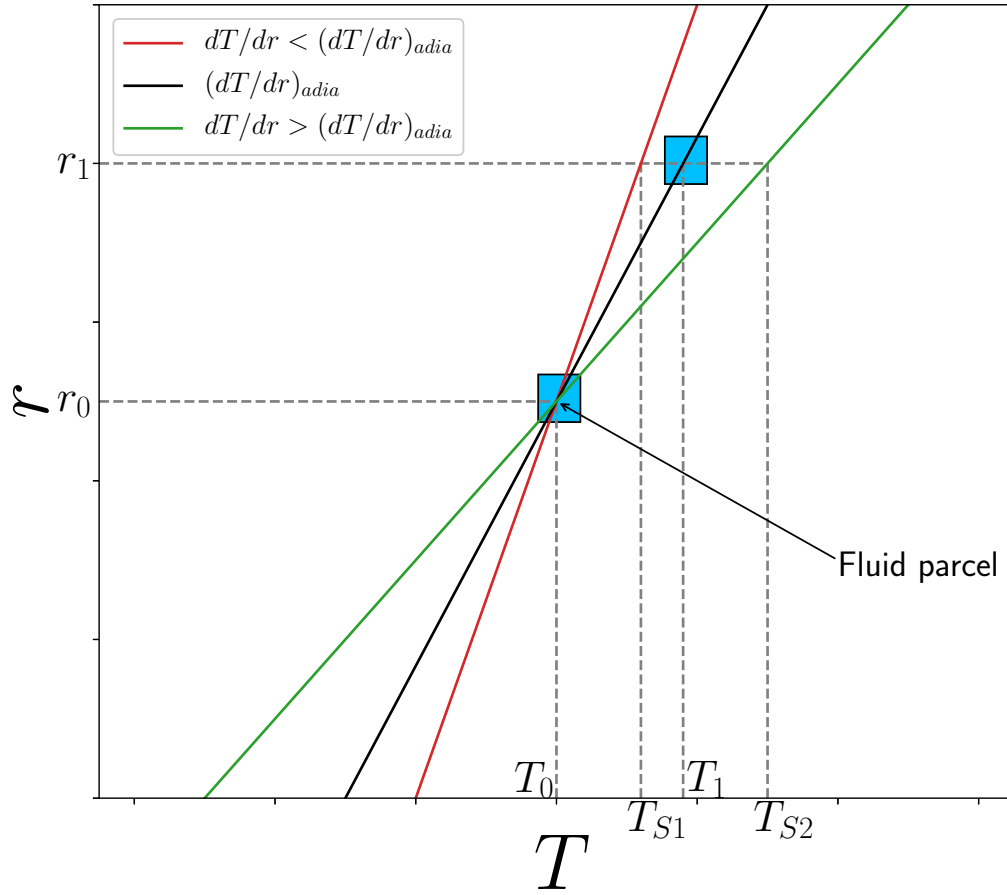


FIGURE 5.13: A schematic to illustrate the stability condition given in equation 5.3, assuming the fluid parcel follows the adiabatic temperature profile. The temperature profiles shown here for different possible scenarios are just the representation to explain the condition given in equation 5.3. The actual profiles might have different slopes.

which separates the stable region from the unstable region. The rate of increase of Ma number with the Ro number is twice on the stability curve. For both the Ra numbers of 10^5 and 10^7 , the conditions at $Ma = 0.2$ and $Ro = 0.199$ satisfy the above-mentioned stability criterion as shown in figure 5.14. To further test the validity of equation 5.4, additional linear stability calculations were conducted at $Ra = 10^7$ with Ro numbers of 0.1 and 0.244. At the lower Ro number of 0.1, the flow is stable even at the low Ma number of 0.05 as it satisfies the condition of stability (equation 5.4). At the higher Ro number of 0.244, the flow is observed to be unstable at $Ma = 0.2$ with the wavenumber 16 being the most unstable. However, for this combination of Ro and Ra , the flow becomes stable at a higher Ma number of 0.3, where it satisfies the stability criterion given in equation 5.4.

Thus, with the increase in the value of Ro number, the value of Ma number for the flow stability also increases as the equation 5.4 is quadratic in terms of Ma number dependence on Ro number.

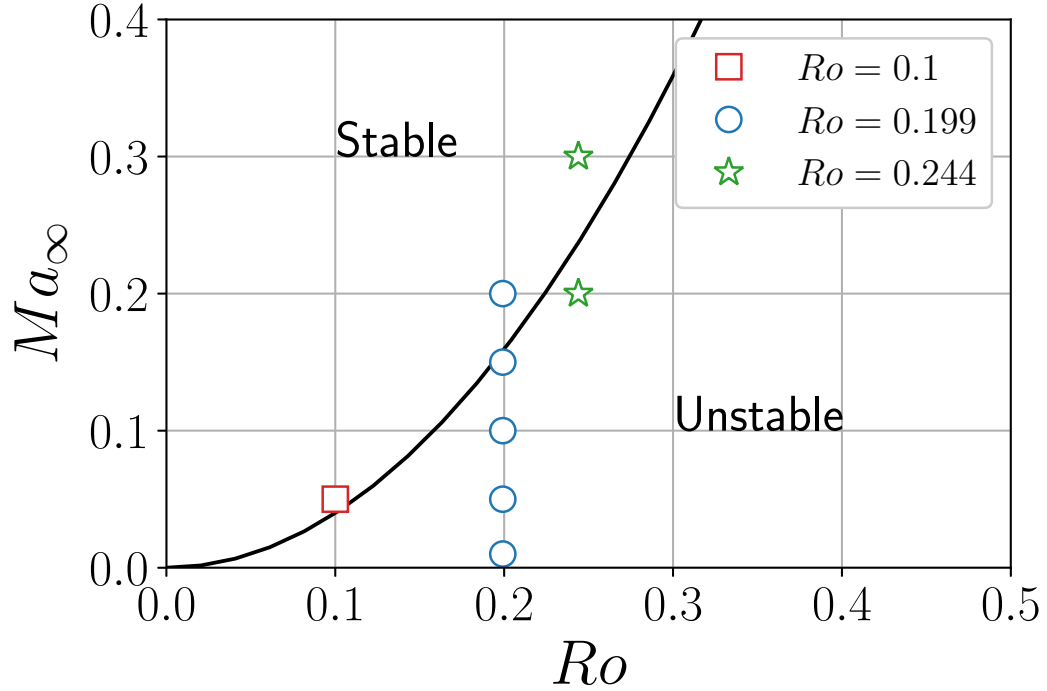


FIGURE 5.14: Stability curve as given in equation 5.4 and showing the conditions at different Ro numbers for $Ra = 10^7$. The black curve is the stability criterion defined by equation 5.3

5.4 Conclusions

The effect of compressibility in the flow was studied by varying the flow Mach number in the range $0.05 < Ma_\infty < 0.2$ for a fixed Ra and Ro number. With the increase in the value of Ma number, an increase in the radial pressure gradient near the hot wall and decrease near the cold wall was seen. As a consequence of this, the density of the fluid increased in the radial direction against the counteracting effects of the temperature gradient between the outer and the inner walls. This radial density distribution in the domain approached the stratified conditions as the Ma number was increased to the maximum value of 0.2. A reduction in the wall heat transfer rate or Nu number was reported with the increase in the value of Ma number. Thus, the compressibility effects led to a heat transfer reduction

by stabilizing the flow and suppressing the hydrodynamic instability responsible for the formation of convection rollers.

Further, linear stability analyses were conducted to understand the effect of compressibility at $Ra = 10^5$ and 10^7 . The propagation of the instability was observed to be asymmetric in the azimuthal direction. The results of linear stability analysis showed the amplification of multiple modes in the linear-regime and the superposition of all these modes appeared as a localized disturbance. Further, the linearly most amplified modes obtained from LSA have much higher wavenumbers compared to the mode that appeared to dominate in the saturation state of DNS. An increase in the Ma numbers appeared to have a stabilizing effect and suppressed the formation of convection rollers. A stability criterion was proposed that showed that both Rossby and Mach number affect the stability of the flow in the rotating cavities. The flow Mach number imposed a constraint on the critical value of Rossby number for the flow to be linearly stable.

Chapter 6

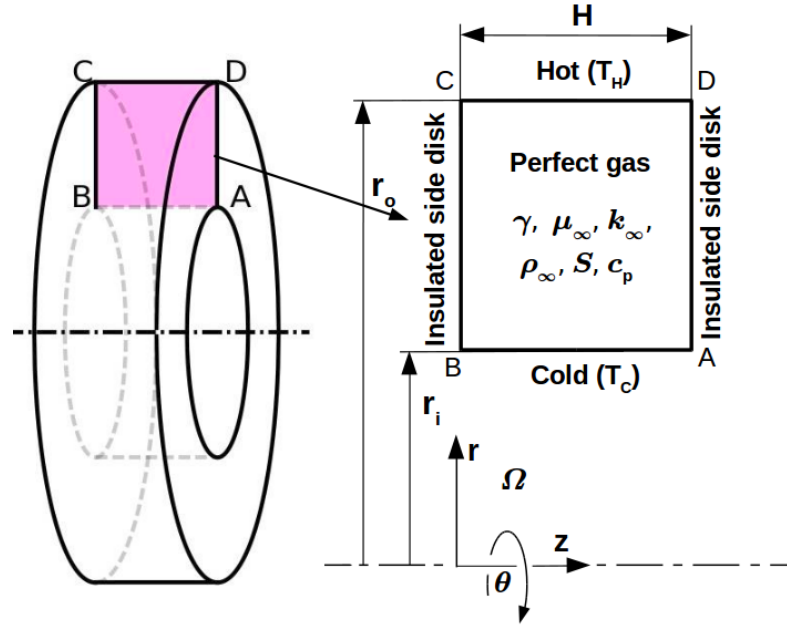
Effects of Rossby number in centrifugal buoyancy-induced flow

The objective of this chapter is to understand the effects of Rossby number in centrifugal buoyancy-induced flow in a closed rotating cavity. In the first section, the discussion revolves around understanding the influence of Ro on the flow structure and consequently on the heat transfer rate. The following section features the role of plumes in the local heat transfer enhancement.

6.1 Problem Setup

The details about the problem setup with the boundary conditions can be found in the chapter 4. The non-dimensional numbers that specify the problem setup (figure 6.1) are given as

$$\begin{aligned} Ra_\infty &= \frac{\rho_\infty^2 c_p (\Omega_\infty^2 r_m) (\Delta T / T_\infty) (r_o - r_i)^3}{\mu_\infty \kappa_\infty} = Re_\infty^2 Pr_\infty, \quad Pr_\infty \equiv \frac{\mu_\infty c_p}{\kappa_\infty} = 0.71, \\ \gamma &= 1.4, \quad \frac{S}{T_\infty} = 0.368, \quad Ma_\infty \equiv \frac{U_\infty}{(c_p(\gamma - 1)T_\infty)^{1/2}} = 0.1, \quad \frac{\Delta T}{T_\infty}, \\ \frac{r_i}{L_\infty} &= 1.083, \quad \frac{r_o}{L_\infty} = 2.083, \quad \frac{H}{L_\infty} = 1.043, \end{aligned} \quad (6.1)$$

FIGURE 6.1: Schematic of closed cavity rotating about the z -axis.

where Ra_∞ , Re_∞ , Pr_∞ and Ma_∞ are the reference Rayleigh (Ra), Reynolds (Re), Prandtl (Pr) and the Mach (Ma) numbers, respectively. The non-dimensional parameter, Rossby number (Ro) characterizes the strength of the Coriolis forces and is defined as

$$Ro \equiv \frac{[\Omega_\infty^2 r_m (\Delta T / T_\infty) (r_o - r_i)]^{1/2}}{2\Omega_\infty (r_o - r_i)} = \frac{1}{2^{3/2}} \left(\frac{1 + r_i/r_o}{1 - r_i/r_o} \right)^{1/2} \left(\frac{T_H}{T_C} - 1 \right)^{1/2} \quad (6.2)$$

A low value of Rossby number signifies strong influence of the Coriolis forces and vice-versa.

6.1.1 Definition of Nusselt number

The Nusselt number is the measure of effectiveness of convective heat transfer against the conductive mode of heat transfer and is defined as the ratio of convective heat flux at the wall to the conductive heat flux,

$$Nu \equiv \frac{q_{convection}}{q_{conduction}}. \quad (6.3)$$

The conductive heat flux in the domain can be derived by integrating the heat conduction equation

$$\frac{1}{r} \frac{d}{dr} \left(\kappa(T) r \frac{dT}{dr} \right) = 0. \quad (6.4)$$

In equation 6.4, $\kappa(T)$ is the thermal conductivity, a function of molecular viscosity μ and thus depends on the temperature as per Sutherland's law [52] given as

$$\kappa(T) = \frac{\mu(T)}{RePrMa^2(\gamma - 1)}, \quad \mu(T) = T^{3/2} \frac{1 + S/T_\infty}{T + S/T_\infty}. \quad (6.5)$$

The conductive heat flux in the domain can be derived by integrating the equation 6.4 as given below

$$\begin{aligned} q_{conduction} = & - \frac{2(1 + S/T_\infty)(S/T_\infty)^{3/2}}{Re_\infty Pr_\infty Ma_\infty^2(\gamma - 1) \log(r_o/r_i)} \\ & \times \left[\frac{1}{3} \left(\frac{T_H}{S/T_\infty} \right)^{1.5} - \frac{1}{3} \left(\frac{T_C}{S/T_\infty} \right)^{1.5} - \left(\frac{T_H}{S/T_\infty} \right)^{0.5} + \left(\frac{T_C}{S/T_\infty} \right)^{0.5} \right. \\ & \left. + \tan^{-1} \left(\sqrt{\frac{T_H}{S/T_\infty}} \right) - \tan^{-1} \left(\sqrt{\frac{T_C}{S/T_\infty}} \right) \right]^{-1}. \end{aligned} \quad (6.6)$$

The convective heat flux at the hot wall is defined as

$$q_{convection} = \frac{-1}{Re_\infty Pr_\infty Ma_\infty^2(\gamma - 1)} \int_0^{2\pi} \int_0^H \mu(T) \left[\frac{dT}{dr} \right]_{wall} r d\theta dz. \quad (6.7)$$

Therefore, by using the equations 6.6 and 6.7, the final form of Nusselt number is given as

$$\begin{aligned} Nu = & \frac{r_o \log(r_o/r_i)}{2H(S/T_\infty)^{3/2}(1 + S/T_\infty)} \left[\int_0^H \mu(T) \left[\frac{dT}{dr} \right]_{wall} dz \right] \\ & \times \left[\frac{1}{3} \left(\frac{T_H}{S/T_\infty} \right)^{1.5} - \frac{1}{3} \left(\frac{T_C}{S/T_\infty} \right)^{1.5} - \left(\frac{T_H}{S/T_\infty} \right)^{0.5} + \left(\frac{T_C}{S/T_\infty} \right)^{0.5} \right. \\ & \left. + \tan^{-1} \left(\sqrt{\frac{T_H}{S/T_\infty}} \right) - \tan^{-1} \left(\sqrt{\frac{T_C}{S/T_\infty}} \right) \right]^{-1}. \end{aligned} \quad (6.8)$$

This definition of Nusselt number takes into account the variation of thermal conductivity of the ideal gas with the temperature and thus provides the accurate

estimation of wall heat flux for different Ro number cases.

6.2 Results and Discussions

Direct numerical simulations for Rayleigh numbers in the range $10^5 < Ra < 10^9$ are carried out to study the effect of Ro number, as given in table 6.1. The Ro number values are considered in the range $0.25 \leq Ro \leq 0.75$ at a constant Ma number of 0.1. A constant value of Ma number signifies a constant effect of pressure or compressibility induced density variation. As per the definition of Ro given in equation 6.2, the way to vary its value is by specifying the respective values of temperature at the walls and the rotational speed of the cavity as given in table 6.1.

Case	Ra	Ro	T_H/T_C	Ω	Ma
1	$10^5, 10^6, 10^7, 10^8, 10^9$	0.25	1.157	2.000	0.1
2	$10^5, 10^6, 10^7, 10^8, 10^9$	0.50	1.630	1.000	0.1
3	$10^5, 10^6, 10^7, 10^8, 10^9$	0.75	2.417	0.667	0.1

TABLE 6.1: DNS cases to study the effect of Rossby number.

6.2.1 Grid quality

The grid-spacings from chapter 4 are considered as a reference to estimate the grid resolution requirements for the current Rossby number study. The grid metrics discussed in chapter 4 are used here to assess the grid quality. To assess the near-wall grid resolution, the grid-spacings in wall units for Ra numbers 10^7 , 10^8 and 10^9 at $Ro = 0.75$ are shown in figures 6.2 and 6.3. For all the cases, the radial wall-normal spacing Δr^+ at the top wall is less than 0.08 as shown in figure 6.2 (a). In the axial direction, the grid is stretched with the maximum spacing $\Delta z^+ < 0.75$ for all Ra numbers and in the azimuthal direction, the grid-spacing $r\Delta\theta^+$ is close to 1.5 for $Ra = 10^9$ and less than that for the other cases. For the side insulated wall, the maximum wall-normal spacing Δz^+ is less than 0.3 for all

the cases, and the spacing in the radial direction Δr^+ has a highest value close to 1.0. The grid spacing in the azimuthal direction $r\Delta\theta^+$ at the insulated wall has a peak value close to 6.0 near the top wall for the $Ra = 10^9$ case. Further, the maximum values of the near-wall grid-spacings for the hot and the insulated walls for other Ro number cases are provided in table 6.2 and all the values are less than 0.5. In the azimuthal direction, the grid-spacing is kept close to or less than the resolution used in chapter 4 and in wall units the value of $r\Delta\theta^+$ is less than 7.0 for all the cases.

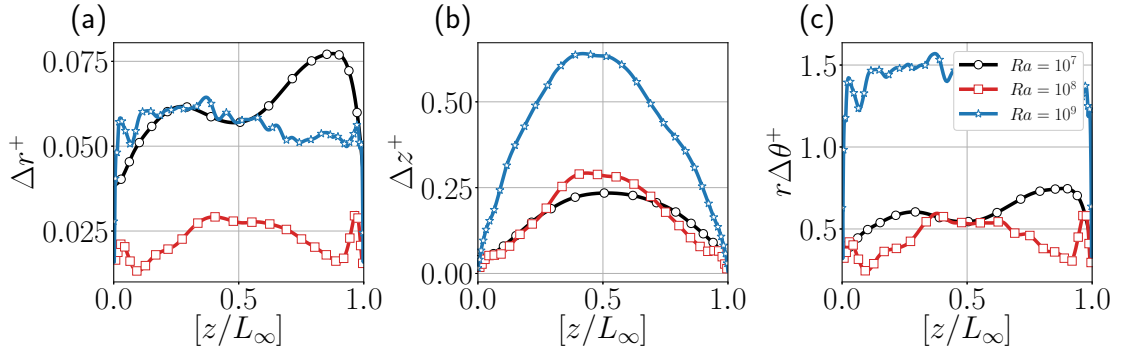


FIGURE 6.2: Grid-spacing in wall units for the top hot wall at $Ra = 10^7$, 10^8 and 10^9 for $Ro = 0.75$.

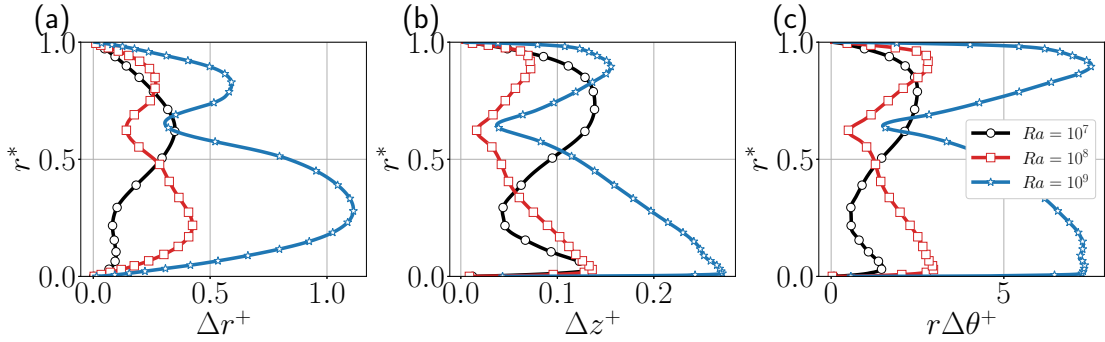


FIGURE 6.3: Grid-spacing in wall units for the side insulated wall at $Ra = 10^7$, 10^8 and 10^9 for $Ro = 0.75$, where $r^* = [r(r_i/L_\infty)]/[(r_o r_i)/L_\infty]$.

The near-wall resolution for the thermal boundary layers formed at the hot and the cold walls are compared against the lower bounds for the same provided by Shishkina et al. [61] in table 6.3. The thermal boundary thickness is calculated based on the maximum value of T_{rms} near the hot and the cold walls. The comparison indicates that the thermal boundary layers have a sufficient grid resolution.

Ra	$N_z \times N_r \times N_\theta$	Δr_{max}^+ at the hot wall for $Ro = 0.25, 0.5, 0.75$			Δz_{max}^+ at the side wall for $Ro = 0.25, 0.5, 0.75$		
10^5	$80 \times 80 \times 194$	0.120,	0.085,	0.067	0.065,	0.100,	0.093
10^6	$160 \times 160 \times 194$	0.088,	0.066,	0.062	0.071,	0.102,	0.098
10^7	$192 \times 192 \times 258$	0.106,	0.078,	0.076	0.102,	0.163,	0.142
10^8	$320 \times 320 \times 514$	0.050,	0.044,	0.031	0.104,	0.150,	0.136
10^9	$380 \times 380 \times 514$	0.160,	0.120,	0.069	0.350,	0.320,	0.290

TABLE 6.2: The comparison of grid resolution in the thermal boundary layer against the guidelines provided by Shishkina et al. [61]

Ra	Cold wall N_{BL}^{th} for $Ro = 0.25, 0.5, 0.75$			Hot wall N_{BL}^{th} for $Ro = 0.25, 0.5, 0.75$			min N_{BL}^{th} required [61]
10^5	16,	17,	20	16,	21,	32	4
10^6	20,	20,	31	19,	21,	37	5
10^7	16,	17,	22	18,	17,	23	7
10^8	30,	29,	33	33,	30,	34	10
10^9	22,	19,	23	20,	20,	24	14

TABLE 6.3: The comparison of grid resolution in the thermal boundary layer against the guidelines provided by Shishkina et al. [61]

Thus, the grid assessment and the comparison discussed above indicate that the simulations are well resolved.

6.3 Effect of Rossby number on the flow structure and the heat transfer

Figure 6.4 shows the instantaneous temperature and the radial velocity contours in the r - z plane at the mid-axial location of the cavity for different Ro numbers at $Ra = 10^8$ and 10^9 . The consecutive pairs of convection rollers is a distinct feature for both the Ra numbers. These convection rollers are mainly responsible for thermal mixing in the core region. The plumes formed at the centre of each roller carry the cold and the hot fluid from the respective boundary layers as shown by positive and negative radial velocity in figures 6.4(d-f) and 6.4(j-l). The number of convection rollers in the domain varies with the Ro number. There are five rollers for $Ro = 0.25$ while there are only four for the $Ro = 0.5$ and 0.75 cases for

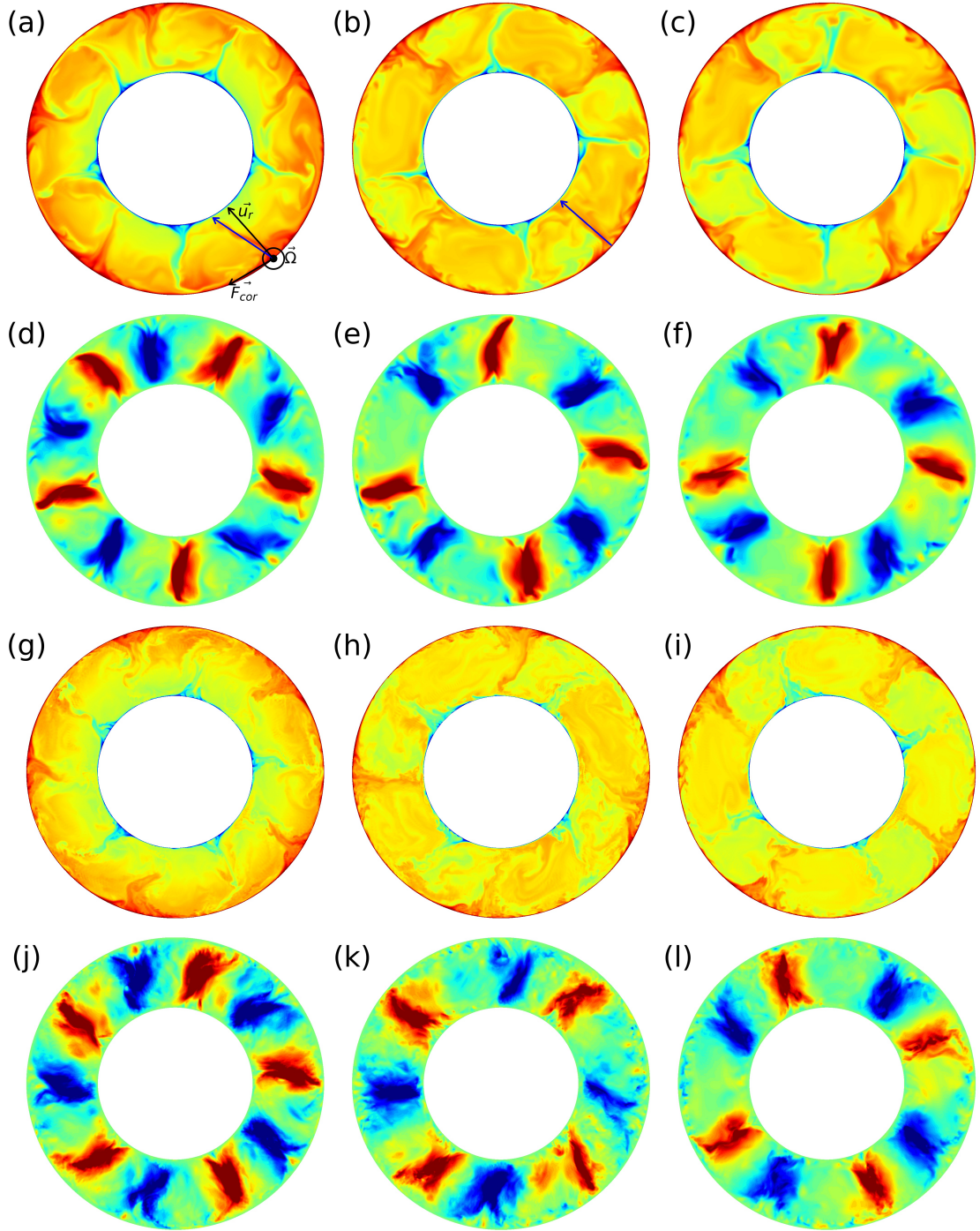


FIGURE 6.4: Instantaneous temperature and radial velocity contours at the mid-axial position for $Ra = 10^8$ at (a,d) $Ro = 0.25$, (b,e) $Ro = 0.5$ (c,f) $Ro = 0.75$, respectively, and ; for $Ra = 10^9$ at (g,j) $Ro = 0.25$, (h,k) $Ro = 0.5$ (i,l) $Ro = 0.75$, respectively. The contour levels for the temperature and the radial velocity are $[-0.5, 0.5]$ and $[-0.25, 0.25]$, respectively.

both the Ra numbers. Another effect of Ro number is the deflection of the plumes that rise from the cavity walls away from the wall-normal direction. As shown in figure 6.4 (a), the plume rises from the hot wall (blue arrow) deviating from its wall-normal direction to a certain angle. A similar effect can be perceived for $Ra = 10^9$ as well at approximately the 7 o'clock position in figure 6.4 (g). On the other hand, for $Ro = 0.5$ in figure 6.4(b), the plume path is nearly aligned with the wall-normal direction. The reason for this deflection of plumes is the higher magnitude of the Coriolis forces at $Ro = 0.25$ case. The Coriolis force (F_{cor}) is defined as $\vec{F}_{cor} = 2\vec{\Omega} \times \vec{u}_r$, where $\vec{\Omega}$ is the angular velocity in the direction out of the plane as the cavity is rotating in an anti-clockwise direction and $\vec{u}_r$ is the radial velocity, as shown in figure 6.4 (a). The direction of the Coriolis force $\vec{F}_{cor}$ is perpendicular to the direction of radial velocity $\vec{u}_r$ (refers to figure 6.4 (a)). Thus, the higher magnitude of F_{cor} at $Ro = 0.25$ leads to a larger deviation of the plumes away from the radial direction. Figure 6.5 shows the temperature and the radial velocity contours in r - z plane. The circumferential locations are chosen to highlight the different characteristics of the convection in the domain. In figure 6.5(a) and 6.5(d) show a section of the convection roller where in the middle of the cavity radial velocity is positive with hot and cold fluid at the outer and the inner walls, respectively. The circulation region or wind is shown in figures 6.5(b) and 6.5(e) with a plume region shown in figures 6.5(c) and 6.5(f). Further discussion on these different regions is presented in section 6.6.1 .

