Direct and large-eddy simulation of buoyancy-induced flows in rotating cavities

by

Diogo Berta Pitz

A thesis submitted for the degree of Doctor of Philosophy

UNIVERSITY OF SURREY

Department of Mechanical Engineering Sciences
University of Surrey
United Kingdom
February 2018
Abstract

In this research a spectral element method is used to perform direct numerical simulation (DNS) and implicit large-eddy simulation (LES) of flows induced by centrifugal buoyancy in rotating cavities. These flows occur, for instance, in the compressor cavities of gas turbines internal air systems, in which cooling air is used to extract heat from compressor disks. Buoyancy-induced flows are inherently challenging to study using computational fluid dynamics (CFD), since turbulence models based on the Reynolds-averaged Navier-Stokes (RANS) equations are not able to provide an accurate description of the phenomena induced by the interplay between buoyancy and rotation. For this reason, model-free approaches are desirable, since they can provide an accurate description of the flow physics. First, the method is applied to a rotor/stator configuration, in which regions of laminar, transitional and fully turbulent flow coexist, and the results are compared with experimental data from the literature. Subsequently, flow induced by centrifugal buoyancy in a sealed rotating annulus is investigated using linear stability analysis, DNS and LES. It is shown that the onset of convection for a rotating cavity is similar to that for the problem of Rayleigh-Bénard convection. Analysing flow statistics for different values of the Rayleigh number, it is shown that the disk boundary layer behaves as a laminar Ekman layer, both in terms of its thickness and of its velocity profiles. This is observed even when instantaneous profiles are considered, despite the unsteadiness of the solution. The results also show that the shroud thermal boundary layer scaling is consistent with that of natural convection under gravity. Introducing an axial throughflow of cooling air, some features observed in the sealed cavity are maintained, however a strong reduction in the core temperature and a corresponding increase in the shroud heat transfer occur. The axial throughflow also promotes a significant increase in the range of frequencies observed inside the cavity.
Declaration

This thesis is the result of my own work and it has not been previously submitted, in part or whole, to any university or institution for any degree, diploma or other qualification.

Wednesday, 21 February 2018
Diogo Berta Pitz
Guildford, UK.
Acknowledgements

Firstly, I would like to thank the CAPES foundation for providing financial support for this research through the Science without Borders programme (grant no. 13507-13-3). I am also grateful to Rolls-Royce plc. and to the Thermo-Fluid Systems University Technology Centre for providing the computing resources used throughout the study.

I express my sincere gratitude to my supervisor, Prof. John Chew, whose constant support, availability, encouragement and knowledge were crucial for the successful completion of this thesis. I feel lucky for having had the opportunity to be his student during these years. I extend my gratitude to my co-supervisor, Dr. Olaf Marxen, who has helped me to widen the scope of this research and to consider more fundamental aspects of the problems I have studied.

I also would like to thank Prof. Hugh Blackburn for making the spectral element code Semtex available and for providing me guidance with its usage in the early stages of my PhD.

Many of my friends were particularly important and supportive during these years as a PhD student. First, I thank my colleagues from the office, Maz, Sulfi, Cyril, Gonzalo, Ramesh, Michel and Donato, for all the jokes and funny moments we have shared over the years. I am especially grateful to Maz for being such a great friend and for being supportive during difficult times. I also thank Leo and Lucio for all the good moments we have shared and for making me feel closer to home. Finally, I thank my friend Vinicius for his friendship, which has been very important to me in the good and in the bad times.

The encouragement, trust and love from my parents helped me to stay motivated during all these years. Thank you for helping me to be who I am today. I love you. I am also grateful to my sister Dani and to my brother Ígor for their love and support.

Last, but not least, I would like to thank Renata for her love and dedication to me during all these years. Her constant happiness, cheering and optimism were fundamental for the success of this work.
Publications

Journal papers


Conference papers


Conference abstracts


4
# Contents

1 Introduction  
  1.1 Motivation ........................................... 11  
    1.1.1 Background ........................................... 11  
    1.1.2 Simplified models ..................................... 13  
    1.1.3 Applications in Geophysics ............................... 14  
  1.2 Approaches used in Computational Fluid Dynamics ............. 15  
    1.2.1 Levels of turbulence modelling ......................... 15  
    1.2.2 Advantages of high-order numerical schemes .............. 18  
  1.3 Objectives and thesis outline ................................ 18  

2 Literature review  
  2.1 Basic concepts ........................................... 20  
    2.1.1 Turbulence ............................................ 21  
    2.1.2 Rotating flows and buoyancy effects .................... 27  
  2.2 Review of previous work ................................... 32  
    2.2.1 Flow in rotor/stator cavities ............................. 33  
    2.2.2 Buoyancy-induced flow in sealed rotating cavities .......... 37  
    2.2.3 Buoyancy-induced flow in rotating cavities with an axial through-flow ......... 44  

3 Numerical techniques  
  3.1 Spectral and spectral element methods ........................ 50  
    3.1.1 Spectral element method ................................. 52  
    3.1.2 Semtex spectral element code ............................ 56  
  3.2 Linear stability analysis .................................... 60  
    3.2.1 Iterative solution procedure ............................ 62  
    3.2.2 Stability analysis using Semtex .......................... 62  
  3.3 Numerical verification test cases ............................ 63  
    3.3.1 Centrifugal buoyancy term in Semtex ..................... 63  
    3.3.2 Stability analysis with heat transfer .................... 64  
  3.4 Conclusion ................................................... 67  

4 Flow in a rotor/stator cavity  
  4.1 Problem description ....................................... 69  
    4.1.1 DNS and LES grids ....................................... 70  
  4.2 DNS results - $Re = 4 \times 10^5$ .............................. 71  
    4.2.1 Grid-independence ...................................... 72  
    4.2.2 Results and discussion .................................. 74  
  4.3 LES results ............................................... 78
5 Onset of convection induced by centrifugal buoyancy in a rotating cavity