The higher magnitude of the Coriolis forces for low Ro number cases not only deflects the plumes in the r - θ plane, but it also influences their structures in the z - θ plane. Figure 6.6 shows the instantaneous radial velocity contours in the z - θ plane at a fixed radial location close to the thermal boundary layer formed at the hot wall. The plumes which are represented by the positive and the negative radial velocity at $Ro = 0.25$ maintain their coherence along the z direction. The comparison with the other Ro numbers of 0.5 and 0.75 shows the breakdown of the plume structures in the z direction, as shown in figures 6.6(b) and (c). It implies that the flow approaches a quasi-two-dimensional state due to higher magnitude of Coriolis forces at $Ro = 0.25$. To further elaborate on this effect of Coriolis

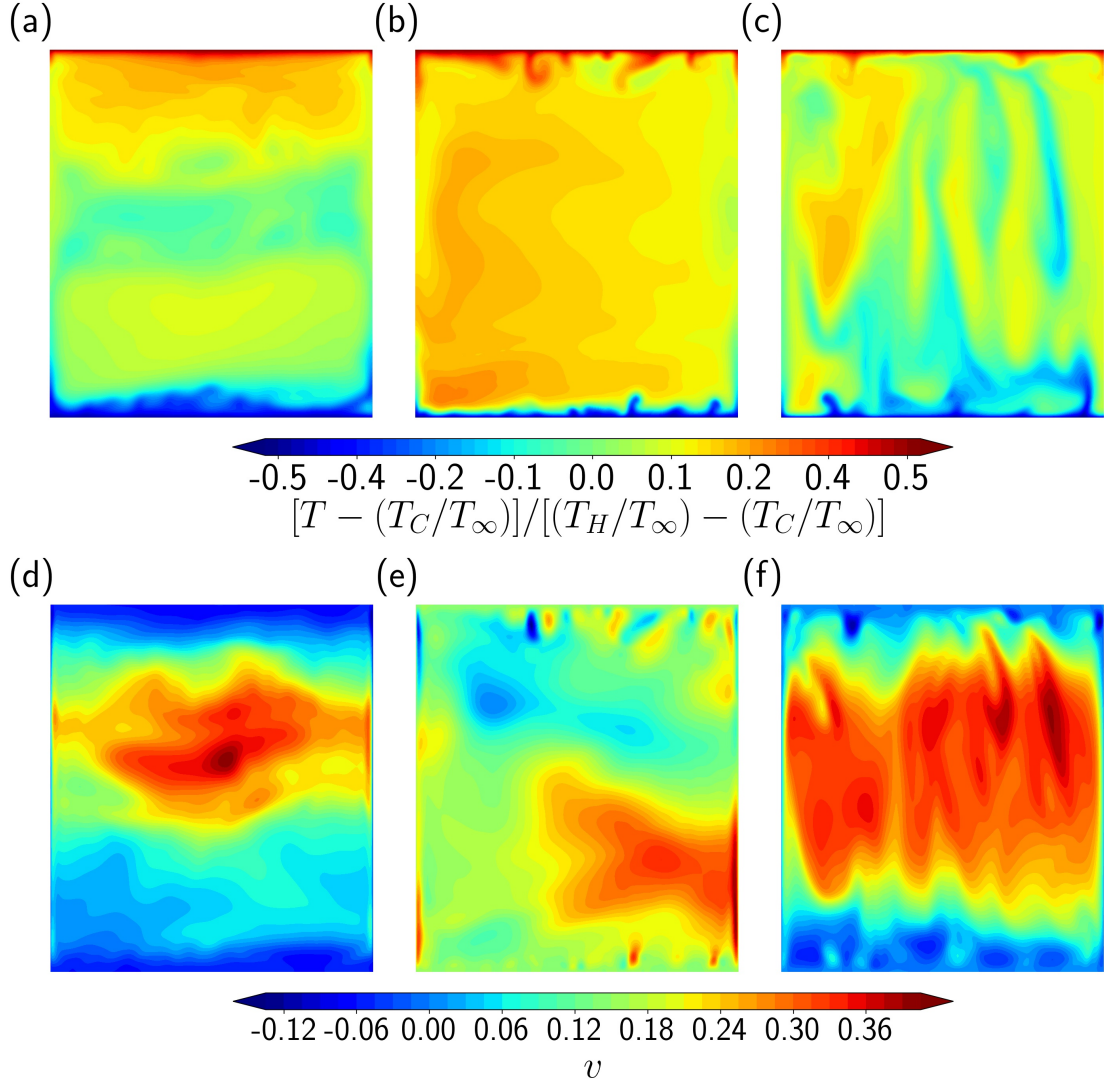


FIGURE 6.5: Instantaneous temperature and radial velocity contours in r - z plane at a fixed circumferential location for $Ra = 10^8$ at (a,d) $Ro = 0.25$, (b,e) $Ro = 0.5$ (c,f) $Ro = 0.75$, respectively.

forces, the momentum equation is considered in the rotating frame of reference and simplified by considering the following assumptions for rapidly rotating turbulent flows:

- the Coriolis terms dominate over the convection terms;
- viscous effects are negligible in the core of the convection at high Ra number;
- the flow is at a statistically steady state.

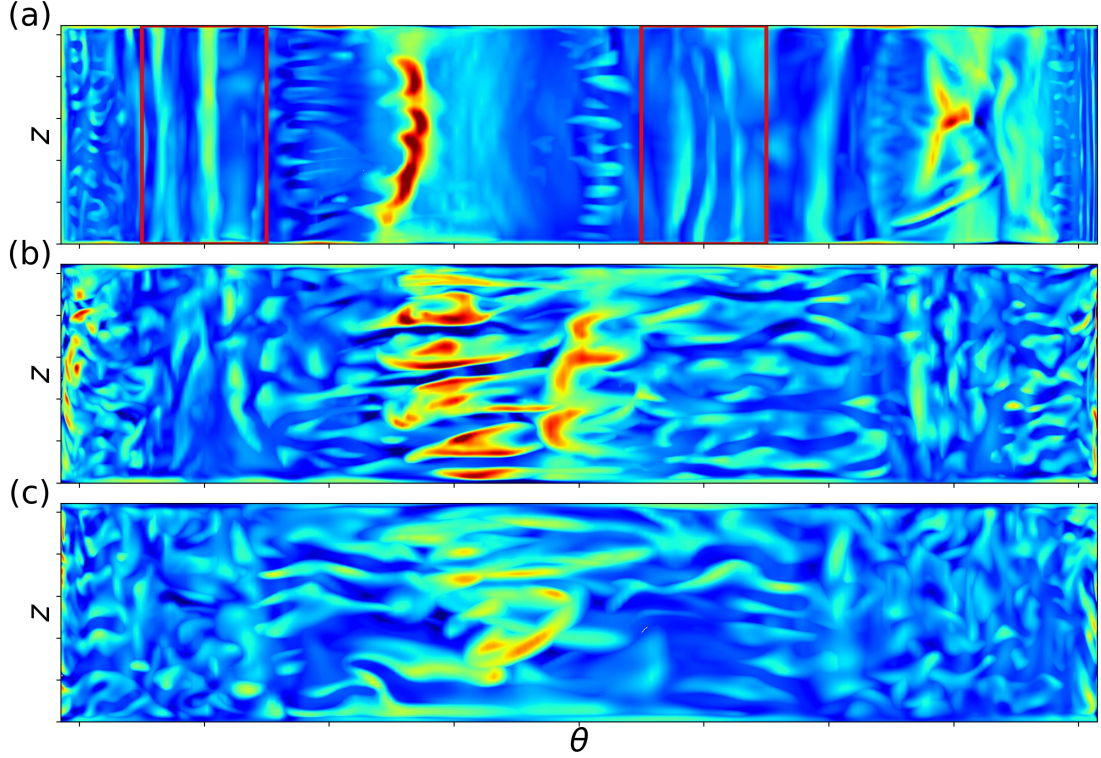


FIGURE 6.6: (a) Instantaneous radial velocity contours in the z - θ plane at radial location $(r - r_i)/(r_o - r_i) = 0.966$ for Ro (a) 0.25, (b) 0.5 and (c) 0.75 at contour levels $[-0.05, 0.18]$.

The momentum equation then becomes

$$-\nabla p = 2\rho\Omega \times u_i + \rho\Omega \times (\Omega \times r). \quad (6.9)$$

The equation 6.9 is also known as equation of geostrophic balance or Taylor–Proudman theorem, where the pressure gradient in the system gets balanced by the Coriolis term and the centrifugal term. Further simplifications can be made to equation 6.9 by taking the curl and average it over time to yield

$$\partial_z(\bar{\rho} \bar{u}_z) + \partial_z(\overline{\rho' u_z'}) = 0. \quad (6.10)$$

The equation 6.10 signifies that rotating flows in the geostrophic balance have a zero variation of ρu_z in the axial direction, which implies that the flow becomes two-dimensional. The terms in the equation 6.10 are computed in the core of the cavity away from the walls for high Ra number cases, as shown in figure 6.7. In

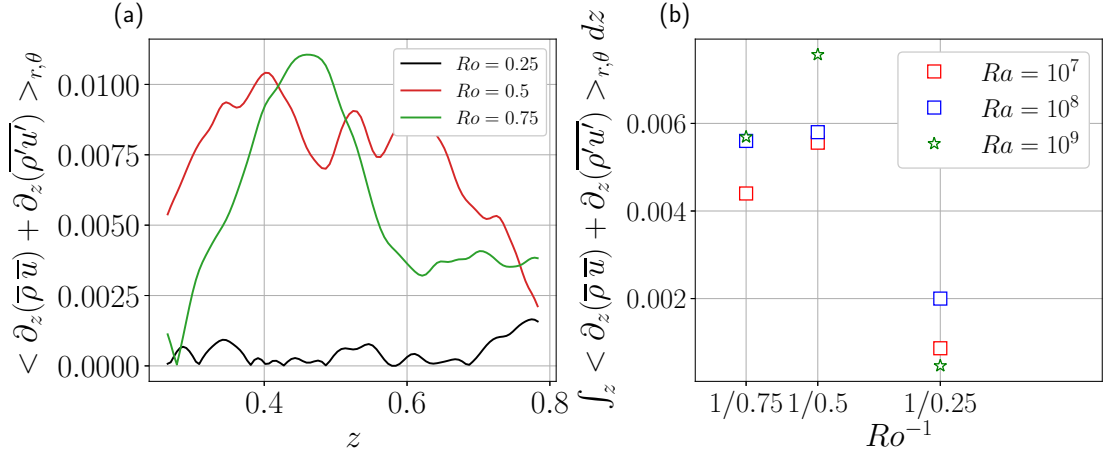


FIGURE 6.7: (a) The variation of quantity given in equation 6.10 with axial coordinate for $Ra = 10^9$, and (b) the variation of integrated values over axial direction of the same term with Ro number for different Ra numbers.

figure 6.7(a), the equation 6.10 is averaged over the radial and the circumferential directions for the $Ra = 10^9$ case at different Ro numbers. For $Ro = 0.25$, the value is close to zero with minimal variation in the axial direction and it satisfies the conditions given in equation 6.10. For the other Ro numbers a significant variation indicates the axial breakdown of the plumes. For the other Ra number cases, the equation 6.10 is further integrated in the axial direction as well, as shown in figure 6.7(b). The comparison shows that at low Ro number the Coriolis forces stabilize the flow in the axial direction as it satisfies the condition given in equation 6.10 for all Ra numbers. Thus, the higher magnitude of the Coriolis forces at low Ro number constrains the flow to a quasi-two-dimensional state in the z - θ plane.

The variation of the Ro has an influence on the Nu number as well. Figure 6.8 shows that for the low Ra number cases ($Ra = 10^5, 10^6$), the difference in the values of Nu number is not significant, as the flow resembles the laminar conditions. But, for $Ra \geq 10^7$, with the decrease in the value of Ro number, the heat transfer rate reduces. Additional data points for $Ro < 0.25$ from chapter 4 and LES of Gao et al. [29] provide more evidence to the trend of heat transfer reduction at low Ro number. Unequivocally, the reason for this reduction in the value of Nu number is due to the stabilizing effect of Coriolis forces which constrains the flow to a quasi-two-dimensional state and inhibits the thermal mixing in the core of the

cavity.

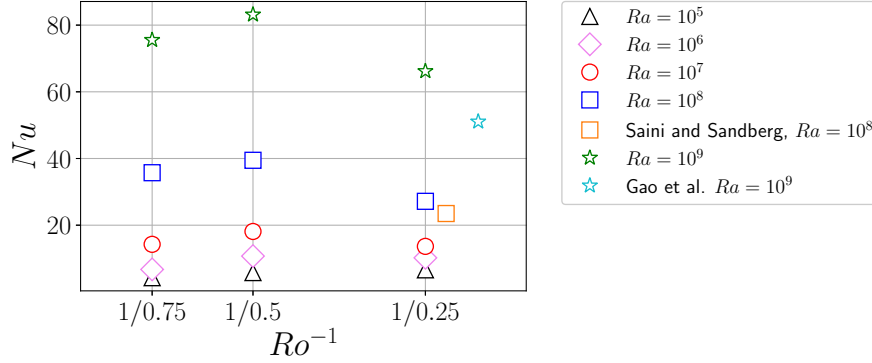


FIGURE 6.8: The variation of Nusselt number at the hot wall with Rossby number and the data from other studies by Saini and Sandberg [58] at $Ro = 0.23$ and $Ra = 10^8$ and by Gao et al. [29] at $Ro = 0.202$ and $Ra = 10^9$.

6.4 Effect of viscosity variation with temperature on the heat transfer rate

The molecular viscosity as a function of temperature is given by Sutherland's law as

$$\mu(T) = T^{3/2} \frac{1 + S/T_\infty}{T + S/T_\infty}, \quad (6.11)$$

which increases with the temperature. The effect of the change in the fluid properties with the temperature in the Rayleigh-Bénard convection are known as the Non-Oberbeck-Boussinesq (NOB) effects and were studied extensively by Ahlers et al. [73]. Though the focus of this chapter is to understand the effects of Ro number, the NOB effects become an inevitable part of the problem as Ro number is a function of temperature difference. Therefore, in this section the evaluation of the NOB effects on the wall heat transfer rate is conducted.

Figures 6.9 and 6.10 show the mean temperature profiles at different Ra numbers compared at different Ro numbers. For low Ra numbers of 10^5 and 10^6 , the temperature profiles have a maximum wall-gradient at $Ro = 0.25$, which decreases with the increase in the Ro number. A large temperature gradient in the convection core also exists for the $Ro = 0.75$ case. The reason behind this is a dominance of

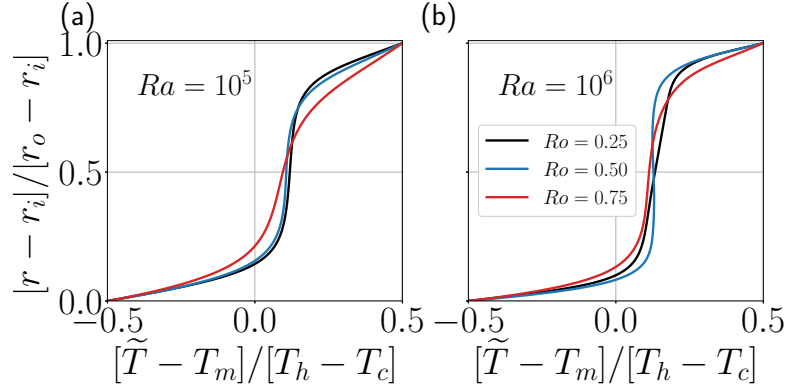


FIGURE 6.9: The comparison of mean temperature profiles at low Ra numbers for different Ro numbers.

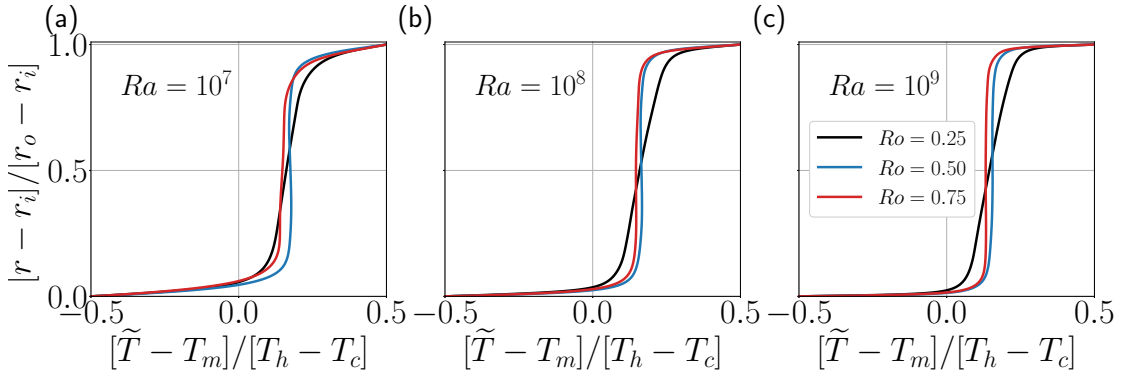


FIGURE 6.10: The comparison of mean temperature profiles at high Ra numbers for different Ro numbers.

viscous forces at high Ro number cases due to a higher temperature ratio, which also leads to a reduction in the value of Nu number, as shown in figure 6.8. Thus, at low Ra numbers the strong viscous forces at high Ro number counteracts the effect of inertial forces to the heat transfer enhancement. For $Ra \geq 10^7$, the inertial forces dominate over the viscous and the Coriolis forces. At $Ra = 10^7$ and $Ro = 0.5$ the temperature profile has a maximum wall-gradient, which drops slightly at $Ro = 0.75$ as shown in figure 6.10(a). However, at $Ra = 10^8$ and 10^9 , the near-wall temperature gradient rises with the Ro number and the isothermal conditions emerge in the core.

Another notable NOB effect is an asymmetric mean temperature profiles about the centre of the cavity. Figure 6.11 shows a typical mean temperature profile with the notations that are used in the following discussion.

The thermal boundary layer thickness λ^θ is defined as a radial location near the

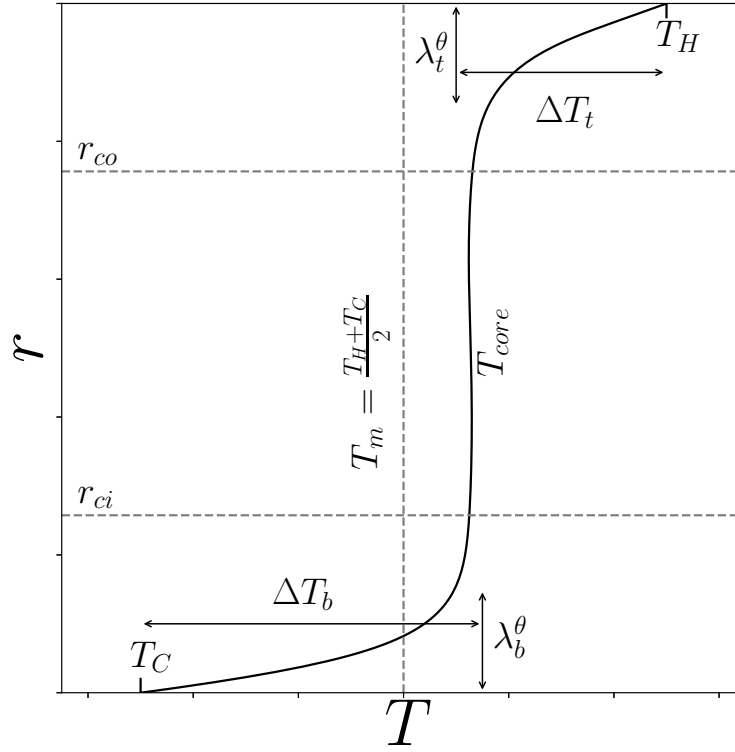


FIGURE 6.11: A schematic of mean temperature profile highlighting the asymmetry and nomenclature in the analysis.

inner and the outer walls where the temperature variance T_{rms} has a maximum value [74]. Due to the non-isothermal conditions in the core at $Ro = 0.25$, the core temperature in the cavity is calculated by averaging over the radial locations r_{ci} to r_{co} as shown in figure 6.11. The asymmetry in the temperature profiles is quantified by defining a parameter χ_θ as

$$\chi_\theta = \frac{T_{core} - T_C}{T_H - T_{core}}, \quad (6.12)$$

where T_{core} is the temperature of the core and T_H , T_C are the temperature values at the hot wall and the cold wall, respectively. The model of Wu and Libchaber [64] is extended here to devise a theoretical form of χ_θ to understand the various terms responsible for asymmetric mean temperature profiles. In this analysis, the thermal boundary layers are assumed to be marginally stable, which leads to a condition of Ra numbers, based on the top and the bottom thermal boundary

layers thicknesses, being equal as

$$Ra_t = Ra_b$$

$$(\Omega^2 r_o) \left(\frac{\beta}{\nu \alpha} \right)_t (\lambda_t^\theta)^3 (T_H - T_{core}) = (\Omega^2 r_i) \left(\frac{\beta}{\nu \alpha} \right)_b (\lambda_b^\theta)^3 (T_{core} - T_C). \quad (6.13)$$

In equation 6.13, β , ν and α are the values of the thermal expansion coefficient, kinematic viscosity and thermal diffusivity for the respective boundary layers. The heat fluxes from the inner cylindrical surface to the outer cylindrical surface are related by

$$\frac{\kappa_t}{\lambda_t^\theta} (T_H - T_{core}) (2\pi r_o H) = \frac{\kappa_b}{\lambda_b^\theta} (T_{core} - T_C) (2\pi r_i H), \quad (6.14)$$

where κ is the thermal conductivity. Thus, by using equations 6.13 and 6.14, the relationship for χ_θ can be derived as

$$\chi_\theta = \frac{T_{core} - T_C}{T_H - T_{core}} = \left(\frac{r_o}{r_i} \right) \left[\left(\frac{\kappa_t}{\kappa_b} \right)^3 \frac{[\beta/(\nu\alpha)]_t}{[\beta/(\nu\alpha)]_b} \right]. \quad (6.15)$$

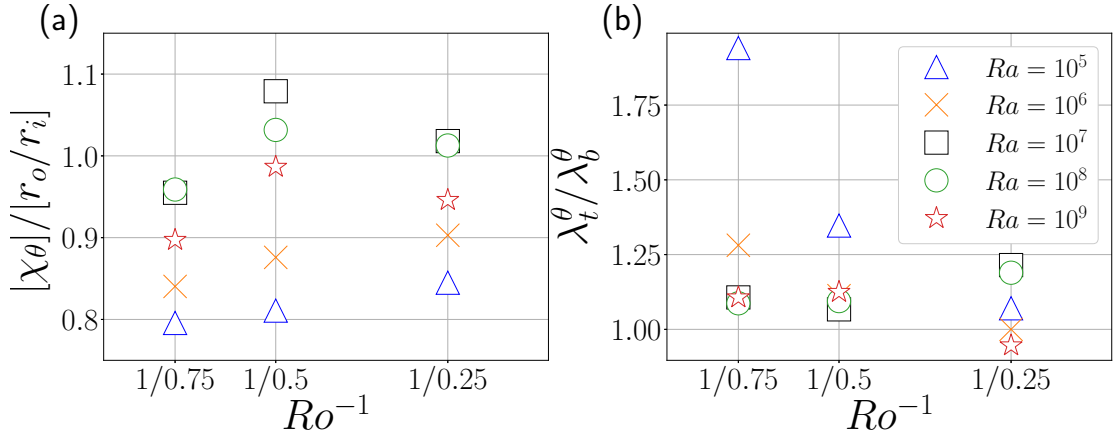


FIGURE 6.12: The comparison of (a) χ_θ , which represents the core temperature rise and (b) the ratio of thermal boundary layer thickness for different Ra numbers. Both these quantities estimate the contribution of NOB effects towards the asymmetry of the temperature profiles.