5.1 Problem description
5.1.1 Geometry and boundary conditions
5.1.2 Details of the computations

5.2 Linear stability analysis
5.2.1 Onset of convection
5.2.2 Linear growth of unstable solutions

5.3 Transition to the non-linear regime

5.4 Conclusion

6 Buoyancy-induced flow in a sealed rotating cavity

6.1 Problem description
6.1.1 Geometry and boundary conditions
6.1.2 Details of the computations

6.2 Results and discussion
6.2.1 Disk boundary layers
6.2.2 Shroud boundary layer
6.2.3 Heat transfer

6.3 Conclusion

7 Buoyancy-induced flow in a rotating cavity with an axial through-flow

7.1 Problem description
7.1.1 Geometry and boundary conditions
7.1.2 Details of the computations

7.2 Results and discussion
7.2.1 Kinetic boundary layers
7.2.2 Temperature profiles and heat transfer
7.2.3 Heated disks
7.2.4 Spectral analysis

7.3 Turbulent inlet

7.4 Conclusion

8 Conclusions
8.1 Summary of findings and limitations
8.1.1 Contributions
8.1.2 Limitations

8.2 Suggestions for future research

A Boussinesq approximation for centrifugal buoyancy

B Natural convection in a differentially heated tall enclosure
List of symbols

\( r, \theta, z \) \ axes of cylindrical coordinate system [m]
\( u_r, u_\theta, u_z \) \ velocity components [m/s]
\( u^*_r \) \( = u_0/\Omega r \) normalised velocity [-]
\( u^*_\theta \) \( = u_0/\Omega a \) normalised velocity [-]
\( u_0 \) \( r_{rms} \) \ velocity rms fluctuation [m/s]
\( x, y, z \) \ axes of a general coordinate system [m]
\( u, v, w \) \ velocity components [m/s]
\( a \) \ inner radius [m]
\( b \) \ outer radius [m]
\( C_m \) \ moment coefficient for a rotating disk [-]
\( d \) \ distance between disks [m]
\( d_h \) \ hydraulic diameter \( 2(a - r_s) \) [m]
\( D \) \ characteristic length [m]
\( E(k) \) \ energy spectrum [m\(^3\)/s\(^2\)]
\( f(x) \) \ general function
\( g \) \ acceleration of gravity [m\(^2\)/s]
\( g_D \) \ Dirichlet boundary conditions
\( g_N \) \ Neumann boundary condition
\( h \) \ characteristic element size [m]
\( h_p(\xi) \) \ Lagrange polynomial
\( \ell \) \ size of a large eddy [m]
\( J_e, J_i \) \ order of integration of non-linear and diffusive terms [-]
\( J^\alpha, \beta_p(x) \) \ Jacobi polynomial
\( k \) \ wavenumber [m\(^{-1}\)]
\( k_c \) \ critical wavenumber [m\(^{-1}\)]
\( K \) \ Krylov subspace dimension
\( L_P(\xi) \) \ Legendre polynomial of order \( P \)
\( M, M_\theta, M_P \) \ wavenumber cut-off for spectral vanishing viscosity [-]
\( M_t \) \ moment on a rotating disk [N m]
\( N \) \ number of modes [-]
\( N_{el} \) \ number of spectral elements [-]
\( N_\theta \) \ number of planes along \( \theta \) [-]
\( p, P \) \ polynomial order [-]
\( p \) \ pressure [Pa]
\( p^* \) \ reduced pressure [Pa]
\( \dot{q} \) \ heat transfer rate [W]
\( \dot{q}_c \) \ conduction heat transfer rate [W]
\( Q \) \ number of quadrature points for numerical integration [-]
Q criterion, used for vortex visualisation [s^{-1}]

$Q_k$ spectral vanishing viscosity kernel

$r^* = (r - a)/(b - a)$ non-dimensional radius [-]

$r_s$ shaft radius [m]

$r_m$ mean radius $(a + b)/2$ [m]

$S_0$ smoothed step function

$t$ time [s]

$t^* = \Omega t/(2\pi)$ non-dimensional time [-]

$T$ temperature [K]

$\hat{T}$ temperature in Fourier space [K]

$T_a, T_b$ temperature at $r = a$ or $b$ [K]

$T_k(x)$ Chebyshev polynomial

$u^\delta$ approximate solution

$u^H$ homogeneous part of solution

$u^D$ solution satisfying Dirichlet boundary conditions

$\hat{u}_k$ series expansion coefficient

$u_r$ friction velocity [m/s]

$v(x)$ test function

$V$ characteristic velocity [m/s]

$w_p$ quadrature weights for numerical integration

$W$ axial velocity at cavity inlet [m/s]

$z$ axial coordinate [m]

$z^* = z/d$ non-dimensional axial coordinate [-]

**Greek letters**

\(\alpha\) thermal expansion coefficient [K^{-1}]

\(\beta\) azimuthal wavenumber [m^{-1}]

\(\delta_E\) Ekman layer thickness [m]

\(\epsilon\) rate of energy dissipation [m^2/s^3]

\(\epsilon\) spectral vanishing viscosity amplitude [m^2/s]

\(\eta\) Kolmogorov length scale [m]

\(\phi_p(\xi)\) general expansion basis

\(\nu\) Kolmogorov velocity scale [m/s]

\(\kappa\) thermal diffusivity [m^2/s]

\(\lambda\) Ekman depth [m]

\(\lambda_r, \lambda_i\) real and imaginary parts of an eigenvalue

\(\rho\) density [kg/m^3]

\(\nu\) kinematic viscosity [m^2/s]

\(\mu\) dynamic viscosity [Pa s]

\(\Omega\) angular velocity [rad/s]

\(\Omega\) domain for Galerkin discretisation

\(\Omega_d\) standard elemental region

\(\Pi\) rate of energy transfer down the turbulent cascade [m^2/s^3]

\(\tau_w\) wall shear stress [Pa]

\(\tau_t\) tangential wall shear stress [Pa]

\(\xi\) coordinates for the standard elemental region [m]

\(\xi_p\) nodal values of \(\xi\) [m]
Superscripts and subscripts

\( \bar{a} \) average over time and along a homogeneous direction
\( a_0 \) reference value
\( a' \) fluctuating quantity
\( a_f \) filtered quantity
\( a^* \) residual quantity
\( a_I \) inertial frame
\( a_R \) rotating frame
\( a^+ = au_r/\nu \) quantity in wall units