The equation 6.15 signifies that there are two terms which are responsible for the asymmetric temperature profiles: a factor of geometrical asymmetry r_o/r_i and the variation of fluid properties with the temperature. Thus, in equation 6.15, the term $[\chi_\theta/(r_o/r_i)]$ represents the contribution of the fluid properties alone

towards the asymmetry. The value of $[\chi_\theta/(r_o/r_i)]$ will be 1.0 for an Oberbeck–Boussinesq fluid and is considered as a baseline for comparison. Figure 6.12(a) shows the comparison of $[\chi_\theta/(r_o/r_i)]$ for different cases. At $Ra = 10^5$ and $Ro = 0.75$, $[\chi_\theta/(r_o/r_i)]$ has a minimum value close to 0.8 and it increases with a decrease in the value of Ro number. A similar trend is observed for the $Ra = 10^6$ case, with the values of $[\chi_\theta/(r_o/r_i)]$ relatively closer to 1.0 compared to the $Ra = 10^5$ case. For $Ra \geq 10^7$, $(\chi_\theta)/(r_o/r_i)$ varies in the range $0.9 < [\chi_\theta]/[r_o/r_i] < 1.1$, which is much closer to the reference value of 1.0. Thus at high Ra number, there is a maximum 10% contribution of fluid properties towards the asymmetry of mean temperature profiles.

Another quantity used to evaluate the asymmetry of the temperature profiles is the ratio of thermal boundary layers thicknesses at the top and the bottom walls as shown in figure 6.12(b). The comparison shows that the variation of $\lambda_t^\theta/\lambda_b^\theta$ is significant for low Ra number cases and the ratio increases with the Ro number. For $Ra \geq 10^7$, the variation of $\lambda_t^\theta/\lambda_b^\theta$ with the Ro number is minimal. Hence, the mean temperature profiles approach a symmetric state at high Ra numbers as the contribution of the fluid properties as a function of temperature attenuates.

6.5 Scaling of Nusselt number with Rayleigh number

The focus of this section is to understand the scaling of Nu number with the Ra number. Figure 6.13 shows the mean temperature profiles averaged over the axial and the circumferential directions for different Ro numbers. A distinctive non-isothermal core exists for all Ra numbers at $Ro = 0.25$ as shown in figure 6.13(a). The reason for this, as discussed in section 6.3, is the higher magnitude of the Coriolis forces, which leads to two-dimensionality of the flow in the axial direction and suppresses the breakdown of plumes, thus reducing the thermal mixing in the core. However, as the Ro number increases the isothermal convection core appears with the increase in the value of Ra numbers as shown in figures 6.13(b)

and 6.13(c). The extent of the convection core also increases with the Ra number which leads to a decrease in the thickness of the thermal boundary layers.

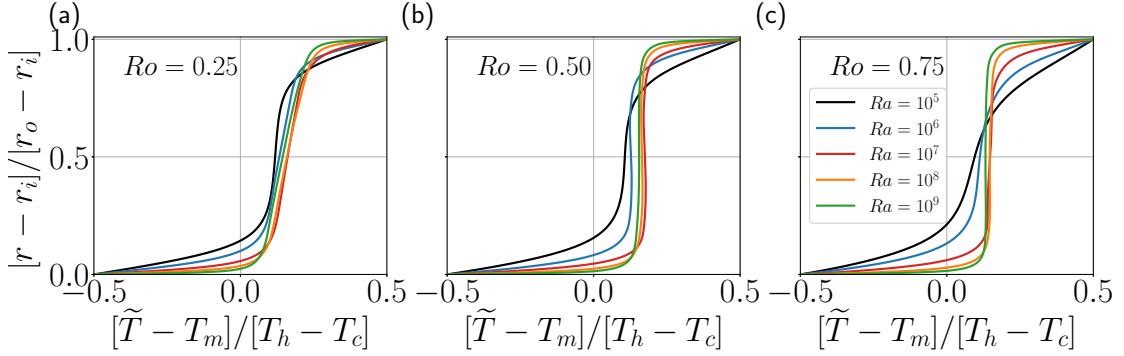


FIGURE 6.13: The comparison of mean temperature profiles for different Ro numbers.

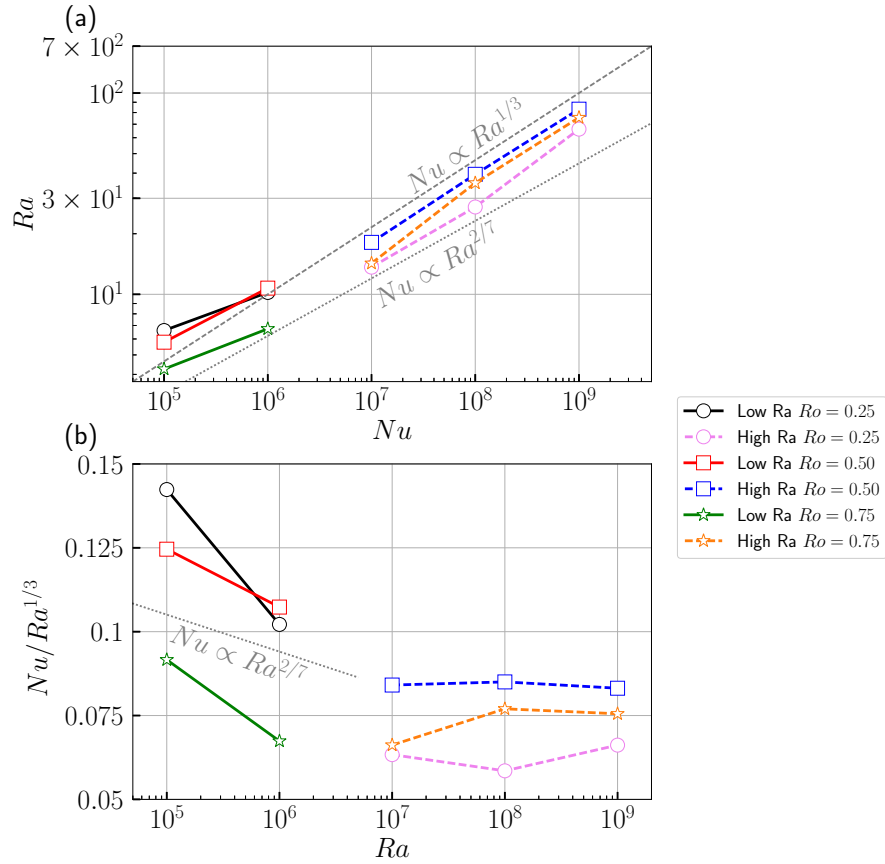


FIGURE 6.14: Comparison of Nu number scaling with Ra number for different Ro number.

The effectiveness of heat transfer is measured by Nu number. The scaling of Nu number with Ra number connotes the change in the vigour of convection with the change in the control parameters as shown in figure 6.14. For low Ra numbers of

10^5 and 10^6 , the Nu – Ra power exponent has a value close to $2/7$ for $Ro = 0.5$, shown in figure 6.14. For other Ro numbers, the scaling exponent value further drops to a value less than $2/7$.

The Nu – Ra scaling exponent for $Ra \geq 10^7$ for all Ro numbers follows the $1/3$ scaling law, as shown in figure 6.14(a-b). There is a slight drop in the value of Nu number at $Ra = 10^7$ for $Ro = 0.75$, which leads to a small deviation from $1/3$ scaling law. For the other Ro numbers, the Nu – Ra scaling exponent conforms to $1/3$, a classical scaling law for Rayleigh–Bénard convection [75].

6.5.1 Comparison of Nu – Ra scaling with other studies

The scaling of Nu with Ra number is compared with the results of other studies for the same cavity configuration, as shown in figure 6.16. The initial comparison is with the experimental results of Bohn et al. [23]. In the experiments, the cavity was pressurized at high Ra number to achieve the required physical conditions at a low RPM for a small temperature difference between the outer and the inner walls of the cavity. As a consequence, the value of Ro number at a higher Ra number was smaller than its value for low Ra number cases. The comparison of Nu number variation with Ro number is shown in figure 6.15. The experimental conditions of Bohn et al. [23] show that Ro number values decrease with an increase in Ra number and it leads to a decrease in the value of Nu number. The Nu – Ra scaling exponent in the experiments was 0.211, which increased to 0.297 when corrected for heat loss from the side disks. In chapter 4, the Nu – Ra scaling law for conditions replicating the experiments was found to be consistent with the corrected value of 0.297 as shown in figure 6.16.

For a consistent variation of Ro number the classical scaling law of $1/3$ for Rayleigh–Bénard convection [75] appears at $Ra \geq 10^7$ as shown in figure 6.16. This provides evidence that the decrease in the value of Ro number at higher Ra numbers was the reason behind the reduced scaling coefficient observed by Bohn et al. [23]. The

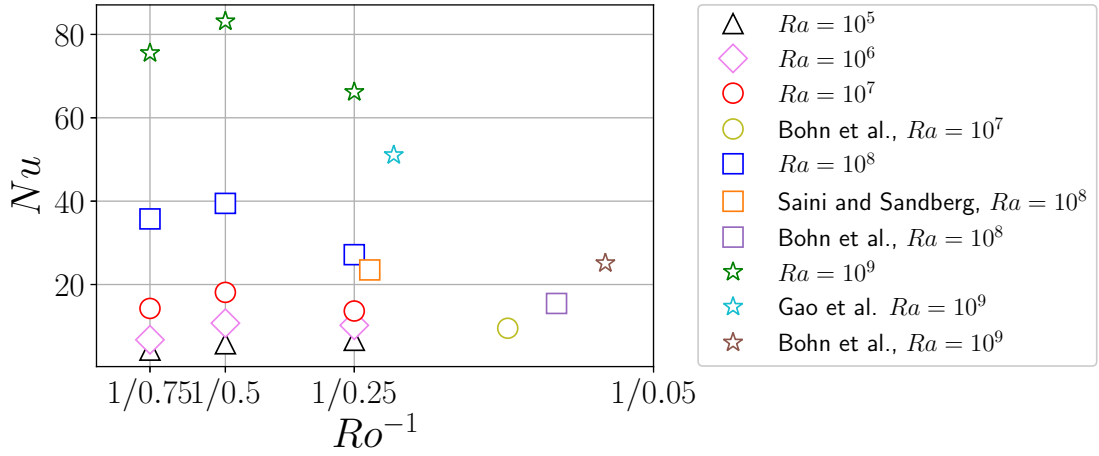


FIGURE 6.15: The variation of Nusselt number at the hot wall with Rossby number with the data from other studies by Saini and Sandberg [58] for $Ra = 10^8$ at $Ro = 0.23$, Gao et al. [29] for $Ra = 10^9$ at $Ro = 0.202$ and Bohn et al. [23] for $Ra = 10^7$, 10^8 and 10^9 at $Ro = 0.109$, 0.084 and 0.065 , respectively.

smaller value of Ro number at high Ra numbers led to a reduction in the Nu number values (refer to figure 6.15) and this gradual drop in the value of Nu number ultimately appeared as a reduced scaling coefficient in the experiments [23]. Further, the comparison of Nu – Ra scaling law with the results of an incompressible solver using the Boussinesq approximations by Pitz et al. [27] and Jiang et al. [41] shows the prediction of reduced heat transfer rate. The value of scaling exponent in their studies was closer to $2/7$ compared to the $1/3$ observed in current study. The reason behind this is unclear.

In conclusion, the power exponent of the Nu – Ra scaling law for $Ra \geq 10^7$ obtained for different Ro number agrees well with the $1/3$ power law. Further, the stabilization of the plumes in the axial direction at low Ro number leads to a reduced heat transfer rate. Thus, the plumes play a crucial role in the heat-carrying effectiveness at different physical conditions. In the next section, the discussions are centred around understanding the role of plumes in the local heat transfer rate.

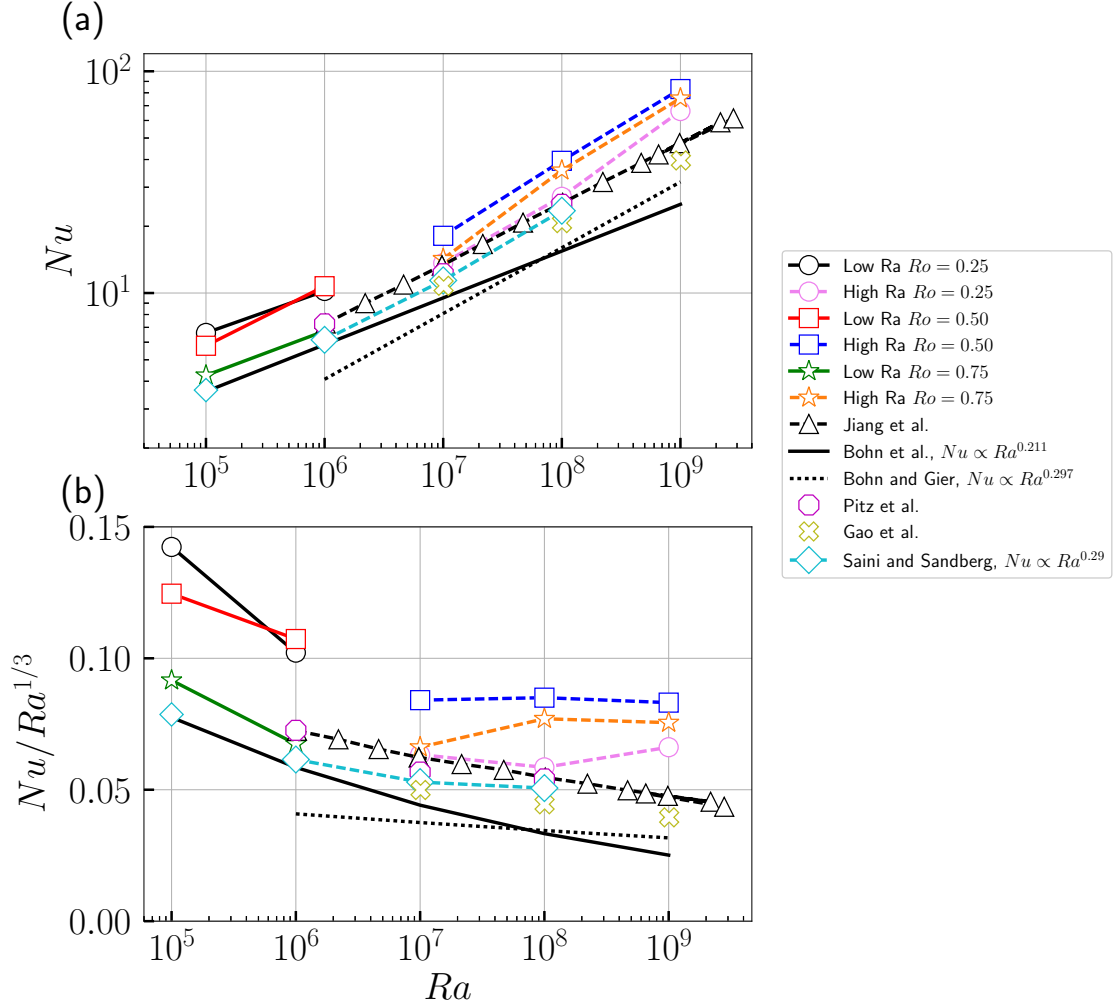


FIGURE 6.16: Comparison of Nu number scaling with Ra number with results reported by Bohn et al.[23], Pitz et al.[27], Gao et al.[29], Saini and Sandberg [58] and Jiang et al.[41].

6.6 The role of plumes in the heat transfer

The plumes emanate from the boundary layers due to the buoyancy and are responsible for the overall convection cycle [76] as they carry the hot and the cold fluid from the near-wall regions to the bulk of the flow. The plumes are responsible for feeding the large scale circulations in the bulk, which are accountable for thermal mixing for convection heat transfer to take place. In the Rayleigh-Bénard convection, Ahlers et al. [77] found that the temperature profiles in the near-wall region varied logarithmically with the wall-normal distance. They speculated that the origin of the logarithmic profile was due to the existence of near-wall

plumes and made an analogy with a logarithmic profile of velocity in the shear flows. A subsequent two-dimensional DNS study by Van Der Poel et al. [78] for the Rayleigh-Bénard convection at $Ra = 5 \times 10^{10}$ concluded that the logarithmic profiles exist only for the regions of the boundary layer where thermal plumes were ejected into the bulk. Thus, the presence of near-wall plumes make the temperature profiles in the near-wall region physically richer than the core region of the convection.

To scrutinize the role of plumes in the centrifugal buoyancy-induced flow, the conditional averaging of the flow field is carried out to isolate them from the rest of the flow field.

6.6.1 Conditional averaging

The conditional averaging is carried out such that the plumes are detected from the instantaneous flow field by following the criteria of Huang et al. [79]. Based on the first criterion, the plumes always have a higher (hot plume) or a lower (cold plume) values of temperature than the rest of the flow-field, which can be expressed as

$$\pm [T(z, \theta) - \langle T \rangle_{z, \theta}] > c \langle T_{rms} \rangle_{z, \theta}. \quad (6.16)$$

In equation 6.16, the plus sign is for a hot plume and the minus sign is for a cold plume. The second criterion is related to a flow field region which carries the heat flux in the wall-normal direction, and a condition for this is

$$(RaPr)^{1/2} u_r(z, \theta) T(z, \theta) > cNu. \quad (6.17)$$

Here, c is an empirical constant and a value of 0.8 [79] is used in this analysis. By using the criteria given in equations 6.16 and 6.17, the thermal plumes are detected from the instantaneous flow-field as shown in figure 6.17. The plume detection criteria are then applied to all time instants to carry out plume-to-plume conditional averaging.

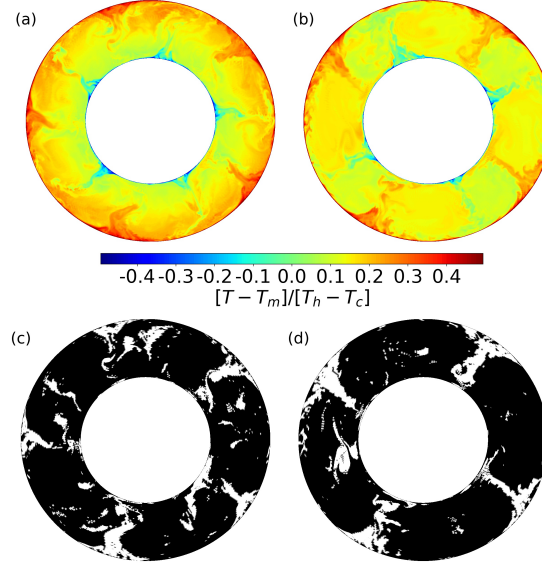


FIGURE 6.17: Instantaneous temperature contour of $Ra = 10^9$ for (a) $Ro = 0.25$ and (b) $Ro = 0.75$. The hot thermal plumes detected from the instantantaneous data shown in (a) and (b)

Figure 6.18 shows the conditionally averaged temperature contours for $Ra = 10^8$ and 10^9 at different Ro numbers. Distinct plume regions at the centre of each convection roller emerge in the averaged flow-field, which are widespread in the near-wall region. The different regions in the conditionally-averaged flow field are

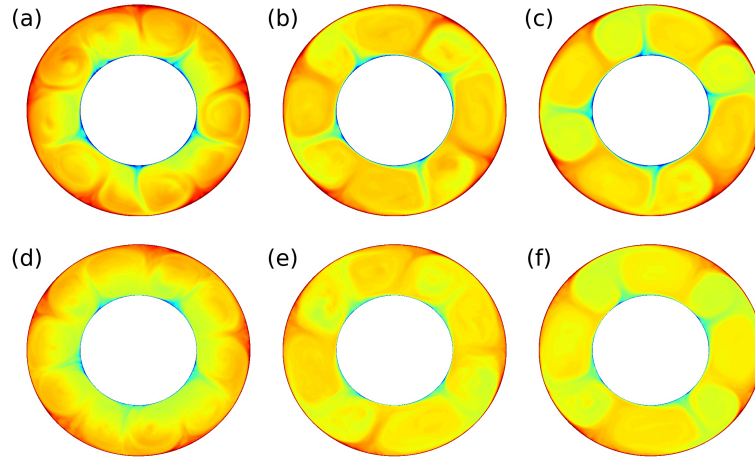


FIGURE 6.18: Conditionally-averaged contours of temperature for $Ra = 10^8$ at (a) $Ro = 0.25$ (contour levels $[1.0, 1.1575]$), (b) $Ro = 0.5$ (contour levels $[1.0, 1.630]$) and (c) $Ro = 0.75$ (contour levels $[1.0, 2.4178]$) and for $Ra = 10^9$ at (d) $Ro = 0.25$ (contour levels $[1.0, 1.1575]$), (e) $Ro = 0.5$ (contour levels $[1.0, 1.630]$) and (f) $Ro = 0.75$ (contour levels $[1.0, 2.4178]$).

highlighted in figure 6.19. The Region I corresponds to the azimuthal locations where the hot plumes rises, and flow is directed in the radial direction towards the

cold wall. Similarly, Region II in the flow-field represents the area of cold plumes emission. The rest of the flow-field on either side of the plumes is the “Wind” of the convection and is mainly responsible for thermal mixing in the bulk region.

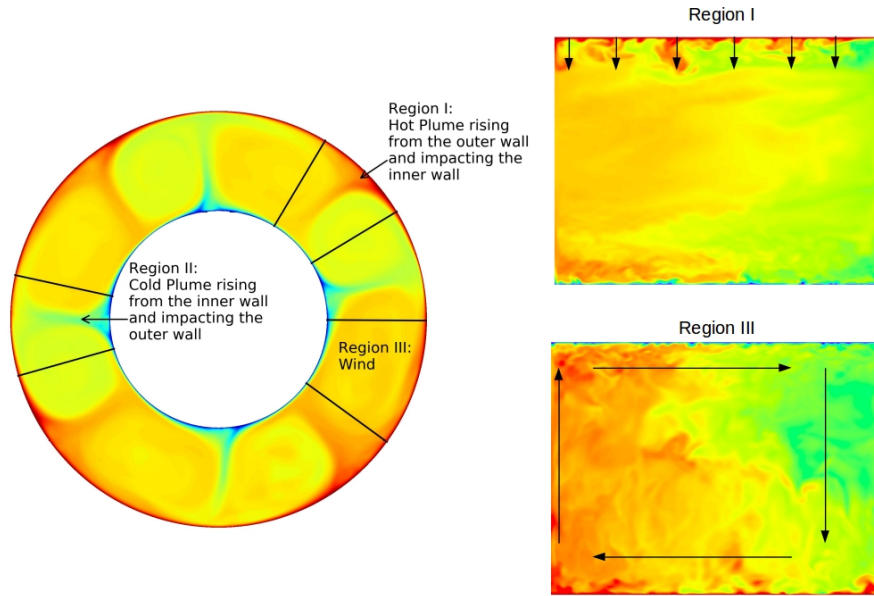


FIGURE 6.19: (a) A schematic of different regions identified in the flow which include Region I, where hot plume rises from the outer wall, Region II, where cold plume rises from the inner wall and Region III for the wind of the convection, (b) instantaneous contour of temperature in the Region I in the $r-z$ plane shows the near wall hot plumes rising and (c) shows the instantaneous contour of temperature in the Region III in the $r-z$ plane shows the large scale circulation or convection wind.

The focus in this analysis is Region I to understand the role of plumes ejected from the hot wall on the wall heat transfer rate. Two approaches are followed to isolate these regions from the conditionally averaged flow-field. In the first approach, the time-averaged temperature field of the plumes detected by equations 6.16 and 6.17 are used to distinguish the hot plume ejection regions as shown in figures 6.20 (a) and 6.20 (b). The near-wall plume regions at the edge of the thermal boundary layer are represented by the peaks in the temperature field as shown in figure 6.20 (c). Another approach is by defining the local Richardson number (Ri), which is often used to distinguish the natural convection dominated regions from the forced convection. The basic idea here is to isolate the buoyancy dominated

region (plumes) from the shear dominated region (wind). The definition of a local Ri number is

$$Ri = \frac{[\Omega^2 r][\langle T \rangle_t - \langle T \rangle_{t,\theta}]}{\langle u_\theta^2 \rangle_t}. \quad (6.18)$$

It has a large value in the plume region where the temperature gradients are large and a small value in the high shear regions. The outcome of both the approaches is strongly correlated and provides the same plume-dominated regions in the flow field, as shown in figure 6.20(c).

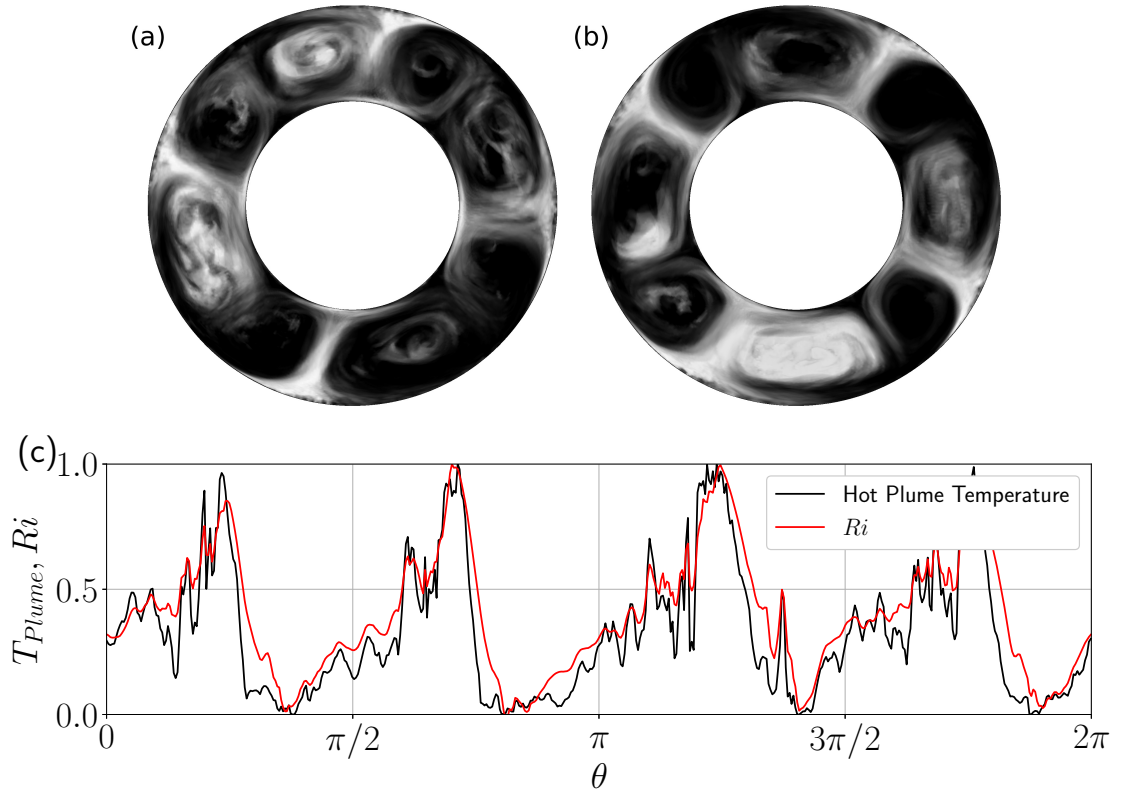


FIGURE 6.20: The averaged plumes temperature for $Ra = 10^9$ at (a) $Ro = 0.5$ and (b) $Ro = 0.75$. The comparison of plume-dominated regions at the edge of thermal boundary layer thickness predicted by the plume temperature and Ri number values.