Vectors and tensors

\( \mathbf{a} \) acceleration \([m^2/s]\)
\( \mathbf{D} \) differentiation matrix
\( \mathbf{D}^{SVV} \) modified SVV differentiation matrix
\( f \) general function vector
\( \mathbf{g} \) gravitational acceleration vector \([m^2/s]\)
\( \mathbf{L} \) Laplacian matrix
\( \mathbf{L}(\mathbf{u}) \) diffusive term of the momentum equations
\( \mathbf{L}(\mathbf{U}) \) linear operator of the momentum equations
\( \mathbf{M} \) mass matrix
\( \mathbf{N}(\mathbf{u}) \) advective term of the momentum equations
\( \mathbf{n} \) unit vector normal to a surface
\( \mathbf{r} \) position vector \([m]\)
\( \mathbf{T} \) discrete polynomial transform matrix
\( \mathbf{u} \) velocity vector \([m/s]\)
\( \hat{\mathbf{u}} \) velocity in Fourier space \([m/s]\)
\( \mathbf{u}' \) perturbation velocity vector \([m/s]\)
\( \mathbf{U} \) base flow velocity vector \([m/s]\)
\( \hat{\mathbf{u}}, \hat{\hat{\mathbf{u}}} \) intermediate velocity fields \([m/s]\)
\( \Omega \) rotation vector \([\text{rad/s}]\)
\( S_{ij}, \Omega_{ij} \) symmetric and antisymmetric parts of the velocity gradient tensor \([s^{-1}]\)
\( \tau_{ij} \) viscous stress tensor \([\text{Pa}]\)
\( \tau_{ij}^R \) Reynolds stress tensor \([\text{Pa}]\)

Non-dimensional quantities

\( A \) cavity aspect ratio \(d/(b-a)\)
\( R \) curvature parameter \((a+b)/(b-a)\)
\( \eta \) radius ratio \(a/b\)
\( Fr \) Froude number \(\Omega^2 r/g\)
\( Pr \) Prandtl number \(\nu/\kappa\)
\( Gr \) Grashof number \(Ra/Pr\)
\( Nu \) Nusselt number \(\dot{q}/\dot{q}_c\)
\( Nu' \) Modified Nusselt number \(q_{conv}(d/2)/k(T_b - T_{core})\)
\( Nu_d \) disk Nusselt number \(qr/k(T_{disk} - T_a)\)
\( Ra \) centrifugal Rayleigh number \(\Omega^2 r m_\alpha \Delta T(b-a)^3/\kappa \nu\)
\( Ra' \) modified Rayleigh number \( \Omega^2b\alpha(T_b - T_{core})(d/2)^3/\kappa\nu \)

\( Ra_c \) critical Rayleigh number

\( Ra_g \) gravitational Rayleigh number \( g\alpha\Delta T d^5/\kappa\nu \)

\( Re \) Reynolds number \( V D/\nu \)

\( Re_\tau \) \( u_r D/\nu \)

\( Re_\Omega \) rotational Reynolds number \( \Omega b^2/\nu \)

\( Re_\theta \) rotational Reynolds number \( \Omega r_m(b - a)/\nu \)

\( Re_z \) axial Reynolds number \( 2W(a - r_s)/\nu \)

\( Ro \) Rossby number \( W/\Omega a \)

\( Ta \) Taylor number \( 4\Omega^2(b - a)^5/\nu^2d \)

\( \Theta \) thermal Rossby number \( g\alpha\Delta T d/\Omega^2(b - a)^2 \)

**Acronyms**

- CFD computational fluid dynamics
- DNS direct numerical simulation
- uDNS under-resolved DNS
- LES large-eddy simulation
- iLES implicit LES
- RANS Reynolds-averaged Navier-Stokes
- SVV spectral vanishing viscosity
- RB Rayleigh-Bénard
- GLL Gauss-Lobatto-Legendre
Chapter 1

Introduction

In this introductory chapter the practical motivation behind the studies considered in this thesis is presented. An overview of numerical techniques and strategies used to treat turbulence when solving the equations governing fluid flow is also given, and at the end of the chapter the objectives of the research and an outline of the thesis are described.

1.1 Motivation

1.1.1 Background

In modern jet engines used in civil aviation, air coming from the atmosphere is compressed, mixed with fuel and, after exiting the combustion chamber, is expanded and transfers some of its energy to the engine turbine, before being expelled at the exit nozzle. This process of compression, combustion and expansion of air is responsible for the rotation of the engine, but has only a small direct contribution to the thrust, which is mostly created by the air that flows through the by-pass section, i.e., that does not enter the engine core. The process is illustrated schematically in Fig. 1.1 for a two-spool turbofan engine. Note that the low- and high-pressure compressors are attached to two independent shafts, which are connected, respectively, to the low- and high-pressure turbines. In some modern configurations three separate shafts are used, one of which is connected to the fan and to the low-pressure turbine. For this case the compression and expansion occur in low-, intermediate- and high-pressure stages, thus forming a three-spool configuration.

Although the operating principle of a jet engine described above is rather simple, obtaining a design able to deliver high power, good fuel efficiency, safety and reliability, and whose cost is competitive, is a complicated task. To obtain high efficiency from the cycle it is desirable to operate at a high pressure ratio, which is the ratio between the air pressure after and before compression, and at high turbine inlet temperatures. These requirements for high efficiency impose severe conditions on the engine components, as the materials are required to survive extreme temperatures and thermal and mechanical stresses without compromising the system’s reliability. Cooling of the components, in particular, is of great importance to ensure that the temperatures do not exceed the designed values. As described in more detail below, the work presented in this thesis is motivated by an application involving a jet engine cooling system.
The cooling system is responsible for supplying cold air to a range of components across the engine, such as the compressor disks and turbine blades. An example of a cooling system for a compressor is represented in Fig. 1.2. Air is bled from a certain stage into the cavities formed between the disks. This air is used for cooling the disks, and flows through the cavities formed between the disks that support the rotating compressor blades. During acceleration of the engine, the throughflow air may be hotter than the disks and the shroud, which is the outer cylindrical surface of the cavities. In this situation buoyancy in the centrifugal force field stabilises the flow and the temperature field tends to remain stratified. When the engine is running at a constant speed, on the other hand, the air entering the cavities is colder than the disks and the shroud. In this case buoyancy effects dominate the flow dynamics, as the centrifugal force induced by rotation tends to push the cold, heavier fluid, towards the outer part of the cavity, whilst the hotter fluid will tend to move inwards. The fact that this flow configuration is inherently unsteady and three-dimensional suggests that models commonly used in industry, such as RANS (Reynolds-averaged Navier-Stokes)-based turbulence models solved in steady form, are not appropriate to perform accurate investigations of buoyancy-induced flows in rotating cavities.