To understand the role of plumes, the temperature profiles, spatially averaged over the plume-dominated regions, are extracted. In figure 6.21, temperature profiles are scaled with total temperature difference ($T_H - T_C$) and plotted as a function of radial distance scaled with thermal boundary layer thickness at the hot wall. The solid lines show the spatially averaged profiles over the plume-ejection regions and dashed lines shows the profiles averaged over 2π circumferences of the hot wall.

The results show a linear variation of temperature in the near-wall region in the thermal boundary layer (up to $r/\lambda_{th}^\theta = 1.0$). However, in the bulk region outside the thermal boundary layer, the profiles averaged over the plume-ejection region show the logarithmic variation over the decade long wall-normal distance which can be represented as

$$\frac{\langle T \rangle_t - T_m}{\Delta T} = A \log\left(\frac{r}{\lambda_{th}^\theta}\right) + B. \quad (6.19)$$

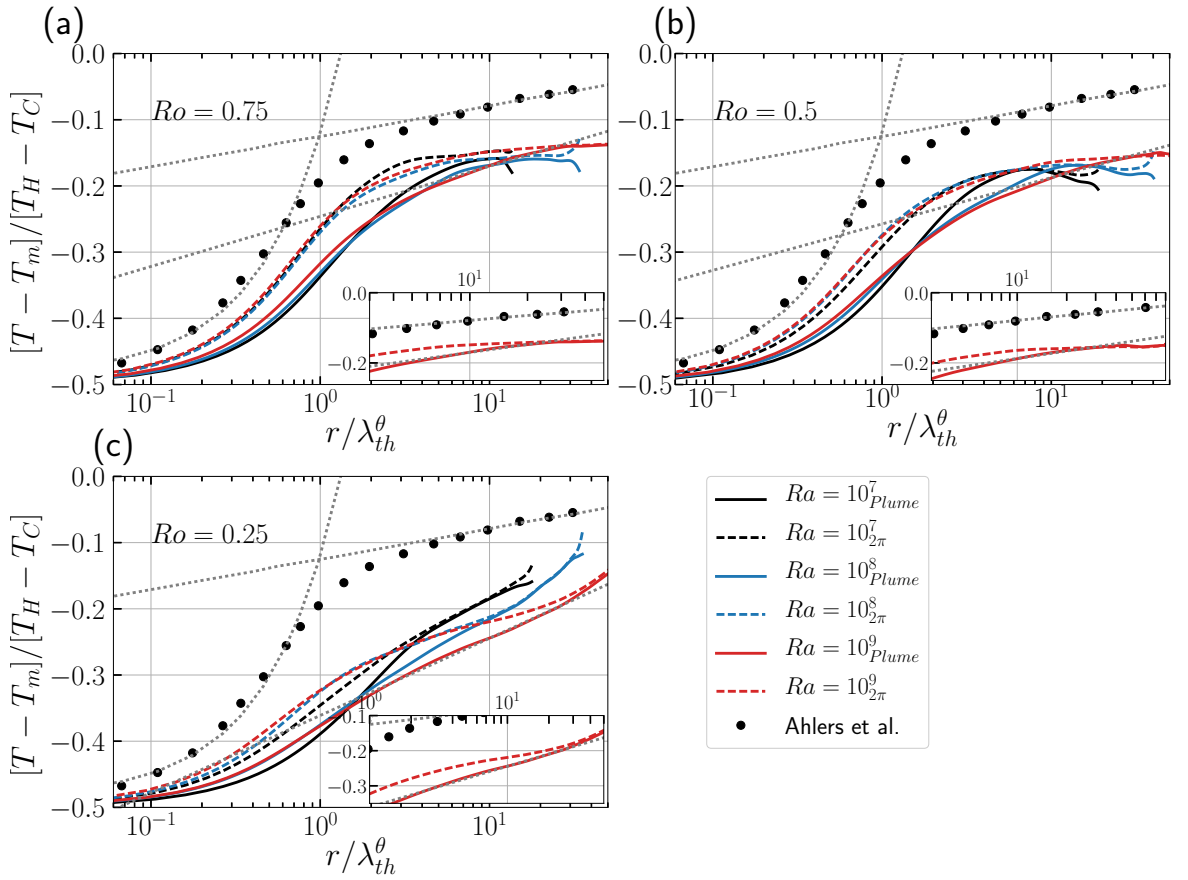


FIGURE 6.21: The comparison of time averaged profiles for $[\langle T \rangle_t - T_m]/[\Delta T]$ as a function of wall normal distance r/λ_{th}^θ for different Ra number at (a) $Ro = 0.75$, (b) $Ro = 0.5$ and (c) $Ro = 0.25$. Solid circles show the DNS temperature profile from Ahlers et al. [80] at $Ra = 2 \times 10^{12}$.

For a comparison the temperature profile from Ahlers et al. [80] at $Ra = 2 \times 10^{12}$ is also plotted in figure 6.21. The comparison shows a resemblance of the logarithmic region for $Ra = 10^9$ at $Ro = 0.75$ for temperature profiles averaged over the plume-ejection region with the temperature profile of Ahlers et al. [80] (refer to inset plot

in figure 6.21(a)). This implies that the near-wall bulk region approaches the fully turbulent state in the plume-emission areas. The slope of the logarithmic region (A in equation 6.19) for $Ra = 10^9$ at $Ro = 0.75$ case is found to be 1.6 times higher than the profile of Ahlers et al. [80]. The trend of an increase in the slope of the logarithmic region at $Ra = 10^9$ compared to $Ra = 2 \times 10^{12}$ is consistent with the findings of Ahlers et al. [80]. Further, for $Ra = 10^8$, there is a small wall-normal region over which the profiles resembles the logarithmic behaviour, which completely disappears for $Ra = 10^7$. With the decrease in the value of Ro number to 0.5, the logarithmic behaviour of temperature profiles persists with a slope almost similar to $Ro = 0.75$ case, as shown in figure 6.21(b). However, for both the Ro numbers of 0.5 and 0.75, the profiles averaged over the full 2π circumference deviate from the logarithmic behaviour as shown in the inset plot of figures 6.21(a) and 6.21(b). In the case of $Ro = 0.25$, the linear variation of temperature in the thermal boundary layers extends into the bulk region as well, without any logarithmic variation with the wall-normal distance, as observed for the other Ro number cases. The reason for this behaviour at $Ro = 0.25$ is due to the temperature gradient in the core that exists for all Ra numbers. The non-isothermal conditions in the core lead to a gradual variation in the temperature in the near-wall bulk region, which connects the thermal boundary layer with the convection core. Thus, the linear variation of temperature extends from the boundary layer to the convection core for the $Ro = 0.25$ case. In contrast, for the other Ro number cases, the isothermal conditions in the core region of the convection require the steep variation of the temperature in the near-wall bulk region and thus the logarithmic temperature profile appears.

Further, the variation of root-mean-square of temperature fluctuations is also consistent with the logarithmic variation with the wall-normal distance, as shown in figure 6.22. The logarithmic variation is more pronounced for Ro number of 0.5 and 0.75, which is consistent with the earlier observation for logarithmic temperature profiles. For $Ra = 10^9$, the logarithmic dependence extends over a considerable wall-normal distance compared to the low Ra number cases. In the case

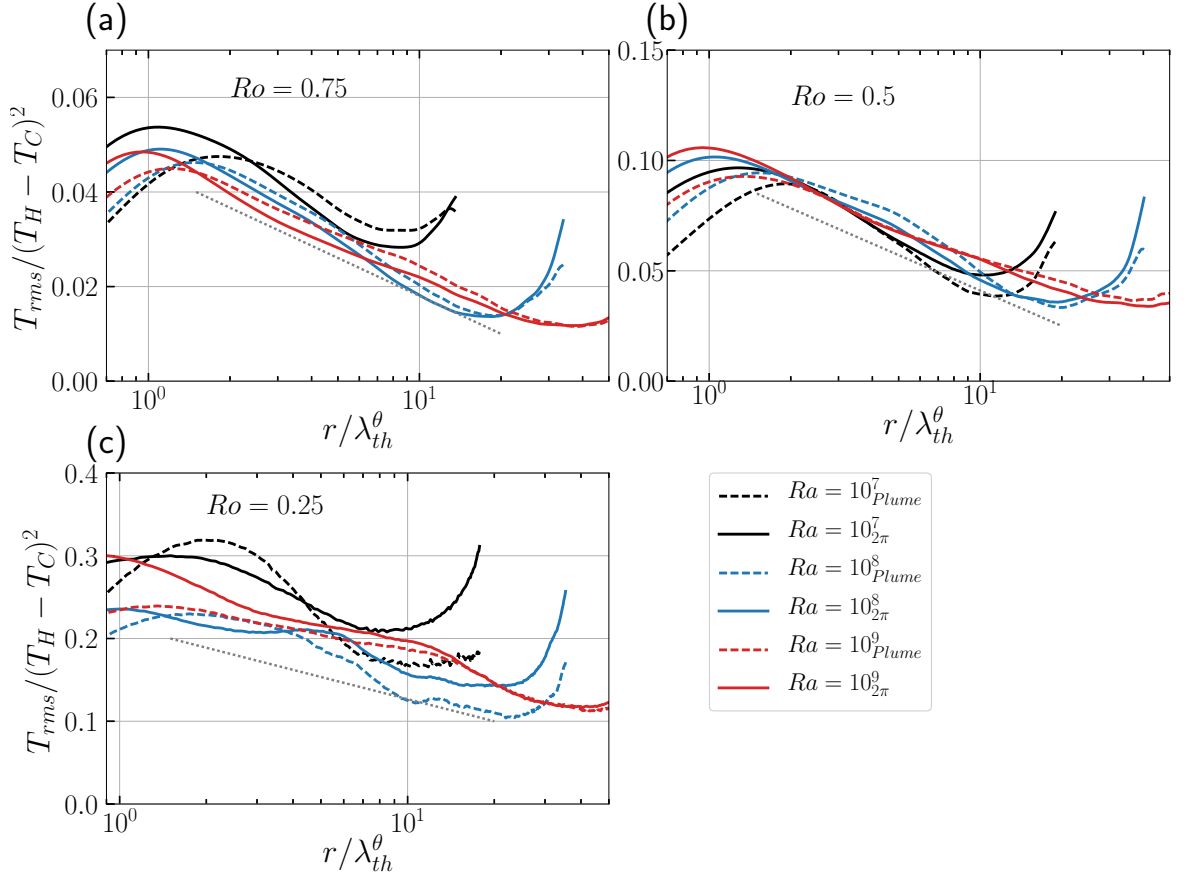


FIGURE 6.22: The variation of root-mean-square of temperature fluctuations as a function of wall normal distance r/λ_{th} for different Ra number at (a) $Ro = 0.75$, (b) $Ro = 0.5$ and (c) $Ro = 0.25$.

of $Ro = 0.25$, larger deviations from the logarithmic behaviour are observed as shown in figure 6.22(c).

The local turbulent boundary layer in the plume-ejection regions as corroborated by the logarithmic dependence of mean temperature, as well as the root-mean-square of its fluctuations with the wall-normal distance, has a direct influence on the wall heat transfer rate. Figure 6.23 shows the comparison of the $Nu - Ra$ scaling for the plume ejection regions with the Nu number values calculated over the full 2π domain. The results show an increase in the scaling coefficient to 0.379 for the plume ejection regions as compared to the value of $1/3$ when averaged over the full 2π domain. Further, the Ro number appears to have a minimal effect on the scaling of Nu number with the Ra number for plume ejection regions. In conclusion, the local turbulent spots in the plume ejection regions lead to an

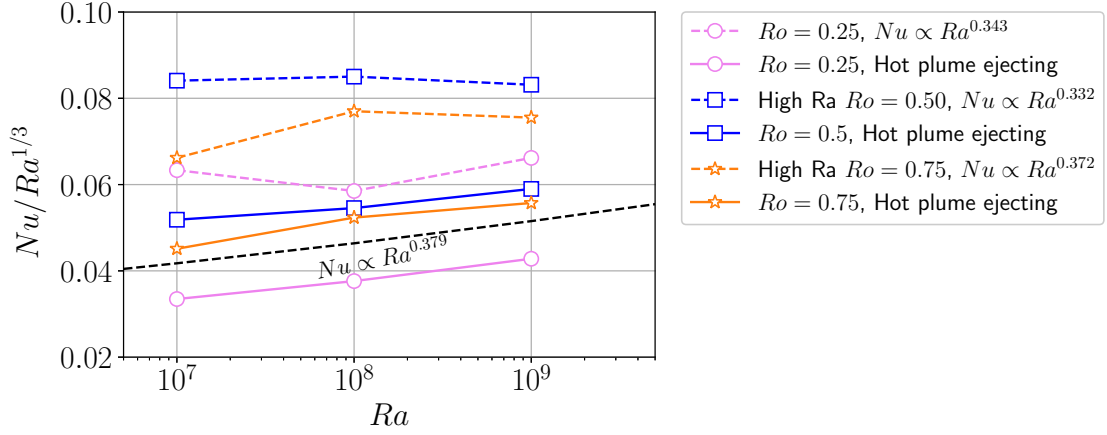


FIGURE 6.23: Comparison of Nu number scaling with Ra number for plume-ejection region with the full 2π averaged values

increase in the Nu – Ra scaling coefficient, and this could be the transition phase of convection to achieve the ultimate state [22], where $Nu \propto Ra^{1/2}$ and becomes independent from the fluid properties.

6.7 Conclusions

The effect of Rossby number in the centrifugal buoyancy-induced flow was studied for Rayleigh numbers in the range $10^5 \leq Ra \leq 10^9$. Direct numerical simulations were carried out for three Rossby numbers, $Ro = 0.25, 0.5$ and 0.75 at a fixed flow Mach number. A quasi-two-dimensional state in the core region of the flow was observed for the low Ro number case. The reason for this was the balance between the Coriolis force, centrifugal force and the pressure gradient in the convection core. In this state, the convection plumes appeared to have a strong coherence in the axial direction compared to the higher Ro number cases. As a consequence of this, the heat transfer rate reduced with the decrease in the value of Ro number.

The scaling of Nu number with the Ra number was found to be in consensus with the $1/3$ scaling law of classical Rayleigh–Bénard convection for $Ra \geq 10^7$. For the low Ra number cases, the scaling exponent for Nu – Ra power law was close to or less than $2/7$. A consistent value of the scaling exponent of Nu number with Ra number for all Ro number cases provided evidence that the reduced

scaling coefficient obtained in the experiment [23] was due to the low values of Ro at high Ra . Further, for plume-ejection regions, a power exponent of 0.379 was observed. The reason for this local heat transfer rate enhancement were the turbulent thermal boundary layers forming in the plume ejection regions. At these local turbulent spots, the mean temperature, as well as root-mean-square of its fluctuations, varied logarithmically with the wall-normal distance in the near-wall bulk region. The behaviour resembled the logarithmic velocity profiles also found for turbulent shear flows and reconciled the transition to the ultimate regime of convection.

Chapter 7

Buoyancy-induced flow in a cavity with axial-bore flow

The focus of this chapter is to understand the effect of Rayleigh number on a high Rossby number flow in a high-pressure compressor (HPC) inter-disk cavity. These cavities form between the compressor disks of a gas turbine engine, and they are an integral part of the internal air cooling system as discussed in chapter 1. Highly resolved large-eddy simulations are performed for two Rayleigh numbers of 0.76×10^8 and 1.54×10^8 at a fixed Rossby number of 4.5, which match the experimental conditions of Atkins and Kanjirakkad [49].

The contents of this chapter are from the proceedings of the ASME Turbomachinery Technical Conference and Exposition 2020 (Saini and Sandberg [81]), which has also been accepted for publication in the Journal of Turbomachinery.

7.1 Problem Description and Numerical Setup

The computational domain is an annular rotating cavity with an axial bore flow (concentric pipe flow region) that rotates at a constant angular velocity of Ω_∞ about its axis. It comprises of five blocks shown in figure 7.1. The inner and outer radius of the bore flow region is r_s , r_i respectively and r_o is the radius of

the shroud surface. In the axial direction, z_{in} , z_{out} and z_m are the domain lengths of the bore flow inlet, outlet and middle sections, respectively, shown in figure 7.1. The outer cavity is enclosed by two side disks, spaced H apart. A radially increasing temperature profile is prescribed on the side disks extracted from the experiments by Atkins and Kanjirakkad [49], as shown in figure 7.2 (a) and also provided in the appendix B.

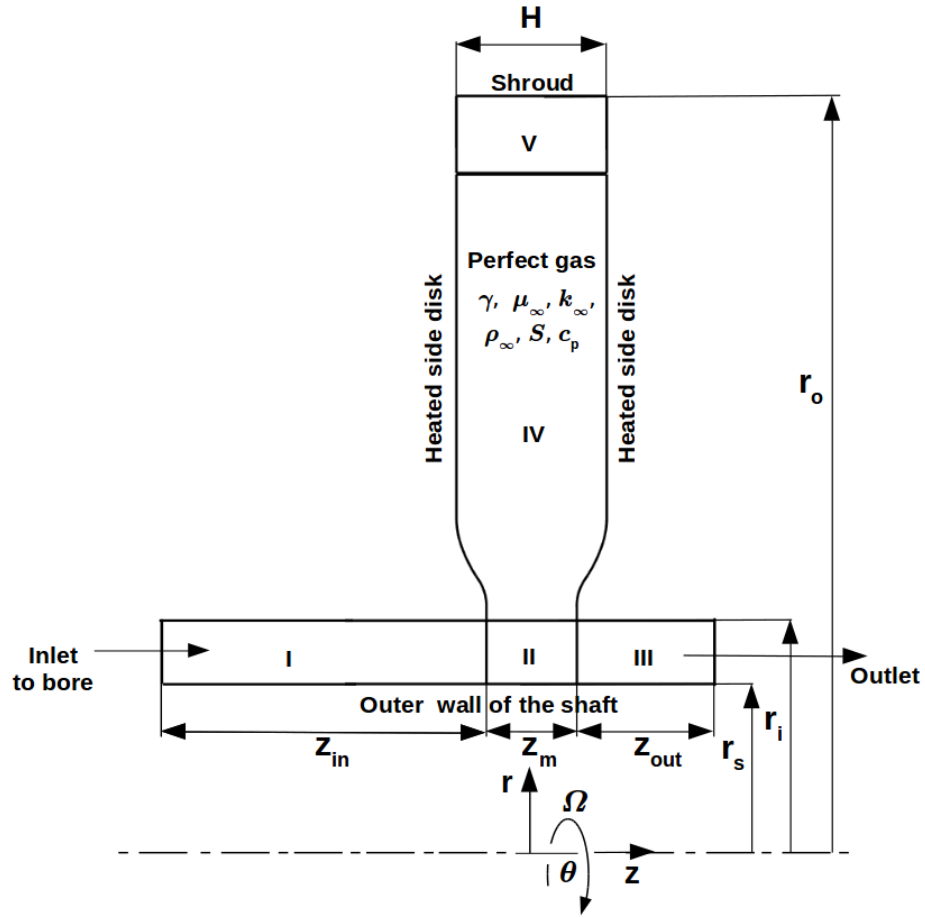


FIGURE 7.1: Schematic of inter-disk cavity. The walls of the cavity undergo solid-body rotation about the z -axis except the shaft, which is held stationary to match the conditions of [49].

The temperature is maximum at the shroud (T_{sh}) and minimum at the outer wall of the shaft (T_s), which is considered as the reference temperature ($T_\infty = T_s$). The outer wall of the bore flow is at temperature $T_i > T_s$. The cavity is filled with a perfect gas with specific heat at constant pressure c_p and ratio of specific heats γ . The details about the governing equations can be found in the chapter 3. The

non-dimensional numbers that completely specify this problem are given as

$$\begin{aligned}
 Re_\infty &\equiv \frac{\rho_\infty(\Omega_\infty r_o)r_o}{\mu_\infty}, & Re_z &\equiv \frac{\rho_\infty U_{bulk}(2(r_i - r_s))}{\mu_\infty}, & \frac{S}{T_\infty} &= 0.37, \\
 Pr_\infty &\equiv \frac{\mu_\infty c_p}{k_\infty} = 0.71, & \frac{\Delta T}{T_\infty} &\equiv \frac{T_{sh} - T_s}{T_s}, & \gamma &= 1.4, \\
 Ma_\infty &\equiv \frac{\Omega_\infty r_o}{(c_p(\gamma - 1)T_\infty)^{1/2}} = 0.3, & \frac{r_s}{r_o} &= 0.2363, & \frac{r_i}{r_o} &= 0.318, \\
 \frac{H}{r_o} &= 0.195, & \frac{z_{in}}{r_o} &= 0.82, & \frac{z_m}{r_o} &= 0.117, & \frac{z_{out}}{r_o} &= 0.305
 \end{aligned} \tag{7.1}$$

where $Re_\infty (= Re_\theta)$, Pr_∞ and Ma_∞ are the reference Reynolds number, Prandtl number and Mach number, respectively. The axial Reynolds number Re_z is defined based on the bulk velocity (U_{bulk}) at the inlet, specified as the boundary condition. A linear temperature profile is specified at the inlet varying from the shaft temperature T_∞ to the temperature at the outer wall of bore flow T_i as provided by Atkins and Kanjirakkad [49]. To further define the relative strength of the axial inertial force with respect to the rotational inertial force, the Rossby number is defined as

$$Ro \equiv \frac{U_{bulk}}{\Omega_\infty r_o} = \frac{r_o^2}{2r_i(r_i - r_s)} \frac{Re_z}{Re_\theta}. \tag{7.2}$$

It is important to define the Rayleigh number to quantify the effect of centrifugal induced buoyancy force relative to the viscous force in the flow as

$$\begin{aligned}
 Ra &\equiv \frac{\rho_\infty^2 c_p [\Omega_\infty^2 (r_o + r_i)/2] (\Delta T/T_\infty) (r_o - r_i)^3}{\mu_\infty k_\infty} \\
 &= Re_\theta^2 Pr_\infty \frac{\Delta T}{T_\infty} \frac{1}{2} \left(1 + \frac{r_i}{r_o}\right) \left(1 - \frac{r_i}{r_o}\right)^3.
 \end{aligned} \tag{7.3}$$

The response of the system is the local Nusselt number on the side disks, which is the non-dimensional form of the wall heat transfer coefficient defined as

$$Nu_r = \frac{r}{T_w - T_{ref}} \left[-\frac{d\bar{T}}{dz} \right]_w. \tag{7.4}$$

In cylindrical coordinates, u , v and w represents the non-dimensional velocity components in the z , r and θ directions, respectively. The boundary conditions for

this problem are thus $u = v = 0$ for all the walls of the cavity. The inner wall of the bore flow region is stationary, $w(z, r = r_s/r_o, \theta) = 0$, and the outer wall is rotating at constant angular velocity ($\Omega = 1.0$), $w(z, r = r_i/r_o, \theta) = \Omega r_i/r_o$. The rotating wall boundary condition for the side disks and shroud is $w(z = \pm H/2, r, \theta) = \Omega r$, $w(z, r = 1, \theta) = 1$, respectively. In this study, one cavity is considered by ignoring the effect of other cavities onto the bore flow conditions. Thus, it is more sensible to have fully developed turbulent flow conditions in the bore flow at the cavity inlet rather than laminar conditions. For that, a precursor periodic concentric pipe flow calculation (Block I in figure 7.1) is carried out at axial Reynolds number of $Re_z = 18,405$ to obtain fully developed flow profiles. The turbulence at the bore flow inlet is then generated by a digital filter technique (Touber and Sandham [82]), which uses the mean velocity and Reynolds stress profiles extracted from the precursor simulation and then superposes the fluctuations onto the mean profiles to generate turbulent inlet conditions. The axial domain size of the bore flow inlet $z_{in}/r_0 = 0.82$ is found to be sufficient to recover the fully developed turbulent velocity profile and turbulent kinetic energy from the periodic precursor simulation, as shown in figures 7.2 (b) and 7.2 (c). The axial domain size of the bore flow outlet (Block III) is $z_{in}/r_0 = 0.305$, evaluated by running different length sizes, which was enough to nullify the effect of the outlet boundary condition on the cavity. To further avoid any disturbance from the outlet boundary to move upstream, the zonal non-reflecting characteristic boundary condition (Sandberg and Sandham [83]) is used over the last $z/r_o = 0.128$ of the outlet block.

The details about the numerical schemes are provided in the chapter 3.

7.2 Results and Discussion

Large eddy simulations have been carried out for two Rayleigh numbers matching the experimental conditions of Atkins and Kanjirakkad [49], as shown in table 7.1. The number of grid points for each block is also reported in table 7.1. The techniques used to generate the grid is provided in Appendix C.

TABLE 7.1: Simulation Parameters

Non-Dimensional numbers	Case I	Case II
Re_θ	77,900	77,900
Re_z	18,405	18,405
Ma_∞	0.3	0.3
S/T_∞	0.37159	0.37159
Ro	4.50	4.50
Ra	0.76×10^8	1.54×10^8
$\Delta T/T_\infty$	0.0846	0.1712
p_{out}	8.187	8.196
Grid Size	N_r, N_z, N_θ	N_r, N_z, N_θ
Block I	100, 162, 514	100, 162, 514
Block II	100, 104, 514	100, 104, 514
Block III	100, 66, 514	100, 66, 514
Block IV	192, 104, 514	208, 104, 514
Block V	108, 104, 514	152, 104, 514
Dimensional quantities to match the conditions of [49]	Case I	Case II
Bore mass flow rate (kg/s)	0.0672	0.0672
RPM	199.0	199.0
$T_\infty (= T_s)(K)$	297.1	297.1
$T_i(K)$	298.99	303.26
$r_o(m)$	0.22	0.22

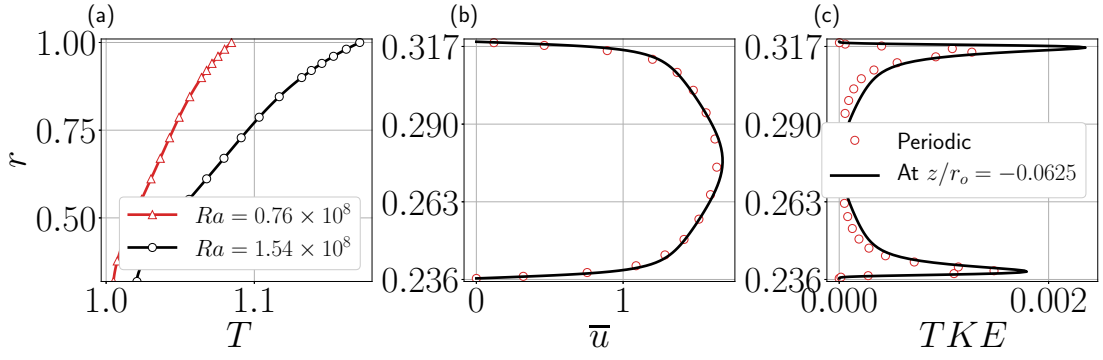


FIGURE 7.2: (a) Temperature profiles for the side disks from [49], Comparison of fully developed (b) mean axial velocity profile and (c) turbulent kinetic energy (TKE) in the bore region obtained by digital filter inflow at the downstream location $z/r_o = -0.0625$ from the inlet with the profiles from precursor periodic simulation.

7.2.1 Grid quality

The standard WALE model as discussed in chapter 3 is used to account for the sub-grid stresses. To assess the influence of the LES model onto the solution, the sub-grid scale viscosity μ_{sgs} is compared with the molecular viscosity as shown in figure 7.3. It is less than 1 in most of the cavity except in the small region near the cob, where the maximum value is close 1.3 for both Ra numbers. For $Ra = 1.54 \times 10^8$, the region with the maximum value of μ_{sgs}/μ has become larger compared to the low Ra number case, but the maximum value remains the same. To further evaluate the grid quality, the sub-grid scale viscosity μ_{sgs} is also compared with the turbulent eddy viscosity for both cases. The comparison shows that the current grid resolution resolves 99% of the turbulent energy.

The grid-spacings in wall units for the shroud, upstream disk and downstream disk are shown in figure 7.4. For both Ra numbers, the wall-normal spacing Δr^+ is less than 0.05, and the grid is stretched in the axial direction with maximum grid-spacing $\Delta z^+ < 2$ at the shroud. In the azimuthal direction, $r\Delta\theta^+$ is less than 5 for both Ra numbers at the shroud. Similarly, for the side disks, the wall-normal resolution Δz^+ is in most regions less than 0.5 with a maximum value close to 0.7 at the inner region of the downstream disk (figure 7.4 (h)), where the axial bore flow enters the cavity. The grid is stretched in the radial direction at the side disk

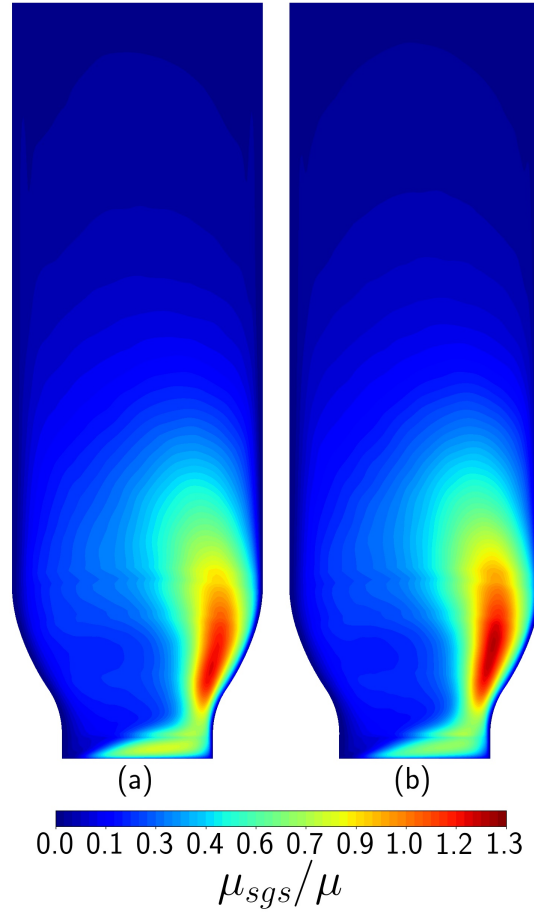


FIGURE 7.3: Comparison of sub-grid scale viscosity with molecular viscosity for (a) $Ra = 0.76 \times 10^8$ and (b) $Ra = 1.54 \times 10^8$.

with a maximum $\Delta r^+ < 10$ in the middle of the cavity at the downstream disk. The grid resolution in the azimuthal direction $r\Delta\theta^+$ has a maximum value close to 20 near to the cavity inlet at the downstream disk.

To further ascertain the adequacy of the grid resolution in the azimuthal direction to resolve the relevant turbulent scales, the wavenumber spectra of radial velocity at different radial locations are plotted in figure 7.5. The spectra follow the 5/3 power law in the inertial range and a decay of about four orders of magnitude is observed in the amplitude of energy at $r = 0.4$ and 0.5 for both the Ra numbers. These two numerical probes are placed in the toroidal vortex region, where the overall energy level is high due to the breakdown of the large structures to small scales. However, the decay in the amplitude of energy is increased to six orders of magnitude for the rest of the points in the cavity, where mostly large scale

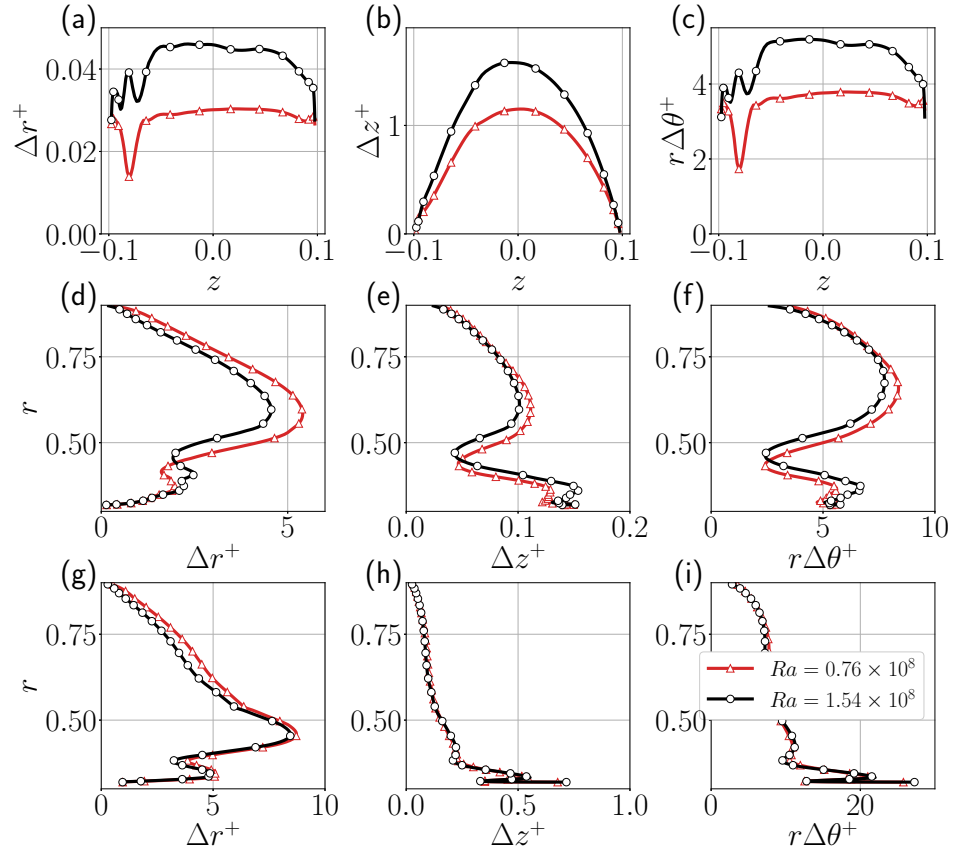


FIGURE 7.4: Grid-spacing in wall units for the shroud (a – b), upstream disk (d – f) and downstream disk (g – i) at different Rayleigh numbers.

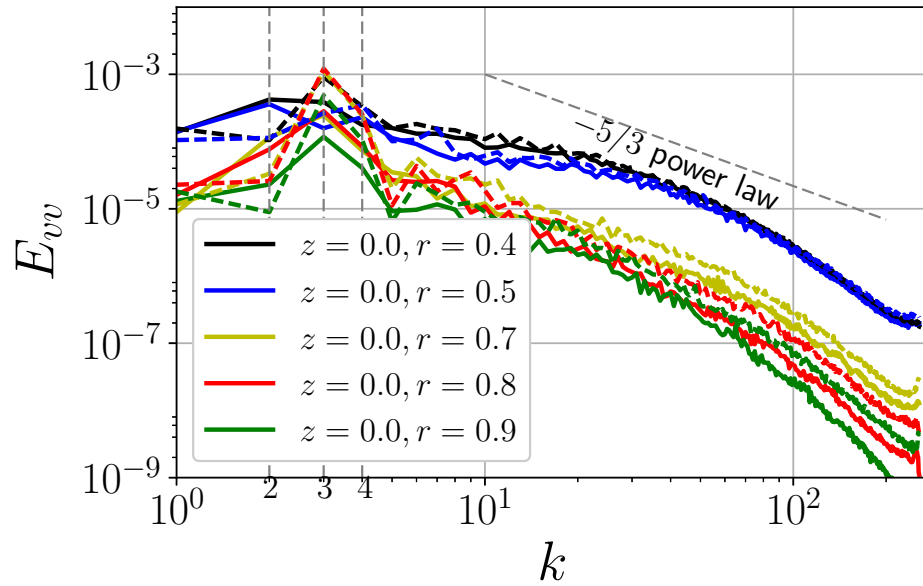


FIGURE 7.5: Wavenumber spectra of different monitor points located in the cavity at different Rayleigh numbers (— $Ra = 0.76 \times 10^8$, --- $Ra = 1.54 \times 10^8$).

structures are present. The values for all these metrics give us high confidence that the simulations are adequately resolved with the current grid resolution in all directions and that the results are reliable.

A non-dimensional time step of 1×10^{-5} was used for both the cases to satisfy the Courant-Friedrich-Lewy (CFL) stability condition with maximum CFL number of 0.71. It took approximately 90 hours on 1,152 and 1,440 CPU cores for one rotation of the cavity for $Ra = 0.76 \times 10^8$ and 1.54×10^8 , respectively. For each case, the statistics were collected for approximately 12 rotations once the flow reached a statistically steady-state.

7.3 Flow Structure

7.3.1 Instantaneous Flow Field

In figure 7.6, contours of instantaneous axial velocity illustrate the fully developed turbulent bore flow is entering into the cavity and forming a shear layer at the interface between the bore flow and the outer cavity. The axial bore flow expands into the cavity to form a strong toroidal vortex in the inner region of the cavity shown in figure 7.7. This toroidal vortex further bifurcates to radial flares in the outer region of the cavity. These radial flares form cyclonic and anti-cyclonic vortices, which rotate in the direction of rotation and in the opposite direction of rotation of the cavity, respectively. These vortices are responsible for transferring cold fluid from the inner region and hot fluid from the outer region of the cavity. Further, near the shroud, small scale vortices can be observed in figure 7.7, which corresponds to the presence of local convection cells analogous to Rayleigh–Bénard convection plumes arising due to the centrifugal buoyancy.

Figures 7.8 (a) and 7.8 (b) show the instantaneous contours of temperature at the mid-axial position of the cavity. The bore flow conditions are the same in both cases. The small scale turbulent structures can be observed in the toroidal vortex region for both cases. However, in the outer region, large scale coherent structures

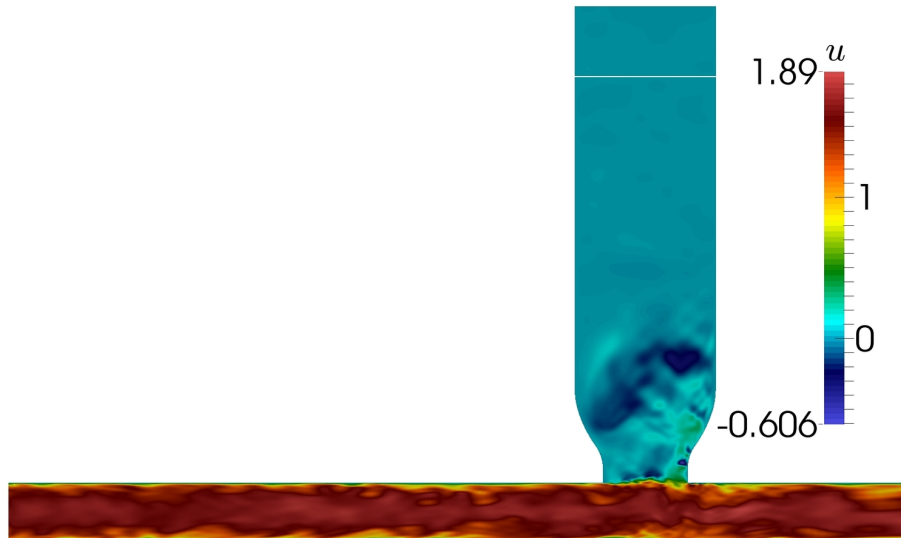


FIGURE 7.6: Instantaneous axial velocity contour for the cavity showing the turbulent bore flow interaction with the cavity.

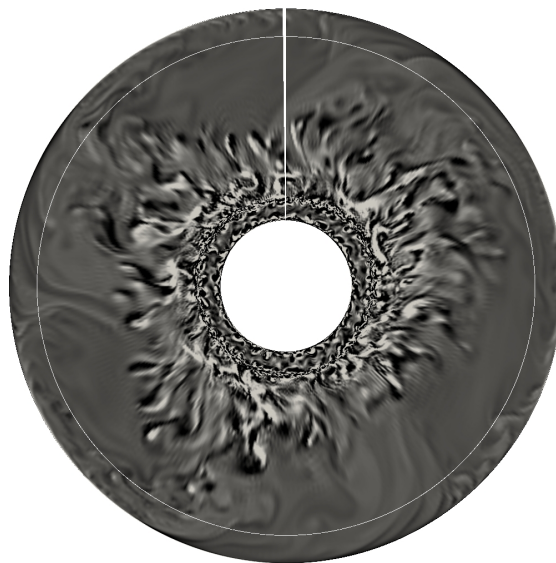


FIGURE 7.7: Instantaneous contour for circumferential vorticity in the cavity for $Ra = 1.54 \times 10^8$.

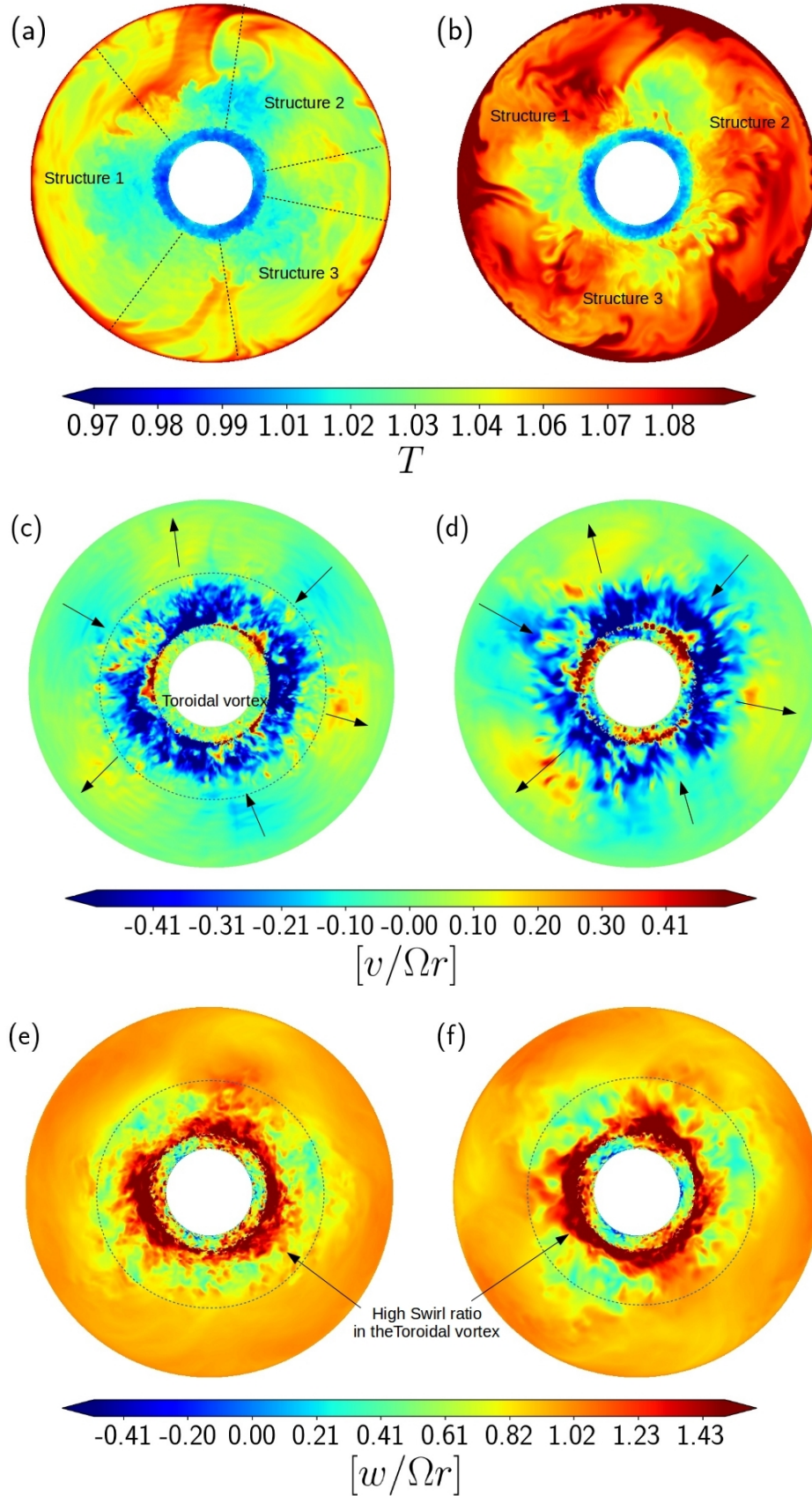


FIGURE 7.8: Comparison of instantaneous contours of temperature at (a) $Ra = 0.76 \times 10^8$ and (b) $Ra = 1.54 \times 10^8$, radial velocity scaled with local disk speed at (c) $Ra = 0.76 \times 10^8$ and (d) $Ra = 1.54 \times 10^8$ and azimuthal velocity scaled with local disk speed at (e) $Ra = 0.76 \times 10^8$ and (f) $Ra = 1.54 \times 10^8$ at mid-axial position of the cavity rotating in the clockwise direction.

are present. There are three dominant structures in the outer region for both the Ra numbers. In figure 7.8 (a), there is one large structure that spans from the 7'o clock to the 11'o clock positions. A second flow structure extends from the 1'o clock to approximately the 3'o clock position and is followed by a third dominant structure in the outer region. Similarly, three coherent structures can be seen for the high Ra number as well in figure 7.8 (b). For high Ra number cases, the breakdown of the large scale structures to small scales can be observed due to the strong centrifugal buoyant force. Further, to quantify the number of large scale structures, wavenumber spectra can be referred to, shown in figure 7.5. The spectra exhibit a peak in the energy at a wavenumber $k = 3$ for all the monitor points beyond $r = 0.5$, which confirms the earlier visual observation made from the instantaneous field. Furthermore, small scale convection plumes can be seen near the shroud region at the 10'o clock position in figure 7.8 (a) and 12'o clock and 3'o clock positions in figure 7.8 (b), respectively.

The instantaneous contours for radial velocity and azimuthal velocity scaled with local disk speed are shown in figures 7.8 (c,d) and 7.8 (e,f), respectively. At most of the radial locations, in the toroidal vortex region, the radial velocity is negative. Thus, at the mid-axial location, the flow is directed radially inwards from the cavity. In the outer region, at the three azimuthal locations approximately at 4'o clock, 8'o clock and 11'o clock, the radial velocity is positive and in-between these locations radial velocity is negative for both the cases. These three dominant radial flares of positive and negative velocity also correspond to the peak observed in the wavenumber spectra in figure 7.5. Another quantity of interest is the *swirl ratio*, defined as the ratio of the azimuthal velocity of fluid particles to local disk speed. The swirl ratio contours shown in figure 7.8 (e,f) show the comparison between the two Ra numbers. In the toroidal vortex region, the local intensity of swirl is higher for the high Ra number case. However, beyond that toroidal vortex region, the values of the swirl ratio reduce to less than one. Thus, fluid particles are rotating slower than the local disk speed and the intensity of this drop increases at high Ra number.

7.3.2 Mean Statistics

The instantaneous flow field revealed some basic flow features inside the cavity and qualitative comparison between the two Ra numbers. To further elucidate the flow structure and provide quantitative comparison to study the effect of Ra number, mean statistics are discussed in this section. Figure 7.9 shows the velocity profiles averaged over time and over the circumferential direction at the mid-axial position of the cavity for different Ra numbers. The velocity components are scaled with local disk speed. To understand the mean flow structure, the cavity is divided into four different regions based on the different physical phenomena observed. The Region I corresponds to an interface between the bore flow and the cavity. A strong shear layer is formed in this region, as shown by the axial velocity profiles in figure 7.9 (a). There is no perceivable effect of Ra number on the velocity profiles in this region as it is dominated by the bore flow conditions, which are the same for both the cases.

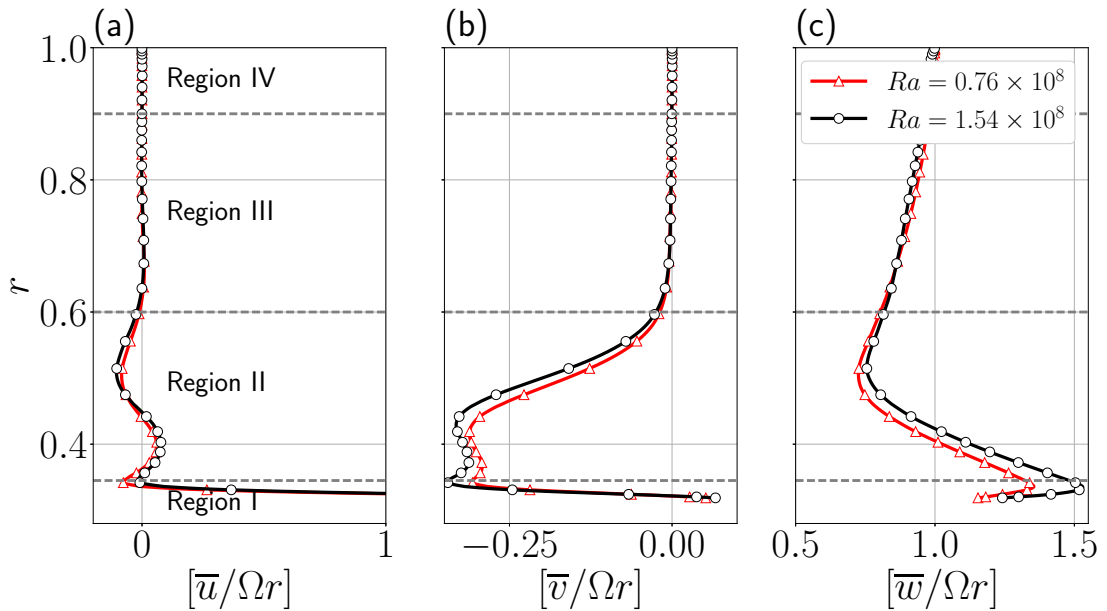


FIGURE 7.9: Comparison of time and circumferentially averaged velocity profiles at the mid-axial position of the cavity at different Rayleigh numbers.

The bore flow expands into the cavity to form a toroidal vortex in the Region II. The signature of the toroidal vortex produces the S-shaped profile of the axial velocity in the Region II shown in figure 7.9 (a), which signifies the flow recirculation

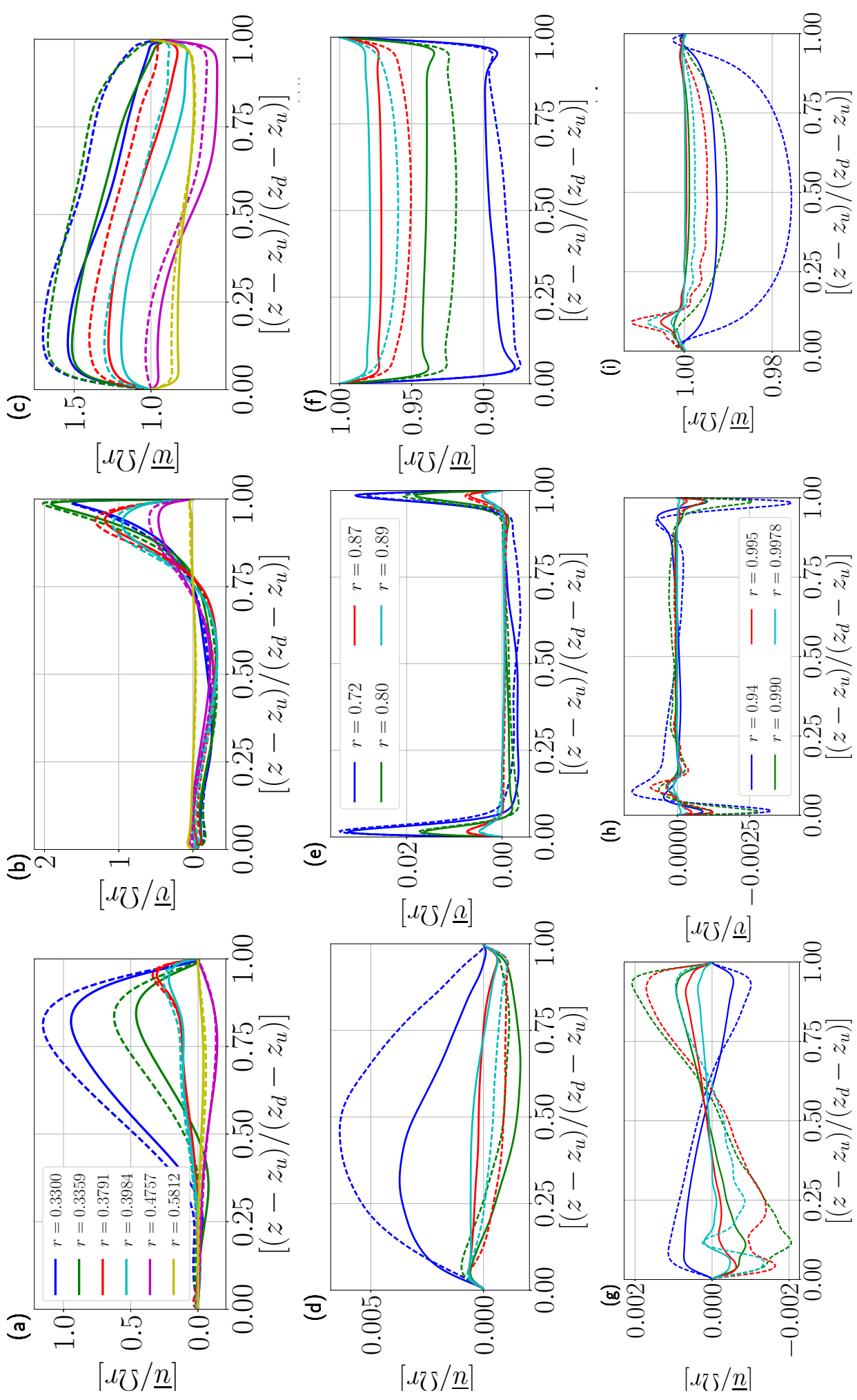


FIGURE 7.10: Comparison of time and circumferentially averaged velocity profiles at the different radial locations of the cavity in Region II (a-c), Region III (d-f) and Region IV (g-i) for different Rayleigh numbers (— $Ra = 0.76 \times 10^8$, --- $Ra = 1.54 \times 10^8$).

due to the vortex ring. There is a subtle change in the magnitude of axial velocity for the higher Ra number case as shown in figure 7.9 (a). However, to analyze the effect of Ra number, the variation of axial velocity in the axial direction of the cavity is provided in figure 7.10 (a). The peak of the axial velocity away from the centre of the cavity is the evidence for the asymmetric shape of the toroidal vortex. An overall increase in the axial velocity is observed for the higher Ra number case. Further, the S-shape profile of axial velocity at the mid-axial position in figure 7.9 (a) provides a rough estimate of the radial extent of the toroidal vortex, which is $r \approx 0.6$ for both the Ra numbers. The radial extent of the vortex predicted in this study is consistent with the experimental results of Long et al. [47]. They predicted the radial extent of the toroidal vortex for $Ro = 3.57$ ($Re_\theta = 4.9 \times 10^5$, $Re_z = 9.2 \times 10^4$ and $\beta\Delta T_{shroud} = 0.23$) to be approximately $r = 0.6$ for the same cavity dimensions as used in our study. Albeit, the value of Ro number was 0.8 times smaller in the experiments; still, there is a good agreement on the extent of the toroidal vortex, which may depend mainly on the cavity dimensions.