From a practical point of view, an engine designer needs to be able to predict the heat transfer rates on the disks in order to obtain temperature distributions. Knowledge of the temperature profiles is particularly important, as the disk temperature influences the thermal expansion of the disks, which affects thermal stresses and blade-tip clearances, i.e., the gap between the blades and the engine casing. For higher efficiency, it is desirable that such clearances are kept at a small value, but it is also important that the blades do not rub the casing during steady-state operation. This required balance indicates that disk heat transfer and temperature prediction have a direct effect on the compressor efficiency.

Another configuration of interest common in turbine internal air systems consists of a rotor/stator system, in which the shaft and one of the disks rotate, while the shroud and the other disk are stationary. In this configuration buoyancy-effects play
a secondary role, since the flow dynamics is dominated by the differential rotation of the system. Still, rotor/stator flows are challenging to study because transitional regions might exist depending on the rotational Reynolds number. Compared to rotor/rotor cavities, flows in rotor/stator configurations are relatively well understood and are commonly estimated using RANS models in engineering design.

1.1.2 Simplified models

The geometry of the rotating cavities described above is complex, since there are, for example, bolts, cobs, changes of radial length and interactions between different cavities. To study the flow physics occurring inside the cavities, however, the geometrical details of the system do not play a major role. For this reason, many studies, both experimental and numerical, rely on the use of simplified geometrical configurations to investigate the flow physics. Figure 1.3 illustrates examples of simplified geometries commonly employed in numerical investigations of rotating cavity flows. The sealed annulus shown in part (a) of the figure can be used to study buoyancy-induced flow in a sealed rotating cavity and flow in a rotor/stator cavity, whereas part (b) shows a simplified configuration of flow in a rotating cavity with an axial throughflow of cooling air. Obviously, appropriate boundary conditions must be imposed in each of these cases, and they will be presented in detail later.

In this thesis fundamental investigations of different configurations relevant to internal air systems are described. The main focus is on buoyancy-induced flows, as such flows have proven to be challenging to engine designers and are not well understood. To study buoyancy-induced flows from a fundamental point of view, a sealed rotating annulus bounded by two disks is considered, as shown in Fig. 1.3(a), where the disks are insulated and the inner cylindrical surface (shaft) is cooled, whilst the outer surface (shroud) is hot. Although this configuration does not include an axial throughflow, it contains the basic ingredients necessary to investigate the effect of buoyancy, namely a centrifugal acceleration induced by the system rotation, and a positive temperature gradient along the radial direction. A configuration including an axial throughflow of cooling air with a central shaft is also considered. A third
configuration consisting of a rotor/stator system, in which buoyancy effects are neglected, is also studied.

1.1.3 Applications in Geophysics

As discussed in the previous paragraphs, the rotating cavities of internal air systems can be modelled as a rotating annulus with certain boundary conditions. This simplified system is also of interest to other scientific communities, and has been the object of many studies in the past decades. In particular, convection processes in stars and in the earth’s core, and the global circulation of the atmosphere, can be modelled in a laboratory using a rotating annulus, and are relatively straightforward to treat numerically.

In the case of convection in the liquid core of a planet, for example, the heat flux points outwards, whereas the dominant body force is gravity, which points inwards. Although a spherical geometry gives a better representation of this system, the use of an annulus makes the analysis much simpler both numerically and experimentally, while keeping the main ingredients of the full system, namely rotation and a radial temperature gradient. Still, the situation of a rotating annulus with a gravitational force directed radially inwards is hard to reproduce experimentally. The alternative used is to reverse the direction of the heat flux, so that it is directed inwards, and apply a rapid rotation to the system such that the resulting centrifugal acceleration is larger than gravity, which is aligned with the axis of rotation. The resulting system is then equivalent to that shown in Fig. 1.3(a), although in many studies the end-walls are not parallel.

The global circulation of an atmosphere can also be modelled using a rotating annulus, however a different rationale is used when compared to the case of a planet’s core. Considering the global circulation occurring between the equator and the poles, it is reasonable to consider a cylindrical annulus with cooled and heated inner and outer surfaces, respectively, to model the problem. In this case the action of gravity along the axis of rotation is also considered, so that the system will have different
dynamics depending on whether convection is dominated by axial gravity or by the action of the centrifugal acceleration. The study of cases where axial gravity is present, which gives rise to the baroclinic instability, is out of the scope of the present work, although it is considered in two test cases for numerical verification purposes. For further details about geophysical flows, see, e.g. Cushman-Roisin & Beckers \[3\].

When investigating the problem of a differentially heated rotating annulus either numerically or experimentally, engineers and geophysicists are generally interested in distinct aspects of the flow. While for engineering purposes one seeks predictions for the heat transfer rates and temperature on the walls, as well as information about the general structure of the resulting turbulent flow, geophysicists are interested in the flow transitions that occur when certain control parameters are varied, or in how modes with different azimuthal wavenumbers interact. In the work presented throughout this thesis, aspects of practical and fundamental interest are studied, as detailed in section 1.3.

1.2 Approaches used in Computational Fluid Dynamics

Computational Fluid Dynamics (CFD) is the study of fluid flow by means of solving a given set of governing equations numerically on a computer. As a first step, one needs to define what equations govern the problem of interest. In many cases the Navier-Stokes equations, either in compressible or incompressible form, combined with the mass and energy conservation equations, are sufficient to model a problem accurately. Situations in which non-Newtonian fluids or reacting flows are modelled, for example, require additional equations. When molecular effects are not negligible and the hypothesis that the fluid is a continuum fails, different approaches are required, such as statistical methods. The discussion presented in this chapter is limited to the Navier-Stokes equations written in incompressible form.

Once the set of governing equations is defined, a geometrical model has to be specified, as well as appropriate boundary conditions and fluid properties. Then, the governing equations have to be discretised and solved using a given technique, and finally, after the solution is obtained, the results are analysed. When the flow is turbulent, which is usually the case in both industrial applications and in nature, a variety of approaches can be used to treat turbulence. In this section aspects regarding the discretisation of the governing equations and the choice of turbulence treatment are presented. Specifically, a brief overview of the ideas behind turbulence modelling and model-free approaches is given, and the advantages of using high-order spatial discretisations to study turbulent flows are discussed.