Figure 7.9(b) shows the variation of radial velocity at the mid-axial location of the cavity. The radial velocity is negative, which indicates that the flow is directed inwards at this location. The variation of radial velocity in the axial direction in Region II is shown in figure 7.10 (b). Again, the profiles show a lack of symmetry in the axial direction of the toroidal vortex. The flow direction into the cavity is biased towards the downstream disk of the cavity. The flow enters the cavity by the downstream disk covering 20% of the axial length for flow ingestion. It leaves the cavity through the remaining 80% of the cavity opening via the upstream disk. The comparison of radial velocity profiles for the two Ra numbers shows a small increase in the magnitude of radial velocity for the higher Ra number case. Overall, a 7% enhancement in the mass flow rate from the bore flow into the cavity is observed for the higher Ra number case. An additional 7.2% pressure loss from the bore flow inlet to the outlet is observed in the case of higher Ra number. Primarily the higher magnitude of the swirl ratio in the lower radii of the cavity leads to the pressure loss in the bore flow passage, which is detrimental to the overall performance of the internal air system [84]. The radial variation

of swirl ratio, which directly affects the pressure loss, is shown in figure 7.9(c). The peak value of swirl ratio is close to 1.5 for $Ra = 1.54 \times 10^8$ and reduces to approximately 1.3 for the low Ra number case. The peak location of swirl ratio $r \approx 0.34$ is also found to be consistent with the experimental results of [47] for the same cavity dimensions. The axial variation of the swirl ratio in Region II is shown in figure 7.10 (c). The peak of the swirl ratio lies near the upstream disk, and a significant effect of Ra number on the swirl ratio is perceived at the cavity inlet. The rise in the swirl ratio signifies the high intensity or strength of the toroidal vortex at the higher Ra number. In addition, the rise in the mass flow rate at the higher Ra number may also be related to the strength of the toroidal vortex, as the centrifugal buoyancy has minimal effect in this region. Thus, it is important to study the effect of various parameters on the vortex strength, which could be the reason behind the increase in the swirl ratio and the mass flow rate into the cavity. However, this is beyond the scope of this study.

The Region III of the cavity, away from the toroidal vortex is less energetic and dominated by the centrifugal-buoyant force. It has radial flares emerging from the toroidal vortex moving radially outwards as well as inwards in the cavity. The magnitude of axial velocity in this region is very small, as shown in figure 7.10(d). Thus, the flow is mostly directed towards the radial and the azimuthal directions. The axial variation of the radial velocity in Region III shows that the flow moves radially outwards through the boundary layers formed on the side disk and inwards by the central core of the cavity, as shown in figure 7.10 (e). The boundary layers formed on the side disks are three-dimensional in nature and called Ekman layers. In the Ekman layer at a particular radial location on the side disk, the flow in the core of the cavity is aligned in the azimuthal direction and near to the wall turns in the radial direction forming an Ekman spiral, as shown in figure 7.11 (a). The velocity components plotted in figure 7.11 (a) are in the frame of reference of rotation. The Ekman spiral shows the direction of the resultant velocity vector, which gradually changes from the radial to azimuthal direction in the wall-normal direction. The distance in the wall-normal direction

at which the radial or transverse velocity component crosses the origin is defined as a boundary layer thickness[5].

The Ekman spiral formed on the side disk is compared with the solution of linear Ekman layer equations [5] by assuming the flow to be steady, laminar and that the Coriolis force term is much larger than the nonlinear inertial terms in the equations. The final solution to the equations is

$$\begin{aligned} v &= v_{core} [1 - e^{-z/\delta} \cos(z/\delta)] - w_{core} e^{-z/\delta} \sin(z/\delta), \\ w &= w_{core} [1 - e^{-z/\delta} \cos(z/\delta)] + v_{core} e^{-z/\delta} \sin(z/\delta), \end{aligned} \quad (7.5)$$

where v_{core} and w_{core} are the axially averaged core radial and the circumferential velocity components, respectively. The comparison shows a reasonable agreement at $r = 0.70$ and 0.75 . However, at $r = 0.80$ and 0.85 , the shape of the Ekman spiral is different than the prediction from the linear Ekman layer equations. The peak radial velocity has reduced at both the radial locations. However, there is good agreement in terms of radial location at which the radial velocity component crosses the origin, which provides the Ekman boundary layer thickness as

$$\delta_{Ek} = \pi \sqrt{\frac{\mu}{\rho \Omega}}. \quad (7.6)$$

Figure 7.11 (b) shows the comparison of the Ekman layer thickness for both side disks for the two Ra numbers. A quasi-constant boundary layer thickness is observed for both the Ra numbers, except at the downstream disk for the higher Ra number case for which the boundary layer thickness reduces in the radial direction. The relative comparison of boundary layer thickness between the two Ra number shows a rise with the Ra number. Thus, at higher Ra number, the radial flow rate inside the Ekman layers towards the shroud increases. The comparison with the analytical result in equation 7.6 shows a good agreement for the low Ra number case at both upstream and downstream disks. However, for the higher Ra number case, the Ekman boundary layer thickness predicted from the simulations is higher than that predicted by the analytical result. One possible reason for these differences could be the rise in the level of root-mean-square of radial velocity v_{rms}

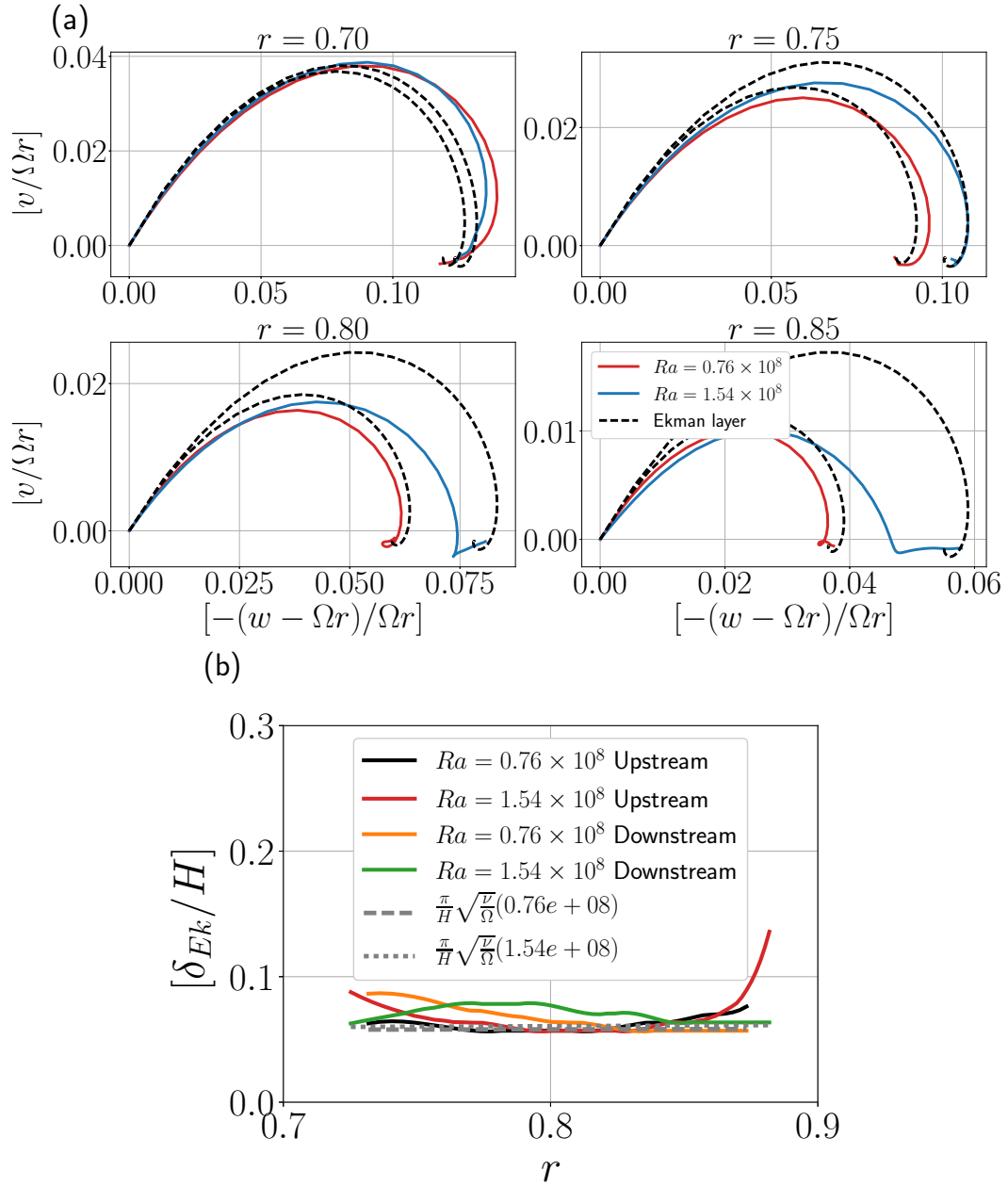


FIGURE 7.11: (a) Ekman spiral formed at different radial locations on upstream disk and comparison with the solution of linear Ekman layer equations, (b) Comparison of boundary layer thickness for different Ra numbers on both the disks scaled with disk gap H .

fluctuations, and it defies the laminar assumption used to derive equation 7.6. For instance, the comparison of v_{rms} at $r = 0.8$ for both Ra numbers integrated over wall-normal distance δ_{Ek} shows the two times stronger fluctuations at higher Ra number. Further, the comparison of the near-wall maximum value of v_{rms} with the radial velocity v shows order-of-magnitude smaller values of fluctuations. This result is in contrast with the closed cavity results as reported by Pitz et al. [65] for which v_{rms} values were order-of-magnitude larger than the radial velocity itself. Thus, they defined the boundary layer thickness based on the maximum value of v_{rms} . However, in this study, the relatively smaller magnitude of radial velocity fluctuations with respect to velocity leads to good agreement with the laminar Ekman layer thickness defined by equation 7.6, as shown in figure 7.11 (b). Thus, the solution to the laminar Ekman layer equations and the derived closed-form equation 7.6 for boundary layer thickness is a good approximation.

Further, the variation of swirl ratio in the axial direction in Region III is shown in figure 7.10 (f). It reduces to a value of less than one in the outer region. Thus, the flow is rotating slower than the local disk speed, and it further reduces with the increase in the Ra number and temperature of the side disks. This phenomenon was also observed in the experiments by Owen and Powell [45]. They observed a decrease in the value of local swirl ratio with a rise in the temperature of the side disk (only one of the disks was heated in the experiments). However, along the radial direction, the local azimuthal velocity approaches the local disk speed.

The Region IV is mostly dominated by small scale convection cells formed due to the centrifugal buoyancy. Figures 7.10 (g-i) show the averaged velocity profiles in this region. The radial velocity is negative in the disk boundary layers, which implies that the flow is directed towards the toroidal vortex. In the core region, the radial velocity is positive, which suggests the formation of local convection plumes, which rise towards the shroud due to the centrifugal buoyancy. The relative comparison between the two Ra numbers shows a significant increase in the magnitude of radial velocity due to the large buoyant force for the higher Ra number case.

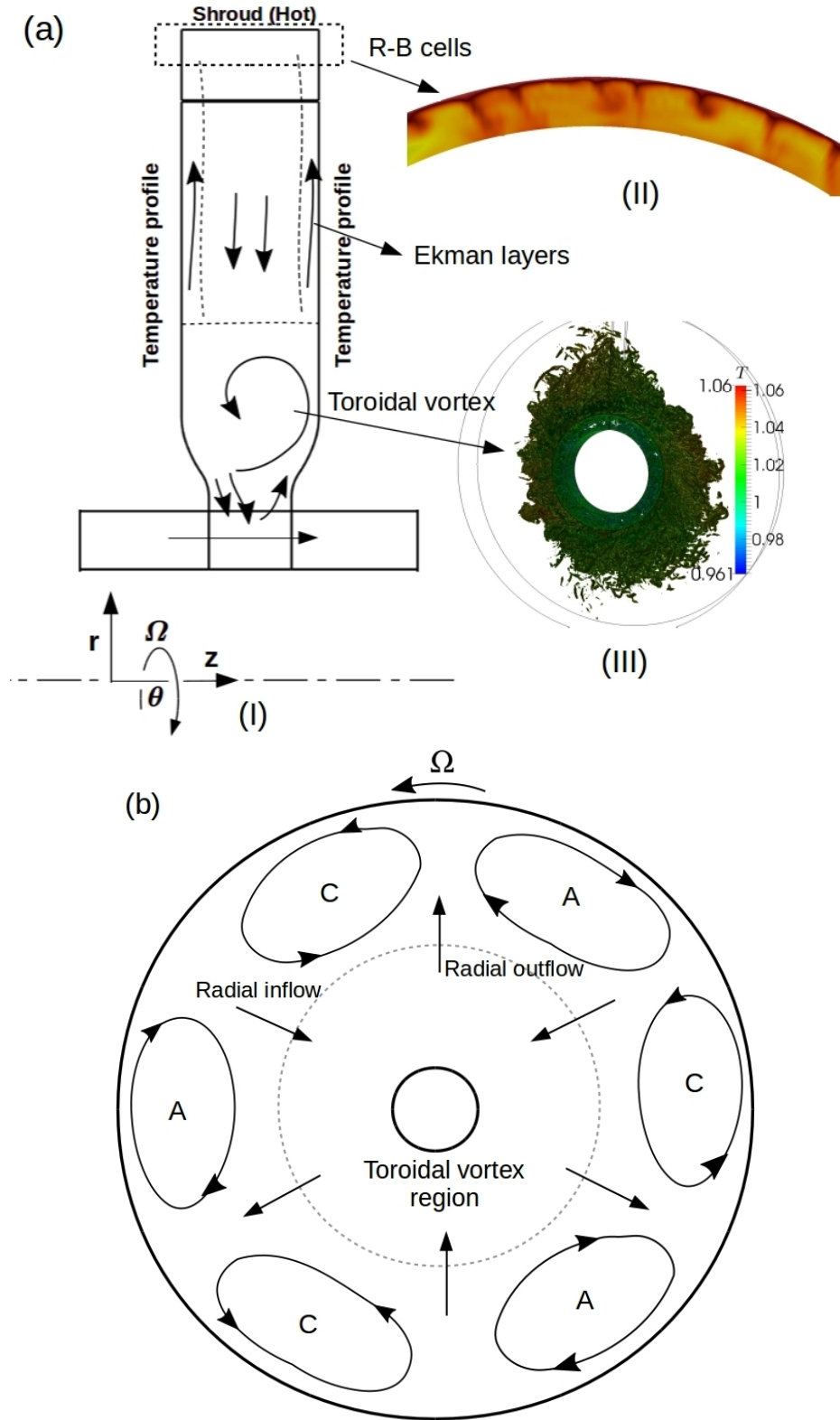


FIGURE 7.12: (a) Schematic of flow structure for the heated shroud and side disks with radially increasing temperature profile (I) interpretation from the time and circumferentially averaged statistics, (II) instantaneous temperature contours near the shroud shows the local convection cells (III) visualisation of toroidal vortex by iso-surfaces of λ -ci colored by local temperature, (b) Flow structure in the radial-azimuthal plane interpreted from the instantaneous flow field, where C refers to cyclonic and A to anti-cyclonic vortices.

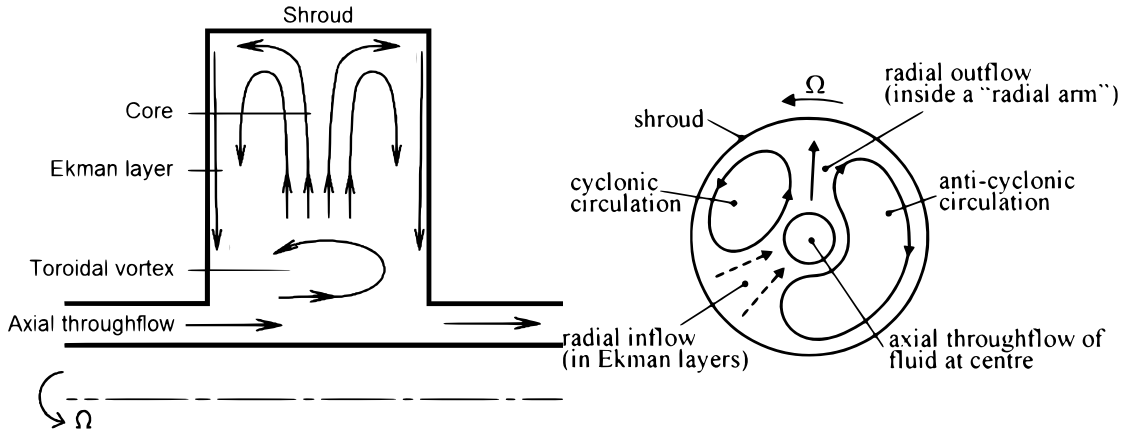


FIGURE 7.13: Schematic of flow structure proposed by Farthing et al. [42] from [4] (Left figure). Schematic of assumed flow structure by [43] (Right figure)

The salient features of the flow structure, as discussed above, have been summarised in figure 7.12. The axial bore flow expands into the cavity and forms a toroidal vortex, which evolves with time. A fluid particle in the toroidal vortex region follows a helical path. The curvature of the coils at the cavity inlet induce the vorticity in the $r - z$ plane, which extends in the azimuthal direction to form a closed loop and rotates with the cavity walls to form the toroidal vortex. The asymmetric toroidal vortex leads to a flow structure biased towards the downstream disk and forms a boundary layer through which the flow enters into the cavity. This physical phenomenon is identical to the one observed by Farthing et al. [42], shown in figure 7.13. From the toroidal vortex region, the flow slows down in the outer region and the radial flares emanating from the toroidal vortex move towards the shroud due to the centrifugal buoyancy. They form the Ekman layer on the side disks and flow moves radially outwards towards the shroud and moves radially inwards by the core region of the cavity as concluded from the time and circumferentially averaged statistics. These results are contrary to the flow structure observed by Farthing et al. [42] (refer to figure 7.13) for a radially decreasing temperature distribution on the side disks. The interpretation of the instantaneous flow field in the radial and circumferential plane is shown in figure 7.12 (b). The toroidal vortex bifurcates to form three dominant radial flares in the outer region of the cavity. The radial flares moving outwards create a pair of cyclonic and anti-cyclonic vortices. The flow between the consecutive anti-cyclonic

and cyclonic vortices created by two different radial outward moving flares is being pushed towards the toroidal vortex, as shown in figure 7.12 (b). In the near-shroud region, the vigour of the radial flares reduces or is limited to a few circumferential locations. Thus the outer region is mostly dominated by the local convection cells induced by the centrifugal buoyancy force, as shown by instantaneous temperature contours in figure 7.12 (a-II).

7.4 Heat Transfer

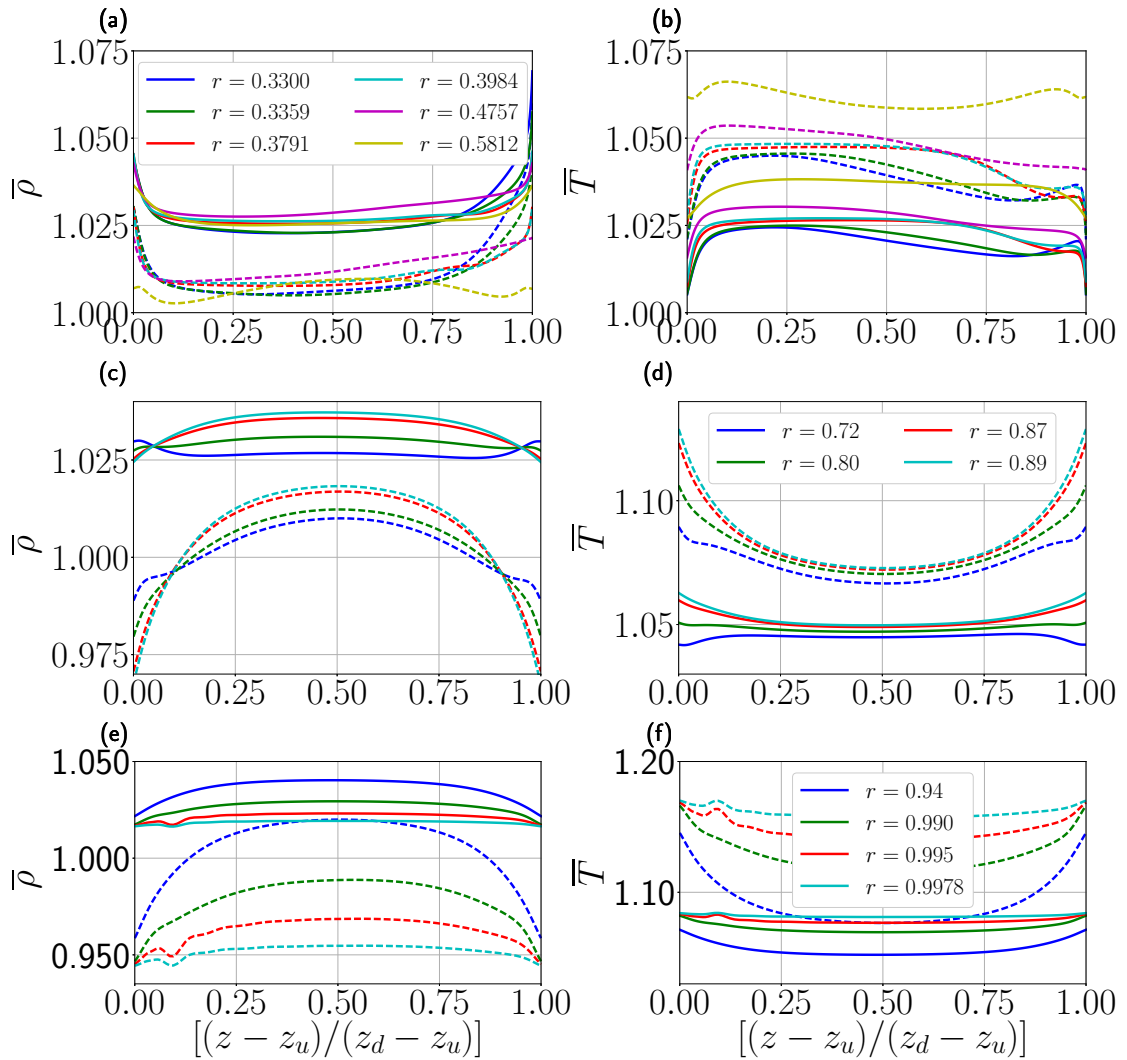


FIGURE 7.14: Comparison of time and circumferentially averaged density $\langle \bar{\rho} \rangle_\theta$ and temperature $\langle \bar{T} \rangle_\theta$ profiles at the different radial locations of the cavity in Region II (a,b), Region III (c,d) and Region IV (e,f) for different Rayleigh numbers (— $Ra = 0.76 \times 10^8$, --- $Ra = 1.54 \times 10^8$).

In this section, the results of wall heat transfer rate from the side disks and the shroud are presented. Figure 7.14 shows the mean density and temperature profile in each region of the cavity. The mean density and temperature profiles in the Region II (figures 7.14 (a) and (b)) of the cavity show the conditions which lead to negative heat transfer rate from the fluid to the side disk. The mixing of hot fluid from the outer region of the cavity with the bore flow leads to a rise in the average temperature of the fluid in this region compared to the temperature of the side disks. The density of the fluid also drops due to this thermal mixing of hot and cold fluid. In the radially outward direction into the cavity to the Region III, the density and temperature profiles change to the conditions conducive for buoyancy-induced convection and positive wall heat transfer rate. The density in the core of the cavity increases and temperature decreases. The radial location at which the profiles overturn depends upon the temperature distribution on the side disks, and it is different for the two Ra numbers. Further away, in the Region IV, the effect of radial flares is minimal and thus the region is wholly dominated by the centrifugal buoyancy-induced convection.

Figure 7.15 shows the variation of Nusselt number Nu_r^∞ on the side disks calculated based on the shaft temperature (T_s). In the lower region of the cavity near to the cobs, the heat transfer rate from both the side disks is negative. In the outer region, the heat transfer rate is positive. There is not much variation observed in the circumferential direction on the side disks. To understand the variation of Nusselt number in the radial direction, the circumferentially averaged values of Nu_r^∞ are plotted in figure 7.16. The heat transfer rate is negative in the toroidal vortex region for both cases. The radial location at which the heat transfer rate becomes positive depends primarily on the temperature distribution on the side disks. For instance, at higher Ra number, positive heat transfer rate is observed approximately at $r = 0.6$ as compared to $r = 0.7$ for the low Ra number case. Overall, a higher heat transfer rate is observed for $Ra = 1.54 \times 10^8$ at both the disks as compared to the low Ra number case. The distribution of Nu_r^∞ is quasi-linear without much variation between the two disks. However, the Nusselt number Nu_r^c calculated based on the local core temperature in the cavity provides

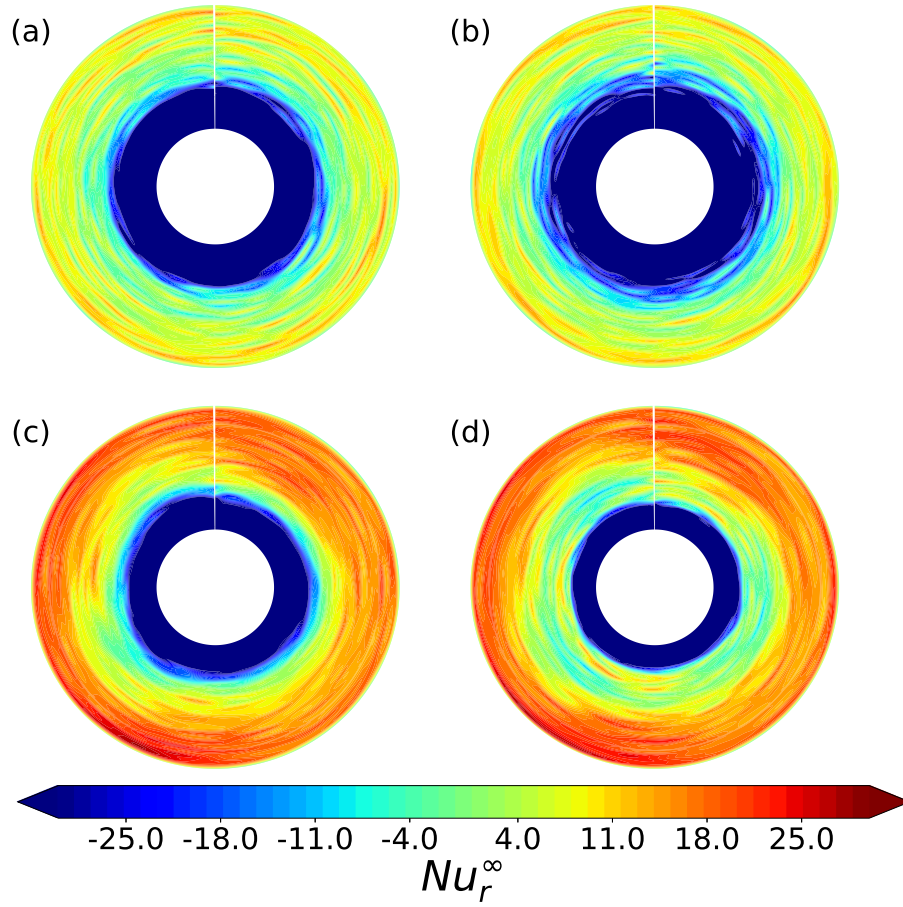


FIGURE 7.15: Comparison of time averaged Nusselt number ($Nu_r^\infty = [r/(T_w - T_\infty)][-d\bar{T}/dz]_w$) calculated based on the shaft temperature (T_∞) for $Ra = 0.76 \times 10^8$ at (a) upstream disk, (b) downstream disk and for $Ra = 1.54 \times 10^8$ at (c) upstream disk, (d) downstream disk.

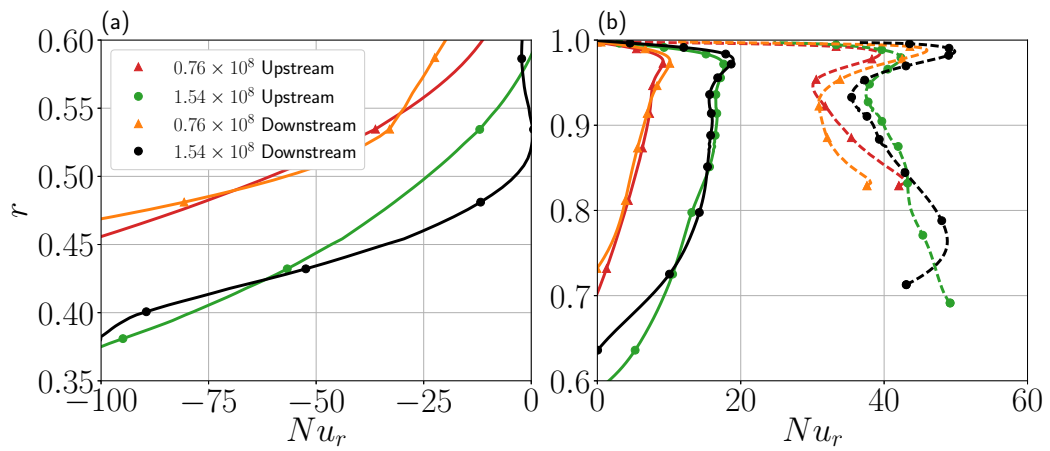


FIGURE 7.16: Comparison of the variation of local Nusselt number defined based on the shaft temperature (T_∞) ($Nu_r^\infty = [r/(T_w - T_\infty)][-d\bar{T}/dz]_w$) and based on the core cavity temperature ($Nu_r^c = [r/(T_w - T_c)][-d\bar{T}/dz]_w$) on the side disks for different Rayleigh numbers.

the more sensible quantification of local heat transfer effectiveness. It should be noted, though, that this definition breaks down at the radial location where the core temperature reaches values close to the local disk temperature and the denominator in equation 7.4 therefore approaches zero. The comparison of Nu_r^c on the side disks for both the Ra numbers shows higher values at the downstream disk as compared to the upstream one, and the heat transfer rate is further enhanced for the higher Ra number case. In the near-shroud region, the Nusselt number approaches zero as the temperature gradient in the axial direction tends to zero due to the same temperature level of the side disks and the shroud.

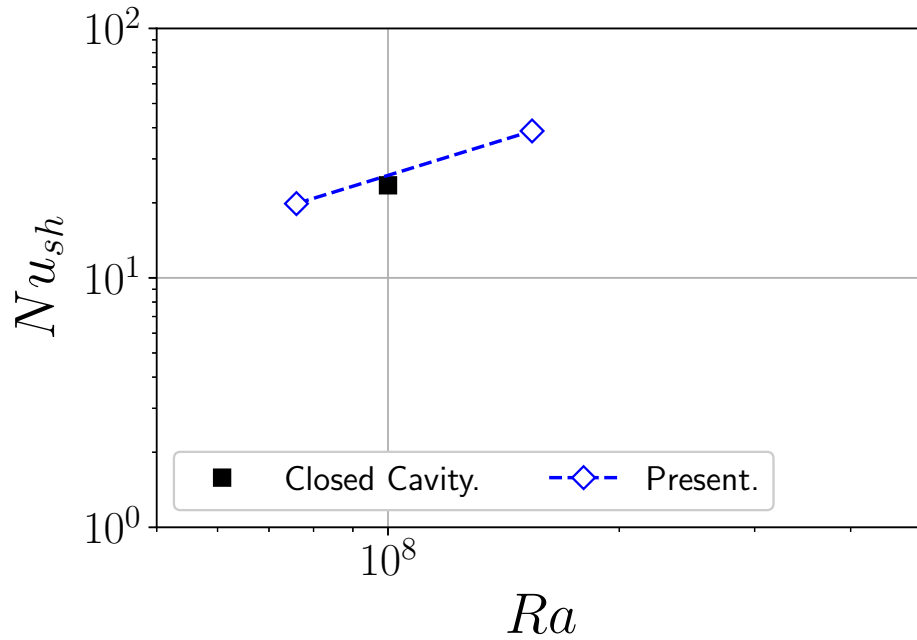


FIGURE 7.17: Comparison of shroud Nusselt number with the results of closed cavity without bore flow from [57]. It is important to highlight that the radius ratio (r_i/r_o) for the closed cavity is 0.52 as compared to the cavity with axial bore flow for which the radius ratio is 0.318. Thus in this comparison, the effect of this difference in the radius ratio is neglected.

Further, the shroud Nu number is calculated based on the definition used for a closed cavity [57]

$$Nu_{sh} = \frac{q_{convection}}{q_{conduction}} = \frac{\ln(r_o/r_i)}{\Delta T/T_\infty} \left[\frac{d\bar{T}}{dr} \right]_{r=1}, \quad (7.7)$$

where $\Delta T = T_{sh} - T_{\infty}$. The Nu numbers calculated for both Ra numbers are shown in figure 7.17. The comparison with the Nu number predicted for a closed cavity without the bore flow for $Ra = 10^8$ [57] is also shown in figure 7.17. All the values of Nu numbers lie on a straight line, which confirms that the centrifugal buoyancy-induced flow governs the heat transfer in the near shroud region and the axial bore flow does not affect it.

7.5 Conclusions

In this study, highly-resolved numerical simulations were conducted for flow in a high-pressure compressor inter-disk cavity for two Ra numbers of 0.76×10^8 and 1.54×10^8 at a fixed Ro number of 4.5, matching the experimental conditions of Atkins and Kanjirakkad [49]. The results of the present study show that the axial bore flow expands into the cavity to form a toroidal vortex. The toroidal vortex formed in the inner region has a radial extent up to $r \approx 0.6$, and the change in Ra number shows no effect on the radial extent. An increase in the radial mass flow rate and swirl ratio was observed for the higher Ra number case. For both the Ra numbers, three dominant structures in the outer region of the cavity, away from the toroidal vortex, were observed which correspond to three pairs of cyclonic and anti-cyclonic vortices. In this outer region, Ekman layers form on the side disk and were responsible for radial flow towards the shroud. The thickness of the Ekman layers increases with the increase in the Ra number. Further, the thickness of the Ekman layers was found to be in good agreement with the solution of linear Ekman layer equations for the low Ra number case and deviated for the high Ra number. The mixing of hot fluid from the outer region with the cold bore flow leads to a rise in the average temperature of the flow in the toroidal vortex region and a negative heat transfer rate to the side disks was observed for both the Ra numbers. The flow features near the shroud region were reminiscent of Rayleigh–Bénard streaks evolved in centrifugal-buoyant situations. The averaged wall Nu number at the shroud for both the Ra numbers and the Nu number for a closed

cavity without bore flow were shown to all lie on a straight line, which highlights the inefficacy of the bore flow to enhance the shroud heat transfer rate.

Chapter 8

Conclusions and Scope for future work

In the present study, the fluid flow and the induced heat transfer rate in the centrifugal buoyancy-induced convection were characterized by carrying out high-fidelity simulations for two different configurations of the computational domain. The first configuration was a canonical setup of a closed cavity, and in the second configuration, the cavity with an axial throughflow was considered. The research questions identified in chapter 2 were the focus of this thesis work and below is the chapter-wise summary with the major conclusions drawn against each of the research question

- **Chapter 1** introduced the scientific and the engineering significance of rotating flows. A brief introduction to the internal air cooling system and its importance to the performance of a gas turbine engine was presented, which also featured the motivation behind this thesis work.
- **Chapter 2** summarized the existing literature pertinent to a closed cavity configuration as well as to a cavity with an axial throughflow. It was observed that for the canonical closed cavity configuration there were only a few experimental studies which mainly paid attention to the wall heat transfer rate and reported a reduced scaling exponent of the $Nu-Ra$ scaling law. Thereafter, most of the

numerical studies were focussed on reproducing the experimental results and tried to explain the $Nu-Ra$ scaling law observed in the experiments. Thus, there existed a gap in the literature to methodically vary the governing parameters that could have provided more physical insights on the flow physics and their effects on the $Nu-Ra$ scaling law.

For a cavity with an axial throughflow, there has been a plethora of experimental studies focussed on understanding the flow dynamics and wall heat transfer rate. The literature lacked in providing a general interpretation of the flow structure that developed in a cavity for engine conditions due to experimental complexity and scarcity of highly resolved numerical studies. Thus, in this study, highly resolved large-eddy simulations were conducted to shed light into the flow structure that previously was poorly understood.

- **Chapter 3** presented the governing equations and numerical techniques used in this study to conduct the high-fidelity simulations.
- **Chapter 4** provided the results of direct numerical simulations for a closed cavity configuration replicating the experimental conditions of Bohn et al. [23]. The simulation results showed a well-defined flow structure in the cavity with the convection rollers in the domain analogous to Rayleigh-Bénard convection. The scaling of Nu number with Ra number agreed well with the corrected scaling exponent of the experiments. Further, the comparison was made with the results of an incompressible solver using the Boussinesq approximation. The investigation showed the deviation in Nu numbers as well as in the core temperature profiles. The reason behind this deviation was due to the use of the Boussinesq approximation, which considered the density as a linear function of temperature.
- **Chapter 5** discussed the importance of compressibility effects in the centrifugal buoyancy-induced flow in a closed cavity. Direct numerical simulations were conducted to isolate the effects of flow Mach number. The analysis showed the suppression of hydrodynamic instability in the cavity with an increase in the value of flow Mach number. As a consequence, the reduced wall heat transfer

rate was observed due to the density stratification in the domain. The compressibility effects were further elaborated by conducting linear-stability analyses. With an increase in the flow Mach number, suppression in the growth of hydrodynamic instability, which led to the formation of convection rollers, was observed. A stability condition based on static stability of the flow was proposed, which showed a strong agreement with the results of the linear-stability analysis. The stability criterion established that the flow Mach number imposed a constraint on the critical value of Rossby number for the flow to be linearly-stable.

- **Chapter 6** looked at the effects of Ro number by conducting the direct numerical simulations for Ra numbers in the range $10^5 \leq Ra \leq 10^9$. At a low Ro number, a strong influence of the Coriolis forces constrained the flow to a quasi-two-dimensional state in the axial direction. This stabilizing effect was also responsible for a reduced heat transfer rate at a low Ro number. However, no effect of Ro number on the Nu – Ra scaling exponent was observed as it appeared to be in agreement with the 1/3 scaling law of Rayleigh–Bénard convection. The comparison of Nu – Ra scaling with the experimental results of Bohn et al. [23] provided evidence that low Ro number values at high Ra numbers led to the reduced scaling exponent in the experiments. Further, the conditional plume-to-plume averaging of the flow field allowed to understand the role of plumes in the local heat transfer enhancement. The mean temperature profiles, as well as root-mean-square of its fluctuations, averaged over the plume-ejection regions, exhibited a logarithmic variation with the wall-normal distance in the near-wall bulk region. The Nu – Ra scaling exponent in those plume-ejection regions also increased to a value of 0.379 due to the increased level of turbulence.
- **Chapter 7** presented highly-resolved large-eddy simulations results for a cavity with an axial throughflow. A depiction of flow structure was provided, which had a toroidal vortex in the inner region of the cavity, which was found to be responsible for a radial inflow as well as thermal mixing. The laminar Ekman layers formed on the side disks in the region away from the toroidal vortex

were responsible for the radial flow towards the shroud region. The thickness of these Ekman layers was found to be in good agreement with the solution of the linear Ekman layer equations, which could be used to estimate the radial mass flow rate in the outer region. The wall heat transfer rate from the shroud region was found to be unaffected by the presence of an axial throughflow. It thus corroborated the dominance of centrifugal buoyancy-induced flow in the near-shroud region.

8.1 Scope for future work

The conclusions drawn from the current study leads to the following research directions, which can be a prospect for future work.

- The maximum value of Ra number in the current study was 10^9 with Ro numbers in the range $0.25 \leq Ro \leq 0.75$. An observation of logarithmic temperature profiles and consequently higher scaling exponent for $Nu-Ra$ power-law showed a transition phase to an ultimate state of convection. The ultimate state of convection is reached when the momentum boundary layers as well as thermal boundary layers become fully turbulent and follows the logarithmic profiles. The flow characteristics at ultimate convection can be extrapolated to high large Ra numbers relevant to various industrial application and natural phenomenon. Thus, an increase in the value of Ro number could provide a pathway to achieve the ultimate state at a relatively low Ra number. Thus, an extension of this study could be to increase the range of Ro number as well as Ra number to reach the ultimate state and analyze its characteristics.
- The linear-stability criteria proposed for centrifugal buoyancy-induced flows can further be extended to a wide range of parameters such as Ro number, Ma number and radius ratio r_i/r_o to find a critical value of Ra number at the onset of convection. This could provide a comprehensive level of information about the effect of different parameters on the flow stability and the heat transfer

rate. The linear-stability approach can also be applied to a cavity with an axial throughflow, which could offer more insights on the effect of different parameters.

- In the current study, only the high Ro number conditions were considered for a cavity with an axial throughflow. A natural extension is to conduct the high-fidelity simulations at low Ro number conditions as well, which could provide an understanding of the influence of axial throughflow on a shroud heat transfer rate. In those conditions, the higher magnitude of rotational Reynolds number will play an important role.
- The high-fidelity data-set from the current study could be used to assess the performance of existing turbulence models. Further, a data-driven approach such as machine learning could be used to propose new turbulence models with an improved performance to predict the physics of centrifugal buoyancy-induced flows.

Appendix A

Derivation for stability criteria

As discussed in chapter 5, for the convection instability to grow, the temperature gradient in the system has to be larger than the adiabatic temperature gradient. Thus, for a stable system the condition

$$\frac{dT}{dr} < \left(\frac{dT}{dr} \right)_{ad} \quad (\text{A.1})$$

has to be satisfied.

To further derive the stability criteria, we consider the first law of thermodynamics which states that

$$dQ = du + dW = c_v dT + p dv. \quad (\text{A.2})$$

We can write $p dv = p d(1/\rho) = (-p/\rho^2) d\rho$ and further by using the equation of state $p = \rho R T$ and then $dp = \rho R dT + R T d\rho$. The final expression for $p dv$ will be

$$p dv = -\frac{p}{\rho^2} d\rho = -\frac{dp}{\rho} + R dT. \quad (\text{A.3})$$

Now we can rewrite the equation A.2 as

$$dQ = (c_p - R) dT - \frac{dp}{\rho} + R dT. \quad (\text{A.4})$$

Since we are assuming the process to be adiabatic, thus $dQ = 0$ and equation A.4 becomes

$$\frac{dp}{\rho} = c_p dT. \quad (\text{A.5})$$

Further, the equation of hydrostatic balance in the rotating flows can be written as

$$\frac{dp}{dr} = \rho \Omega^2 r. \quad (\text{A.6})$$

By using the equations A.5 and A.6, the adiabatic temperature gradient is equal to

$$\left(\frac{dT}{dr} \right)_{adia} = \frac{\Omega^2 r}{c_p}. \quad (\text{A.7})$$

Thus, by using equation A.7, the expression for stability criteria becomes

$$\frac{dT}{dr} < \left(\frac{\Omega^2 r}{c_p} \right). \quad (\text{A.8})$$

Integrate equation A.7 from the inner radius to the outer radius (refer to figure 5.7 in chapter 5).

$$\begin{aligned} \int_{T_C}^{T_H} dT &< \int_{r_i}^{r_o} \left(\frac{\Omega^2 r}{c_p} \right) dr \\ \Delta T = T_H - T_C &< \frac{\Omega^2}{2c_p} (r_o^2 - r_i^2). \end{aligned} \quad (\text{A.9})$$

Further by using the definition of Mach number and Rossby number given in chapter 4, we can write the A.9 as

$$Ro < Ma_\infty^{1/2} \frac{(\gamma - 1)^{1/4}}{2^{3/2}} \left(\frac{1 + r_i/r_o}{1 - r_i/r_o} \right)^{1/2}. \quad (\text{A.10})$$

Appendix B

Cavity with axial-bore flow- Temperature distribution on side disks

TABLE B.1: Non-dimensional side disk temperature with respect to T_∞ for case I and II

r	Case I	Case II
0.3187	1.0049	1.0208
0.3772	1.0075	1.0251
0.4357	1.0120	1.0336
0.4943	1.0177	1.0443
0.5528	1.0239	1.0559
0.6113	1.0302	1.0678
0.6698	1.0365	1.0795
0.7284	1.0427	1.0911
0.7869	1.0492	1.1032
0.8454	1.0564	1.1168
0.9000	1.0643	1.1319
0.9202	1.0677	1.1384
0.9404	1.0714	1.1455
0.9606	1.0754	1.1534
0.9808	1.0799	1.1620
1.0000	1.0846	1.1712

Solution to linear Ekman layer equations

The final solution to the Ekman layer equations [5][65] is

$$\begin{aligned} v &= v_{core} [1 - e^{-z/\delta} \cos(z/\delta)] - w_{core} e^{-z/\delta} \sin(z/\delta), \\ w &= w_{core} [1 - e^{-z/\delta} \cos(z/\delta)] + v_{core} e^{-z/\delta} \sin(z/\delta), \end{aligned} \tag{B.1}$$

where, $\delta = \sqrt{\mu/(\rho\Omega)}$, defined as the Ekman depth. The v_{core} and w_{core} are the axially averaged core radial and the circumferential velocity components, respectively.

Appendix C

Grid generation - Cavity with axial-bore flow

A grid generation tool is developed for cavity with an axial-throughflow based on the method proposed by Eça [85]. Out of several grid quality parameters, the orthogonality is the most important one and difficult to achieve in a complex geometrical configuration. In this approach, a set of partial differential equations are numerically solved to generate the grid given as

$$\begin{aligned}\frac{\partial}{\partial \xi} \left(f \frac{\partial x}{\partial \xi} \right) + \frac{\partial}{\partial \eta} \left(f \frac{\partial x}{\partial \eta} \right) &= 0, \\ \frac{\partial}{\partial \xi} \left(f \frac{\partial y}{\partial \xi} \right) + \frac{\partial}{\partial \eta} \left(f \frac{\partial y}{\partial \eta} \right) &= 0,\end{aligned}\tag{C.1}$$

where f is the distortion function defined as

$$f = \frac{h_\eta}{h_\xi}\tag{C.2}$$

and

$$\begin{aligned}h_\eta &= \left[\left(\frac{\partial x}{\partial \eta} \right)^2 + \left(\frac{\partial y}{\partial \eta} \right)^2 \right]^{1/2}, \\ h_\xi &= \left[\left(\frac{\partial x}{\partial \xi} \right)^2 + \left(\frac{\partial y}{\partial \xi} \right)^2 \right]^{1/2}.\end{aligned}\tag{C.3}$$

The numerical solution of equations C.1 is initially approximated by linear transfinite interpolation to calculate the initial values of distortion function and then iterate until it satisfies the convergence criteria. The final grid generated for the cavity is shown in figure C.1.

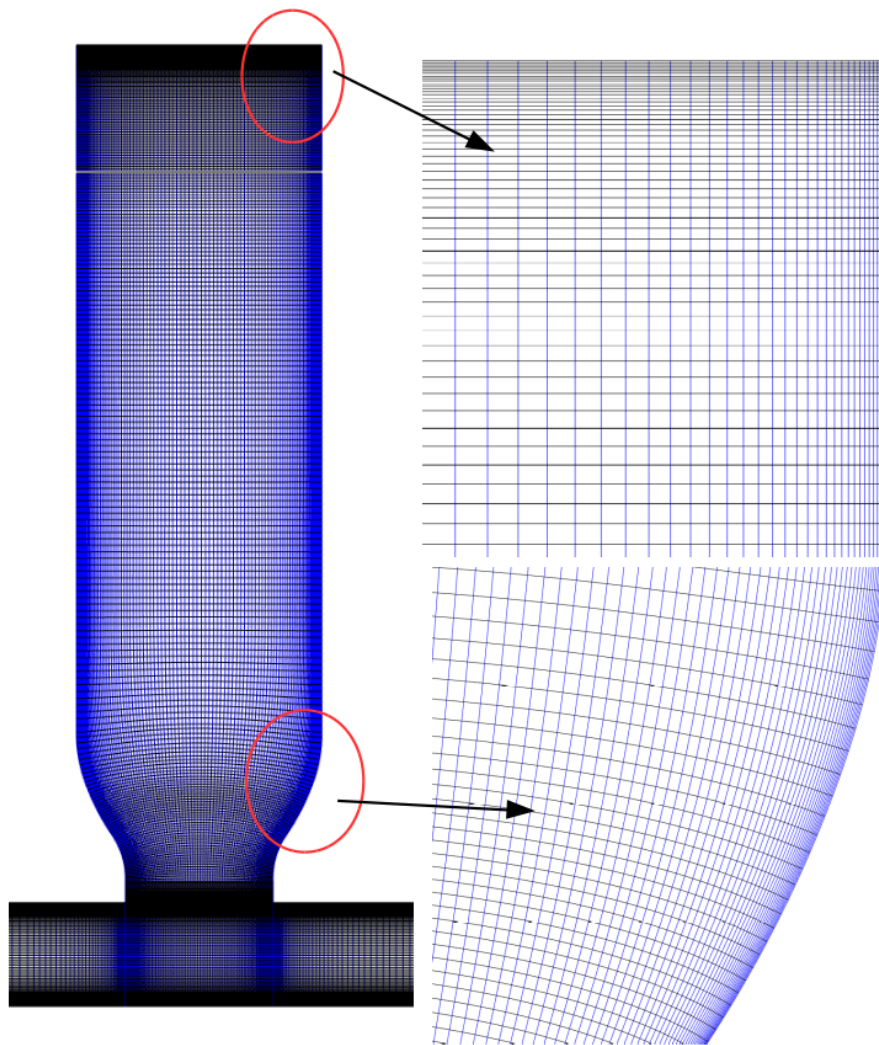


FIGURE C.1: Grid for inter-disk cavity.

Appendix D

List of all the DNS cases for a closed cavity

Here is the list of all the DNS cases for a closed cavity with the corresponding non-dimensional parameters and the Nuseelt number values calculated at the hot outer wall.

Case	Ra	Ro	Ma	Nu
1	10^5	0.25	0.06	3.65
2	10^6	0.30	0.059	6.15
3	10^7	0.26	0.038	11.40
4	10^8	0.23	0.043	23.50

TABLE D.1: DNS conditions replicating the experimental conditions of [23] from chapter 4.

Case	Ra	Ro	Ma	Nu
1	$10^5, 10^6, 10^7, 10^8, 10^9$	0.25	0.1	6.6, 10.2, 13.6, 27.2, 66.2
2	$10^5, 10^6, 10^7, 10^8, 10^9$	0.50	0.1	5.7, 10.7, 18.1, 39.4, 83.1
3	$10^5, 10^6, 10^7, 10^8, 10^9$	0.75	0.1	4.25, 6.7, 14.3, 35.7, 75.5

TABLE D.2: DNS cases from chapter 6 to study the effect of Rossby number.