1.2.1 Levels of turbulence modelling

Turbulent flows, in contrast with laminar flows, have a complex structure and are inherently chaotic and characterised by having a large range of spatial and temporal scales. For centuries scientists and engineers have been trying to formulate theories to describe turbulent flows, and although some of these have helped to elu-
candidate certain aspects of turbulence, no theory is universally valid for all problems encountered in industry and in nature. When solving the governing equations on a computer, the large separation between the large eddies, which contain the greatest part of the kinetic energy, and the small eddies, where viscous dissipation occurs, becomes a problem. If one is interested in solving all the spatial and temporal scales of the flow, the mesh has to be refined enough to capture the smallest eddies, i.e., the Kolmogorov scale. Additionally, the time-step employed to integrate the governing equations has to be small enough to capture high-frequency events. Such technique is called direct numerical simulation (DNS), and is restricted to relatively low-Reynolds number flows and geometries of moderate complexity. If a solution can be obtained using DNS, a wealth of information virtually impossible to be obtained from an experiment becomes available, which helps to study the flow in detail and possibly allows for the development and improvement of theories and models of turbulence.

In most industrial flows of practical interest, obtaining a direct solution to the Navier-Stokes equations is not viable with the computer hardware available today. Even when a DNS can be afforded, the calculation may take from several days to several months to complete on a powerful computer, which makes it prohibitively expensive for design purposes. Therefore, numerical solutions of practical interest rely on the use of turbulence models. When a turbulence model is used, it is assumed that turbulence can be represented by that model, which eliminates the need to solve all the flow scales. Obviously, no model is capable of capturing all the phenomena occurring in the real flow, and for this reason the use of any turbulence model comes with a penalty on how accurately the new set of equations captures the flow physics.

Most turbulence models commonly used in industry are based on the Reynolds-averaged Navier-Stokes (RANS) equations, which are obtained from a decomposition of the turbulent flow field, denoted by \( u \), into an average field \( \overline{u} \) and an instantaneous field \( u' \), so that \( u = \overline{u} + u' \). This decomposition is illustrated in generic form in Fig. 1.4(a) - although this specific curve was obtained from a DNS of flow in a rotor/stator cavity, which will be presented in Chapter 4, it provides a good representation of what a turbulent signal looks like. Note that the Reynolds decomposition itself does not imply on a loss of generality, i.e., no modelling assumption is made.

![Figure 1.4: Generic illustrations of (a) the Reynolds decomposition, and (b) a filtered velocity field.](image)
When the decomposition is substituted into the Navier-Stokes equations, an equation for the mean flow $\mathbf{u}$ is obtained, but it turns out that an extra term containing the velocity fluctuation $u'$ arises, the so-called Reynolds stress tensor. The aim of any RANS-based turbulence model is to model the Reynolds stress tensor, and this can be done with various degrees of complexity. Unfortunately, many phenomena such as transition, flow separation and buoyancy cannot be represented accurately with turbulence models. On the other hand, RANS-based models have proven to be valuable tools for use in industry as they are able to provide sufficiently good results, although in many cases a manual calibration of the constants of a given model is required.

An alternative approach to turbulence modelling consists of the large-eddy simulation (LES) technique. As the name suggests, with this approach the large structures of the flow are resolved, whilst the smaller scales are modelled. The reasoning for this is that most of the flow's kinetic energy is concentrated in the large scales, and also because they make the greatest contribution to momentum and heat transport. The smallest scales, on the other hand, are only responsible for the viscous dissipation of the energy transferred from the large eddies, and have a more universal character. In practice, in LES the flow variables are filtered, so that the filtered velocity field is represented by $u_f$. An example of what a filtered velocity field looks like is shown in Fig. 1.4(b). Note that when a filter is applied, high-frequency information is lost, so that only the relatively slower events, which are associated with the larger scales, are captured. The governing equations are filtered by applying a decomposition of the form $\mathbf{u} = u_f + u^r$, where $u^r$ denotes a residual velocity. When this decomposition is applied to the momentum equations, an equation analogous to that obtained from the Reynolds decomposition is obtained, but it governs the filtered, rather than the averaged, velocity field. A residual stress tensor also arises, and requires modelling. When compared to the RANS approach, LES is considerably computationally more expensive, but is able to provide much greater accuracy, since modelling is applied at the small scales only. With the increasing growth of computing power, LES is starting to be used in industry to study problems where RANS-based models fail, or where greater accuracy is required.

When no model is used to solve the governing equations, the simulation can be either a DNS, when all the flow scales are resolved, or an implicit LES, also referred to as under-resolved DNS. In an implicit LES no turbulence model is used, but the spatial and/or temporal resolution is not fine enough to capture all the flow scales, and therefore the mesh serves as an implicit filter, adding a numerical dissipation to the solution. It is also possible to add a controlled amount of artificial dissipation, instead of relying on the grid to do this automatically. One technique of this kind is the spectral vanishing viscosity (SVV), which is used in the context of spectral methods to use an augmented viscosity at high wavenumbers. This technique is able to stabilise calculations of turbulent flows and provides, in many cases, results with accuracy comparable to that of a DNS at large scales. Throughout this study, DNS and implicit LES are employed using the SVV technique, which is described in detail in Chapter 3.
1.2.2 Advantages of high-order numerical schemes

In the context of industrial flows, the standard methodology in use for several years consists of discretising the Navier-Stokes equations with the finite volume method. This method is robust in the sense that it can be applied to a broad range of problems and supports unstructured grids. Commercial codes such as ANSYS Fluent, as well as open-source packages like OpenFOAM, employ the finite volume discretisation. The method is generally implemented with second-order accuracy and using an implicit time-integration scheme, so that unsteady solutions can be obtained much faster than it is possible with semi-implicit schemes, although this compromises the solution accuracy. As the problems studied in industry become more complex, in terms of the level of detail required to achieve design improvements, the use of high-fidelity numerical tools becomes desirable.

Numerical solutions of turbulent flows using finite volume methods are generally obtained using a turbulence model. For DNS and implicit LES, it is desirable to use a numerical discretisation able to provide higher accuracy. Finite volume methods are usually second-order accurate in space, which would require extremely refined grids in the context of DNS and implicit LES. High-order methods, on the other hand, are able to provide greater accuracy, since diffusion and dispersion errors are lower than for low-order schemes. The use of high-order schemes is particularly relevant when long time-integration periods are required, which is usually the case for DNS and LES applications, since they are associated with a smaller computational effort than low-order schemes in such cases (Karniadakis & Sherwin [4]). Spectral methods, in particular, are able to provide the so-called spectral convergence, which occurs when the error decreases sharply as the number of grid points is increased, i.e., it does not decay at a constant rate. In this work a spectral element discretisation is employed, which combines the flexibility of finite element methods with the high accuracy of spectral methods, thus allowing the use of complex geometries. Further details about spectral methods and a detailed description of the implementation used in this study are given in Chapter 3.

1.3 Objectives and thesis outline

In this study a fundamental investigation of buoyancy-induced flows occurring in gas turbines internal air systems is performed. A spectral element-Fourier technique is used to discretise the governing equations, which are solved in incompressible form. Both DNS and implicit LES are used, and insight into the flows which could not be obtained using turbulence models is achieved. Following consideration of a rotor/stator disk cavity flow, the thesis focuses on the case of a sealed, differentially heated rotating annulus. As this excludes the effect of inlets and outlets, DNS solutions are obtained. The onset of convection from the conductive state to the fully turbulent state is also investigated in detail. Following the sealed cavity analyses, results from an investigation for the case of a rotating cavity with an axial throughflow of cooling air are presented.

The thesis is divided into eight chapters, and an outline of the next seven is given below.

- Chapter 2 presents a brief review of the basic concepts of turbulent flows and
flows in rotating systems. A literature review of the studies most relevant to this thesis is also given.

- In Chapter 3 the spectral element method is presented in a general manner, and aspects of the method relevant to the implementation used in this study are described in greater detail. A description of the time-stepper method employed to perform linear stability analysis is also presented. Additionally, results are presented for test cases used for numerical verification.

- DNS results for flow in a rotor/stator cavity are presented in Chapter 4. Cases where the rotor boundary layer is laminar, transitional and fully turbulent are considered.

- Chapter 5 is focused on a study of the onset of convection induced by centrifugal buoyancy in a rotating cavity. Linear stability analysis is employed to analyse the onset of instabilities, whereas DNS is used to investigate the transition to the non-linear regime.

- In Chapter 6 the same setup used in Chapter 5 is used, but the analysis is focused on the non-linear regime. Mean profiles and heat transfer rates on the cylindrical walls are reported and compared with experimental data.

- In Chapter 7 implicit LES is used to study the problem of a heated rotating cavity with an axial throughflow. Mean profiles are reported and particular attention is given to the disk and shroud boundary layers, and to the interaction between the axial flow and the buoyancy-induced flow.

- A summary of the contributions of the thesis and recommendations for future work are provided in Chapter 8.
Chapter 2

Literature review

This chapter is divided in two parts. In section 2.1 an overview of turbulent flows is given, followed by a presentation of concepts of fundamental importance in the study of rotating flows, including the description of a laminar Ekman layer and the Taylor-Produman theorem. In the second part of the chapter, section 2.2, previous studies for flows in rotor/stator cavities, buoyancy-induced flows in sealed rotating cavities, and flows in heated rotating cavities with an axial throughflow, are reviewed and discussed.

2.1 Basic concepts

The motion of a Newtonian, incompressible fluid of kinematic viscosity $\nu$ is governed by the mass conservation equation and the Navier-Stokes, or momentum equations, which are given, in dimensional form, by

$$\nabla \cdot \mathbf{u} = 0, \quad (2.1)$$

$$\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} = -\nabla (p/\rho) + \nu \nabla^2 \mathbf{u}, \quad (2.2)$$

where $\mathbf{u}$ is the velocity vector, $\nabla p$ is the pressure gradient and $\rho$ is the fluid density. The equations above constitute a system of partial differential equations, which require an initial condition and appropriate boundary conditions. The Navier-Stokes equations can be interpreted as a balance between the fluid acceleration (left-hand side) and pressure and viscous forces (right-hand side) acting on the fluid. Note that, for a Newtonian fluid, it is assumed that the stress tensor is proportional to the rate of deformation, where the constant of proportionality is the dynamic viscosity $\mu = \nu \rho$.

Exact solutions to these equations are rare and of limited use for the analysis of real flow problems. For flow through a circular pipe, for example, an analytical solution can easily be obtained if the flow is laminar and fully developed, in which case the Reynolds number,

$$Re = \frac{VD}{\nu}, \quad (2.3)$$

is small, $Re \lesssim 2000$. In the definition of $Re$, $V$ and $D$ represent a characteristic velocity and length of the flow, and $Re$ can be interpreted as a ratio between inertial
and viscous effects. If $Re$ is relatively small, viscous effects dominate the dynamics, and the assumption of laminar and fully developed flow allows one to neglect the advective term of the momentum equations, $u \cdot \nabla u$. Without the advective term, which is non-linear, the equations become much easier to solve. In practice, however, most flows are turbulent, i.e., inertial effects cannot be neglected, and therefore the advective term must be kept. Still considering the simple example of a pipe flow, when $Re$ is increased above $Re \approx 2000$, transition to turbulence sets in, and the flow becomes fully turbulent for $Re \gtrsim 2300$. In contrast with the laminar flow occurring at low $Re$, which has a very simple structure, the transitional and turbulent regimes develop extremely rich dynamics. Mathematically, such complex behaviour arises from the advective term and from the non-linear interactions it promotes. For further details, the reader may consult refs. [5, 6].

For incompressible flow, an equation of state for the pressure does not need to be specified. This is so because of the solenoidal nature of the velocity field, which allows for a relationship between velocity and pressure to be obtained by taking the divergence of Eq. 2.2. As shown in more detail in chapter 3, a Poisson equation relating pressure and velocity is obtained, which ensures that mass conservation is satisfied. Physically, this implies that pressure affects the entire velocity field, i.e., it has a non-local effect. Alternatively, one can interpret this effect as a result of the fact that pressure waves travel infinitely fast in an incompressible fluid.