Appendix E

Azimuthal Fourier Transforms

In the azimuthal direction, symmetric Fourier transforms were used. The symmetry with respect to the azimuthal direction for each of the flow variable is established to replace physical quantities with the sine and cosine series. To demonstrate this, the governing equations are shown below in the Fourier space

$$\begin{aligned}
 & \frac{\partial}{\partial t} \sum_{k=0}^{kh} \begin{pmatrix} \hat{Q}_1^k \cos \alpha_\theta^k \theta \\ \hat{Q}_2^k \cos \alpha_\theta^k \theta \\ \hat{Q}_3^k \cos \alpha_\theta^k \theta \\ \hat{Q}_4^k \cos \alpha_\theta^k \theta \\ \hat{Q}_5^k \cos \alpha_\theta^k \theta \end{pmatrix} + \frac{\partial}{\partial z} \sum_{k=0}^{kh} \begin{pmatrix} \hat{E}_1^k \cos \alpha_\theta^k \theta \\ \hat{E}_2^k \cos \alpha_\theta^k \theta \\ \hat{E}_3^k \cos \alpha_\theta^k \theta \\ \hat{E}_4^k \cos \alpha_\theta^k \theta \\ \hat{E}_5^k \cos \alpha_\theta^k \theta \end{pmatrix} + \frac{\partial}{\partial r} \sum_{k=0}^{kh} \begin{pmatrix} \hat{F}_1^k \cos \alpha_\theta^k \theta \\ \hat{F}_2^k \cos \alpha_\theta^k \theta \\ \hat{F}_3^k \cos \alpha_\theta^k \theta \\ \hat{F}_4^k \cos \alpha_\theta^k \theta \\ \hat{F}_5^k \cos \alpha_\theta^k \theta \end{pmatrix} \\
 & + \frac{1}{r} \frac{\partial}{\partial \theta} \sum_{k=0}^{kh} \begin{pmatrix} \hat{G}_1^k \sin \alpha_\theta^k \theta \\ \hat{G}_2^k \sin \alpha_\theta^k \theta \\ \hat{G}_3^k \sin \alpha_\theta^k \theta \\ \hat{G}_4^k \sin \alpha_\theta^k \theta \\ \hat{G}_5^k \sin \alpha_\theta^k \theta \end{pmatrix} - \sum_{k=0}^{kh} \begin{pmatrix} \hat{S}_{v1}^k \cos \alpha_\theta^k \theta \\ \hat{S}_{v2}^k \cos \alpha_\theta^k \theta \\ \hat{S}_{v3}^k \cos \alpha_\theta^k \theta \\ \hat{S}_{v4}^k \cos \alpha_\theta^k \theta \\ \hat{S}_{v5}^k \cos \alpha_\theta^k \theta \end{pmatrix} = 0.
 \end{aligned} \tag{E.1}$$

In equation E.1, k is the azimuthal Fourier mode-number and α_θ^k is the azimuthal wavenumber. All the Fourier coefficients, $\hat{\psi} = \hat{\psi}(r, z)$, in equation E.1 are the function of radial and axial directions. The derivatives in the azimuthal direction

in Fourier space for each of the Fourier mode are calculated as

$$\left(\frac{\partial \hat{\psi}}{\partial \theta}\right)_k = \alpha_\theta^k \hat{\psi}_k. \quad (\text{E.2})$$

The Fourier transform is conducted using the FFTW libraries [86], which compute the unnormalized Fourier transform by multiplying with DFT transformation matrix W given as

$$W = \begin{bmatrix} 1 & 1 & 1 & 1 & \dots & 1 \\ 1 & \omega & \omega^2 & \omega^3 & \dots & \omega^{N-1} \\ 1 & \omega^2 & \omega^4 & \omega^6 & \dots & \omega^{2(N-1)} \\ 1 & \omega^3 & \omega^6 & \omega^9 & \dots & \omega^{3(N-1)} \\ \vdots & \vdots & \vdots & \vdots & \ddots & \vdots \\ 1 & \omega^{N-1} & \omega^{2(N-1)} & \omega^{3(N-1)} & \dots & \omega^{(N-1)(N-1)} \end{bmatrix}, \quad (\text{E.3})$$

where $\omega = e^{-2\pi i/N}$.

Appendix F

Variation of Nusselt number with time for a closed cavity

In reference to figure 4.9, the time variation of Nusselt number for $Ra = 10^5$ and 10^7 is given in figure F.1. The comparison of variation of Nu number in the transient phase of the flow shows a large variation of for Ra number of 10^7 , and it takes a longer time interval for Nu number to approach a statistically stable value

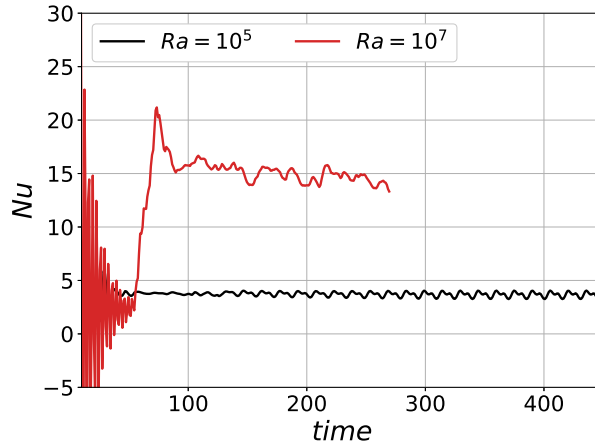


FIGURE F.1: Time variation of Nusselt number at the outer wall $r = r_o/L_\infty$ of the cavity for $Ra = 10^5$ and 10^7 . The initial time interval for $Ra = 10^7$ is shown here as the full dataset was not available. However, it is important to highlight that the flow statistics for $Ra = 10^7$ were calculated over the time interval from 383 to 540 (which is not shown here).

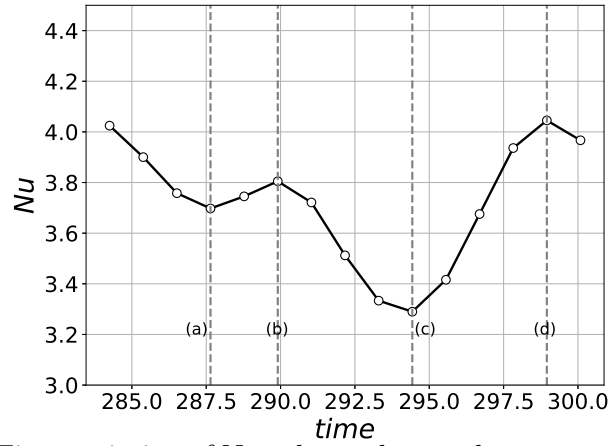


FIGURE F.2: Time variation of Nusselt number at the outer wall $r = r_o/L_\infty$ of the cavity for $Ra = 10^5$ and $Ro = 0.25$

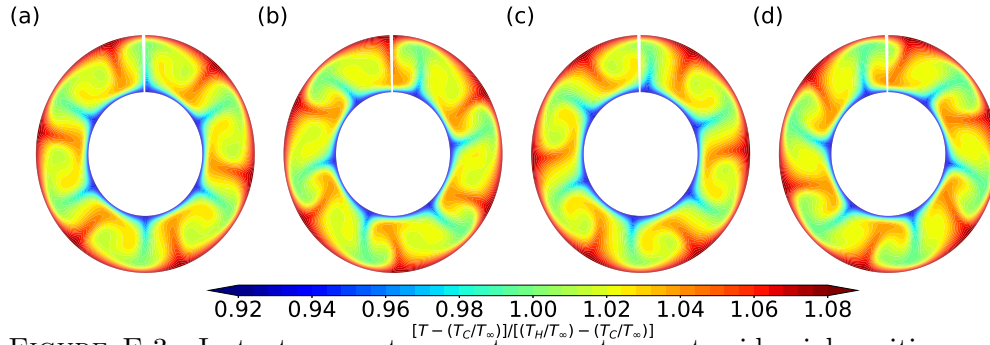


FIGURE F.3: Instantaneous temperature contours at mid-axial position $z = (1/2)(H/L_\infty)$ of the cavity at the corresponding minimum and maximum values of Nu number observed at the outer wall.

Further, after the flow transient, a 'kink' (a second peak) in the value of Nu is observed for $Ra = 10^5$ as shown in figure 4.9. The second peak or 'kink' can be observed more clearly in figure F.2 in the Nu number as shown by the local minima and the maxima. The reason for this variation in the Nu number is due to the oscillations or the movement of the convection plumes back and forth in the azimuthal direction. To explain this, the instantaneous contours of temperature are shown in figure F.3 at the respective minima and maxima observed in the Nu number values. At both the minimum values of Nu number, the plumes (both hot and cold) are almost aligned in the radial direction, as shown in figures F.3(a) and F.3(c). However, the plumes drift or oscillate in the azimuthal direction in figures F.3(b) and F.3(d), where the peaks in the Nu number values are observed and that explains the physical reason behind the 'kink' observed in figure 4.9.

Bibliography

- [1] FH Busse. Convective flows in rapidly rotating spheres and their dynamo action. *Physics of fluids*, 14(4):1301–1314, 2002.
- [2] Aurnou Jonathan, Moritz Heimpel, Lorraine Allen, Eric King, and Johannes Wicht. Convective heat transfer and the pattern of thermal emission on the gas giants. *Geophysical Journal International*, 173(3):793–801, 2008.
- [3] Peter L Read, Edgar P Pérez, Irene M Moroz, Roland MB Young, Th von Larcher, and PD Williams. General circulation of planetary atmospheres: insights from rotating annulus and related experiments. *Modeling Atmospheric and Oceanic Flows*, pages 9–44, 2015.
- [4] J Michael Owen and Christopher A Long. Review of buoyancy-induced flow in rotating cavities. *ASME J. Turbomach.*, 137(11), 2015.
- [5] Peter RN Childs. *Rotating flow*. Elsevier, 2010.
- [6] Jeffrey P Koplow. A fundamentally new approach to air-cooled heat exchangers. *Sandia National Laboratories, Albuquerque, New Mexico*, 87185: 3–48, 2010.
- [7] Global atmospheric circulation and biomes. <https://montessorimiddle.org/2011/04/21/global-atmospheric-circulation-and-biomes/>.
- [8] <https://www.iea.org/news/deep-energy-transformation-needed-by-2050-to-limit-rise-in-global-temperature>.
- [9] PK Nag. *Engineering thermodynamics*. Tata McGraw-Hill Education, 2013.

- [10] Alan H Epstein. Aeropropulsion for commercial aviation in the twenty-first century and research directions needed. *AIAA journal*, 52(5):901–911, 2014.
- [11] Bijay Sultanian. *Gas turbines: internal flow systems modeling*, volume 44. Cambridge University Press, 2018.
- [12] C Freeman. Effect of tip clearance flow on compressor stability and engine performance. *Von Karman Institute for Fluid Dynamics Lecture Series*, 5, 1985.
- [13] J Michael Owen, Hui Tang, and Gary D Lock. Buoyancy-induced heat transfer inside compressor rotors: Overview of theoretical models. *Aerospace*, 5(1):32, 2018.
- [14] JM Owen and ED Bilimoria. Heat transfer in rotating cylindrical cavities. *Journal of Mechanical Engineering Science*, 19(4):175–187, 1977.
- [15] PR Farthing, CA Long, JM Owen, and JR Pincombe. Rotating cavity with axial throughflow of cooling air: flow structure. In *ASME 1990 International Gas Turbine and Aeroengine Congress and Exposition*, pages V001T01A007–V001T01A007. American Society of Mechanical Engineers, 1990.
- [16] Henri Bénard. Les tourbillons cellulaires dans une nappe liquide.-méthodes optiques d’observation et d’enregistrement. *Journal de Physique Théorique et Appliquée*, 10(1):254–266, 1901.
- [17] Milton Van Dyke and Milton Van Dyke. An album of fluid motion. 1982.
- [18] Lord Rayleigh. Lix. on convection currents in a horizontal layer of fluid, when the higher temperature is on the under side. *The London, Edinburgh, and Dublin Philosophical Magazine and Journal of Science*, 32(192):529–546, 1916.
- [19] Subrahmanyan Chandrasekhar. *Hydrodynamic and hydromagnetic stability*. Courier Corporation, 2013.

- [20] Eberhard Bodenschatz, Werner Pesch, and Guenter Ahlers. Recent developments in Rayleigh-Bénard convection. *Annual review of fluid mechanics*, 32(1):709–778, 2000.
- [21] Guenter Ahlers, Siegfried Grossmann, and Detlef Lohse. Heat transfer and large scale dynamics in turbulent Rayleigh-Bénard convection. *Reviews of modern physics*, 81(2):503, 2009.
- [22] X Chavanne, F Chilla, B Castaing, B Hebral, B Chabaud, and J Chaussy. Observation of the ultimate regime in Rayleigh-Bénard convection. *Physical Review Letters*, 79(19):3648, 1997.
- [23] D Bohn, E Deuker, R Emunds, and V Gorzelitz. Experimental and theoretical investigations of heat transfer in closed gas-filled rotating annuli. *ASME J. Turbomach.*, 117(1):175–183, 1995.
- [24] Dieter Bohn and Jochen Gier. The effect of turbulence on the heat transfer in closed gas-filled rotating annuli for different rayleigh numbers. *ASME-paper 98-GT-542*, 542, 1998.
- [25] Zixiang Sun, A Kilfoil, John W Chew, and Nicholas J Hills. Numerical simulation of natural convection in stationary and rotating cavities. *ASME Paper No. GT2004-53528*, 2004.
- [26] Martin P King, Michael Wilson, and J Michael Owen. Rayleigh-Bénard convection in open and closed rotating cavities. *Journal of Engineering for Gas Turbines and Power*, 129(2):305–311, 2007.
- [27] Diogo B Pitz, John W Chew, Olaf Marxen, and Nicholas J Hills. Direct numerical simulation of rotating cavity flows using a spectral element-Fourier method. *Journal of Engineering for Gas Turbines and Power*, 139(7):072602, 2017.
- [28] Jose M Lopez, Francisco Marques, and Marc Avila. The Boussinesq approximation in rapidly rotating flows. *Journal of Fluid Mechanics*, 737:56–77, 2013.

- [29] Feng Gao, Diogo B Pitz, and John W Chew. Numerical investigation of buoyancy-induced flow in a sealed rapidly rotating disc cavity. *International Journal of Heat and Mass Transfer*, 147:118860, 2020.
- [30] Knut S Eckhoff and Leiv Storesletten. On the stability of rotating compressible and inviscid fluids. *Journal of Fluid Mechanics*, 99(2):433–448, 1980.
- [31] Alistair SR Kilfoil and John W Chew. Modelling of buoyancy-affected flow in co-rotating disc cavities. In *Turbo Expo: Power for Land, Sea, and Air*, volume 48845, pages 1113–1122, 2009.
- [32] PR Farthing, CA Long, JM Owen, and JR Pincombe. Rotating cavity with axial throughflow of cooling air: heat transfer. *ASME J. Turbomach.*, 114(1): 229–236, 1992.
- [33] Richard Jackson, Dario Luberti, Hui Tang, Oliver J Pountney, James Scobie, Carl Sangan, J Michael Owen, and Gary Lock. Measurement and analysis of buoyancy-induced heat transfer in aero-engine compressor rotors. *Journal of Engineering for Gas Turbines and Power*, 2020.
- [34] Hui Tang and J Michael Owen. Theoretical model of buoyancy-induced heat transfer in closed compressor rotors. *Journal of Engineering for Gas Turbines and Power*, 140(3), 2018.
- [35] Feng Gao and John W Chew. Ekman layer scrubbing and shroud heat transfer in centrifugal buoyancy-driven convection. *Journal of Engineering for Gas Turbines and Power*, 143(7):071010, 2021.
- [36] RPJ Kunnen, HJH Clercx, and Bernardus J Geurts. Breakdown of large-scale circulation in turbulent rotating convection. *EPL (Europhysics Letters)*, 84(2):24001, 2008.
- [37] Richard JAM Stevens, Jin-Qiang Zhong, Herman JH Clercx, Guenter Ahlers, and Detlef Lohse. Transitions between turbulent states in rotating Rayleigh-Bénard convection. *Physical review letters*, 103(2):024503, 2009.

- [38] Kai Leong Chong, Yantao Yang, Shi-Di Huang, Jin-Qiang Zhong, Richard JAM Stevens, Roberto Verzicco, Detlef Lohse, and Ke-Qing Xia. Confined Rayleigh-Bénard, rotating Rayleigh-Bénard, and double diffusive convection: A unifying view on turbulent transport enhancement through coherent structure manipulation. *Physical review letters*, 119(6):064501, 2017.
- [39] Susanne Horn and Olga Shishkina. Rotating non-Oberbeck-Boussinesq Rayleigh-Bénard convection in water. *Physics of Fluids*, 26(5):055111, 2014.
- [40] Amirreza Rouhi, Daniel Chung, Ivan Marusic, Detlef Lohse, and Chao Sun. Centrifugal buoyancy driven turbulent convection in a thin cylindrical shell. In *Proceedings of the 21st Australasian Fluid Mechanics Conference, AFMC 2018*, page 154054. Australasian Fluid Mechanics Society, 2018.
- [41] Hechuan Jiang, Xiaojue Zhu, Dongpu Wang, Sander G Huisman, and Chao Sun. Supergravitational turbulent thermal convection. *Science Advances*, 6(40):eabb8676, 2020.
- [42] PR Farthing, CA Long, JM Owen, and JR Pincombe. Rotating cavity with axial throughflow of cooling air: flow structure. *ASME J. Turbomach.*, 114(1):237–246, 1992.
- [43] J Michael Owen and Hui Tang. Theoretical model of buoyancy-induced flow in rotating cavities. *ASME J. Turbomach.*, 137(11):111005, 2015.
- [44] Dieter E Bohn, Gregor N Deutsch, Burkhard Simon, and Claus Burkhardt. Flow visualisation in a rotating cavity with axial throughflow. *ASME Paper No. 2000-GT-0280*, 2000.
- [45] J Michael Owen and Jonathan Powell. Buoyancy-induced flow in a heated rotating cavity. *Journal of Engineering for Gas Turbines and Power*, 128(1): 128–134, 2006.
- [46] J Michael Owen, Hans Abrahamsson, and Klas Lindblad. Buoyancy-induced flow in open rotating cavities. *Journal of Engineering for Gas Turbines and Power*, 129(4):893–900, 2007.

- [47] CA Long, NDD Miché, and PRN Childs. Flow measurements inside a heated multiple rotating cavity with axial throughflow. *International Journal of Heat and Fluid Flow*, 28(6):1391–1404, 2007.
- [48] Zixiang Sun, Klas Lindblad, John W Chew, and Colin Young. LES and RANS investigations into buoyancy-affected convection in a rotating cavity with a central axial throughflow. *Journal of Engineering for Gas Turbines and Power*, 129(2):318–325, 2007.
- [49] Nicholas R Atkins and Vasudevan Kanjirakkad. Flow in a rotating cavity with axial throughflow at engine representative conditions. *ASME Paper No. GT2014-27174*, 2014.
- [50] MR Puttock-Brown, MG Rose, and CA Long. Experimental and computational investigation of Rayleigh-Bénard flow in the rotating cavities of a core compressor. 2017.
- [51] Diogo B Pitz, John W Chew, and Olaf Marxen. Effect of an axial throughflow on buoyancy-induced flow in a rotating cavity. *International Journal of Heat and Fluid Flow*, 80:108468, 2019.
- [52] Frank M White. *Fluid mechanics*. Tata McGraw-Hill Education, 1979.
- [53] RD Sandberg. Compressible-flow DNS with application to airfoil noise. *Flow, Turbulence and Combustion*, 95(2-3):211–229, 2015.
- [54] Richard D Sandberg, Vittorio Michelassi, Richard Pichler, Liwei Chen, and Roderick Johnstone. Compressible direct numerical simulation of low-pressure turbines Part I: Methodology. *ASME J. Turbomach.*, 137(5):051011, 2015.
- [55] Richard Pichler, Richard D Sandberg, Gregory Laskowski, and Vittorio Michelassi. High-fidelity simulations of a linear HPT vane cascade subject to varying inlet turbulence. *ASME Paper No. GT2017-63079*, 2017.
- [56] John Leggett, Stephan Priebe, Aamir Shabbir, Vittorio Michelassi, Richard Sandberg, and Edward Richardson. Loss prediction in an axial compressor

- cascade at off-design incidences with free stream disturbances using large eddy simulation. *ASME J. Turbomach.*, 140(7):071005, 2018.
- [57] Deepak Saini, Daniel Chung, and Richard D Sandberg. Direct numerical simulations of centrifugal buoyancy induced flow in a closed rotating cavity. *In Proceedings of 21st Australasian Fluid Mechanics Conference, Adelaide, Australia, December 2018*, 2018.
- [58] Deepak Saini and Richard D Sandberg. Simulations of compressibility effects in centrifugal buoyancy-induced flow in a closed rotating cavity. *International Journal of Heat and Fluid Flow*, 85:108656, 2020.
- [59] Franck Nicoud and Frédéric Ducros. Subgrid-scale stress modelling based on the square of the velocity gradient tensor. *Flow, Turbulence and Combustion*, 62(3):183–200, 1999.
- [60] Gary N Coleman and Richard D Sandberg. A primer on direct numerical simulation of turbulence-methods, procedures and guidelines. 2010.
- [61] Olga Shishkina, Richard JAM Stevens, Siegfried Grossmann, and Detlef Lohse. Boundary layer structure in turbulent thermal convection and its consequences for the required numerical resolution. *New Journal of Physics*, 12(7):075022, 2010.
- [62] Diogo B Pitz, Olaf Marxen, and John W Chew. Onset of convection induced by centrifugal buoyancy in a rotating cavity. *Journal of Fluid Mechanics*, 826:484–502, 2017.
- [63] Susanne Horn, Olga Shishkina, and Claus Wagner. On non-Oberbeck–Boussinesq effects in three-dimensional Rayleigh–Bénard convection in glycerol. *Journal of Fluid Mechanics*, 724:175–202, 2013.
- [64] Xiao Zhong Wu and Albert Libchaber. Non-Boussinesq effects in free thermal convection. *Physical Review A*, 43(6):2833, 1991.

- [65] Diogo B Pitz, John W Chew, and Olaf Marxen. Large-eddy simulation of buoyancy-induced flow in a sealed rotating cavity. *Journal of Engineering for Gas Turbines and Power*, 141(2):021020, 2019.
- [66] David J Tritton. *Physical fluid dynamics*. Springer Science & Business Media, 2012.
- [67] Deepak Saini and Richard D Sandberg. The role of compressibility in the linear-stability of centrifugal buoyancy-induced flow. In *Proceedings of 22nd Australasian Fluid Mechanics Conference, Brisbane, Australia (2020)*, 2020.
- [68] FH Busse and CR Carrigan. Convection induced by centrifugal buoyancy. *Journal of Fluid Mechanics*, 62(3):579–592, 1974.
- [69] Changwoo Kang, Antoine Meyer, Harunori N Yoshikawa, and Innocent Muta-bazi. Numerical study of thermal convection induced by centrifugal buoyancy in a rotating cylindrical annulus. *Physical Review Fluids*, 4(4):043501, 2019.
- [70] LE Jones, RD Sandberg, and ND Sandham. Stability and receptivity characteristics of a laminar separation bubble on an aerofoil. *Journal of Fluid Mechanics*, 648:257–296, 2010.
- [71] Richard D Sandberg. Numerical investigation of turbulent supersonic axisymmetric wakes. *Journal of Fluid Mechanics*, 702:488–520, 2012.
- [72] Benoit Cushman-Roisin and Jean-Marie Beckers. *Introduction to geophysical fluid dynamics: physical and numerical aspects*. Academic press, 2011.
- [73] Guenter Ahlers, Eric Brown, Francisco Fontenele Araujo, Denis Funfschilling, Siegfried Grossmann, and Detlef Lohse. Non-Oberbeck-Boussinesq effects in strongly turbulent Rayleigh-Bénard convection. *Journal of fluid mechanics*, 569:409, 2006.
- [74] Eric M King, Stephan Stellmach, Jerome Noir, Ulrich Hansen, and Jonathan M Aurnou. Boundary layer control of rotating convection systems. *Nature*, 457(7227):301–304, 2009.

- [75] Kartik P Iyer, Janet D Scheel, Jörg Schumacher, and Katepalli R Sreenivasan. Classical $1/3$ scaling of convection holds up to $Ra = 1015$. *Proceedings of the National Academy of Sciences*, 117(14):7594–7598, 2020.
- [76] Siegfried Grossmann and Detlef Lohse. Fluctuations in turbulent Rayleigh–Bénard convection: the role of plumes. *Physics of fluids*, 16(12):4462–4472, 2004.
- [77] Guenter Ahlers, Eberhard Bodenschatz, Denis Funfschilling, Siegfried Grossmann, Xiaozhou He, Detlef Lohse, Richard JAM Stevens, and Roberto Verzicco. Logarithmic temperature profiles in turbulent Rayleigh–Bénard convection. *Physical review letters*, 109(11):114501, 2012.
- [78] Erwin P Van Der Poel, Rodolfo Ostilla-Mónico, Roberto Verzicco, Siegfried Grossmann, and Detlef Lohse. Logarithmic mean temperature profiles and their connection to plume emissions in turbulent Rayleigh–Bénard convection. *Physical review letters*, 115(15):154501, 2015.
- [79] Shi-Di Huang, Matthias Kaczorowski, Rui Ni, Ke-Qing Xia, et al. Confinement-induced heat-transport enhancement in turbulent thermal convection. *Physical review letters*, 111(10):104501, 2013.
- [80] Guenter Ahlers, Eberhard Bodenschatz, and Xiaozhou He. Logarithmic temperature profiles of turbulent rayleigh-bénard convection in the classical and ultimate state for a prandtl number of 0.8. *arXiv preprint arXiv:1404.3459*, 2014.
- [81] Deepak Saini and R Sandberg. Large eddy simulations of high rossby number flow in the high pressure compressor inter-disk cavity. 2020.
- [82] Emile Touber and Neil D Sandham. Large-eddy simulation of low-frequency unsteadiness in a turbulent shock-induced separation bubble. *Theoretical and Computational Fluid Dynamics*, 23(2):79–107, 2009.

-
- [83] Richard D Sandberg and Neil D Sandham. Nonreflecting zonal characteristic boundary condition for direct numerical simulation of aerodynamic sound. *AIAA journal*, 44(2):402–405, 2006.
- [84] M Pfitzner and W Waschka. Development of an aeroengine secondary air system employing vortex reducers. *22nd ICAS Congress, Harrogate, UK, August 2000*, 2000.
- [85] Luis Eça. 2D orthogonal grid generation with boundary point distribution control. *Journal of Computational Physics*, 125(2):440–453, 1996.
- [86] Matteo Frigo and Steven G Johnson. Fftw users manual. *Massachusetts Institute of Technology*, 1999.