2.1.1 Turbulence

As most of the flows considered in this thesis are turbulent, it is useful to provide some fundamental concepts of turbulence and an overview of the theories generally employed to study these flows.

Turbulent flows are characterised by chaotic motions occurring on a broad range of temporal and spatial scales. This means that turbulent eddies of different sizes interact, having different turnover times – large eddies break up at a relatively low frequency, while the smallest eddies are associated with high frequencies. This chaotic behaviour of turbulent flows can be observed, for example, by measuring a given quantity (say, velocity or temperature) in a specific location of the domain of interest. If repeated measurements are taken and plotted in a graph of the quantity measured versus time, it would be observed that all realisations are different, although they would oscillate around the same mean, assuming that the flow is statistically steady. In a numerical simulation, on the other hand, a given solution is reproduced if the calculation is initialised with the same initial conditions, since the governing equations are deterministic. If the initial condition is changed, however, a different solution will be obtained, although, again, the mean quantities will remain unchanged. In fact, turbulent phenomena are extremely sensitive to initial conditions, so turbulent flows can be regarded as dynamical systems. Many of the tools used in chaos theory, such as phase-space diagrams, Poincaré maps, and Lyapunov exponents, are employed to study transition to turbulence, i.e., the onset of chaotic motion.

One characteristic of turbulent flows is that the higher the Reynolds number, the larger the separation between the flow scales. This will be illustrated in the next subsection, where expressions for the Kolmogorov scales are presented. Such behaviour can be seen in many real flows, as it is noticeable that the structure of
the flow becomes finer when the Reynolds number is increased. Figure 2.1 shows an example of this for flow in a rotor/stator cavity, which is investigated in detail in Chapter 4. The figures show iso-surfaces of the $Q$-criterion near the rotor surface at $Re = 4 \times 10^5$ and $Re = 10^6$; it is clear that the structure becomes much finer when $Re$ is increased. As shown in Chapter 4, the rotor boundary layer is not fully turbulent for $Re = 4 \times 10^5$, thus explaining the presence of large-scale vortices observed in Fig. 2.1(a) at intermediate radial positions. The contours are coloured by the tangential velocity, and the $Q$-criterion is a scalar quantity obtained from the velocity gradient tensor which is commonly used for vortex identification (more details are given in Chapter 4).

**Energy cascade and Kolmogorov scales**

In a turbulent flow, most of the energy is contained in the large eddies, whose nature is dependent on specific details of the flow, such as geometry and boundary conditions. The small scales, in contrast, contain little energy and it is generally accepted that they have a universal character, i.e., their behaviour does not depend on the macroscopic details of the flow. Intermediate eddies are also assumed to have an universal behaviour.

The energy contained in the large eddies is transmitted to eddies of smaller size, which then transfer their energy to other eddies of even smaller size, and so on, until the smallest eddies of the flow are reached, and energy is dissipated via the viscosity. This transfer of energy from the large to the small scales is termed energy cascade, and was originally proposed by Richardson [7]. The idea of an energy cascade plays a crucial role in the study of turbulent flows. Figure 2.2(a) shows a simplified illustration of this process, and Fig. 2.2(b) shows a generic representation of an energy spectrum for a turbulent flow.

The energy-containing (large) eddies are only weakly affected by shear stresses, so that their behaviour is not influenced by viscosity. The rate of destruction of energy $u^2$ of a large eddy of size $\ell$ is given by,
Figure 2.2: (a) Sketch of the eddy break-up mechanism. Adapted from Frisch & Donnelly [8]. (b) Generic representation of the energy spectrum for a turbulent flow.

\[
\frac{du^2}{dt} \sim - \frac{u^2}{\ell/u} \sim - \frac{u^3}{\ell},
\]

so that it only depends on the velocity and size of the eddy. Richardson hypothesised that energy is transferred down the cascade at a constant rate \( \Pi \), which is given by the rate of destruction of the large eddies,

\[
\Pi \sim \frac{u^3}{\ell}.
\]

When the energy coming from the large and intermediate eddies reaches the smallest scales, it is dissipated at a rate \( \epsilon \), which must be equivalent to the rate of energy production by the large eddies,

\[
\epsilon \approx \Pi \sim \frac{u^3}{\ell}.
\]

The relationships presented above are important in the derivation of the Kolmogorov scales for the size and velocity of the smallest eddies. At very small scales, viscous effects are important, so that one can assume that a local Reynolds number calculated in terms of the length scale \( \eta \) and the velocity \( \nu \), is of order one, that is,

\[
\frac{\nu \eta}{\nu} \sim 1.
\]

Now, the rate of energy dissipation depends on the viscosity and on the strain-rate tensor \( S_{ij} \),

\[
\epsilon = 2\nu S_{ij}S_{ij} \sim \frac{\nu \nu^2}{\eta^2},
\]

thus providing a relationship between \( \epsilon \) and the length and velocity of the smallest scales. Combining Eqs. 2.6 and 2.8 and considering the relation 2.7 one obtains expressions relating the large scales with the small, or Kolmogorov, scales:

\[
\eta \sim Re^{-3/4} \ell \sim \left(\frac{\nu^3}{\epsilon}\right)^{1/4},
\]
CHAPTER 2

\[ \nu \sim Re^{-1/4}u \sim (\nu \epsilon)^{1/4}, \quad (2.10) \]

where the Reynolds number is defined in terms of the integral (large) scales. The expressions above are useful since they provide estimates for the dissipation scales based solely on the Reynolds number and on integral quantities. They also give us the information that when the Reynolds number is increased, the scale where dissipation occurs becomes smaller. This has important consequences for the numerical simulation of turbulent flows, as finer grids need to be used when \( Re \) is increased if one is interested in capturing the Kolmogorov scale.

Kolmogorov \[9\] suggested the existence of an intermediate scale, where the size of the eddies is small compared to \( \ell \) but large compared to \( \eta \), and where the distribution of kinetic energy is independent of the viscosity \( \nu \) and also on the large scales and on the anisotropy they introduce. This region of the eddy-size spectrum is called the \textit{inertial subrange}, and is thought to be universal as long as the Reynolds number is large enough. When this hypothesis is formulated in terms of the energy spectrum, \( E(k) \), the following power-law dependency is obtained,

\[ E(k) = \alpha \epsilon^{2/3} k^{-5/3}, \quad (2.11) \]

where \( \alpha \) is a constant and \( k \) is the wavenumber. The \textit{five-thirds law} proposed by Kolmogorov is observed for flows at high Reynolds numbers, although the extent of its validity, in terms of the range of \( k \) for which it holds, can change for different flows. Finally, it is important to note that the theory presented in this section is valid for homogeneous, statistically steady turbulence. Experience shows, however, that it can be applied to many anisotropic flows, provided that regions sufficiently away from the walls are considered.

\textit{Reynolds stress tensor}

In Chapter \[1\] a brief discussion of the RANS equations was presented, and it was noted that when the Reynolds decomposition is applied to the governing equations an extra term, called the Reynolds stress tensor, arises. It should be noted that the Reynolds stress tensor \( \tau_{ij}^R \) does not represent a physical stress acting on the fluid, in contrast with viscous stresses, but it has units of stress and it is rather convenient to analyse its effect in a flow field as if it was a real stress. For a three-dimensional flow, \( \tau_{ij}^R \) is a \( 3 \times 3 \) tensor given by Eq. 2.12, where the primes denote velocity fluctuations and the overbar denotes an ensemble average.

\[ \tau_{ij}^R = -\rho u'_i u'_j, \quad (2.12) \]

Following Davidson \[5\], \textit{“the Reynolds stresses represent the mean momentum fluxes induced by the turbulence”}. Therefore, \( \tau_{ij}^R \) effectively removes energy from the mean flow and feeds it to the turbulence, so that it is connected to the turbulent kinetic energy. From the perspective of numerical simulations that do not use a turbulence model, as it is the case of the simulations considered in this thesis, the evaluation of the Reynolds stress tensor is important mainly for three reasons. First, it gives good physical insight into a given problem, as it provides information about the distribution of turbulent kinetic energy and intensity of fluctuations. Second, it
gives a good idea of whether a simulation is converged or not, both in terms of
the mesh used and of the time period used to collect the averages, since the terms
of $\tau_{ij}^R$ are of second order, and therefore take longer to converge than the average
of a first-order quantity, such as the velocity components. And finally, since $\tau_{ij}^R$
is the quantity modelled in RANS-based turbulence models, it is desirable to have
accurate profiles of its components to evaluate the suitability of a given model.

**Turbulent boundary layers**

In wall-bounded flows a boundary layer develops due to viscous effects, so that the
relative velocity of the fluid is zero at the wall. At the vicinity of the wall, the flow
is effectively laminar and viscous effects dominate, while in regions away from the
wall viscous stresses are negligible, and the fictitious Reynolds stress plays a major
role, transferring energy from the mean flow to the turbulence. In order to have a
consistent measure of what distance from the wall should be regarded as near or far
from a wall, it is useful to define a non-dimensional distance $y^+$ depending on the
viscous stress $\tau_w$ near the wall. To do that, a friction velocity is defined as,

$$u_\tau = \sqrt{\frac{\tau_w}{\rho}},$$

(2.13)

where it should be noted that, although $u_\tau$ does not represent an actual velocity,
it is related to the velocity gradient on the wall. Having defined this quantity, the
dimensionless wall distance is,

$$y^+ = \frac{yu_\tau}{\nu},$$

(2.14)

with $y$ being the dimensional distance from the wall. The concept of $y^+$ is of funda-
mental importance for numerical simulations, as it provides a measure of how well a
given grid is able to cover the near-wall region. When designing a mesh to be used
in the calculation of a turbulent flow, it is important to check that the resolution is
of order $\Delta y^+ = 1$ at the wall. This requires the knowledge of $\tau_w$, which is not known
a priori, except for canonical cases (such as channel and pipe flow), for which there
are correlations between the mean flow velocity and the friction velocity. In most
cases the value of $u_\tau$ is obtained iteratively, i.e., it is computed using a more or less
arbitrary grid, and then the mesh is redesigned so that there is sufficient resolution
near the wall.

For wall-bounded shear flows, the boundary layer can be split into three regions,
as shown in Fig. 2.3(a). In the specific case illustrated in the figure, the profile
was obtained from a DNS of flow through a circular pipe at a Reynolds number
$Re = 5300$ using the numerical technique presented in Chapter 3. Similar profiles
are obtained for many other wall-bounded shear flows. Very close to the wall,
the velocity $u^+$, which corresponds to the axial velocity normalised by the friction
velocity, is equal to $y^+$ up to $y^+ \approx 5$. This linear behaviour occurs because in
this region the wall shear stress is approximately constant. For intermediate values
where $y^+ \gg 1$, but still $y$ is much smaller than the pipe diameter (or other integral
length scale of the problem), a logarithmic region is identified. The region between
the viscous sublayer and the log-law region is referred to as buffer layer. Obviously,
the profiles shown and the discussion presented are valid only in the time-average
sense, as the instantaneous profiles are chaotic, as shown in Fig. 2.3(b).
Energy source for the turbulence

Figure 2.2 illustrates the mechanism by which energy is transferred from the large to the small scales in a turbulent flow. It is also important to consider the source of energy which sustains the chaotic motion. In shear flows a mean flow exists and, for laminar flow, carries all the kinetic energy of the flow. For turbulent flow, on the other hand, energy is transferred from the mean flow to the fluctuations, so that the total kinetic energy can be split into two terms, one corresponding to the mean flow energy and another related to the energy of the fluctuations. This energy transfer from the mean flow to the fluctuations occurs via shearing of vortices, which are moved around, twisted and rotated by the mean flow. The region near the wall, where the velocity gradients are very high, can be regarded as a generator of vorticity, which is then transported to the other regions of the flow. In short, in a shear flow the turbulence is sustained by the mean flow, which continually deforms the turbulent structures of the flow.

If a mean flow does not exist, then the energy to sustain the flow must come from some other source. In particular, a body force such as gravity or a centrifugal force can also provide energy to a turbulent flow. For the problem of flow in a sealed rotating cavity studied in this thesis, the centrifugal force induces a potential energy in the flow, which is then converted to kinetic energy, thus inducing a mean motion, which will eventually transfer energy to the turbulent fluctuations. Note that for this to happen a density gradient must exist, otherwise no potential energy will be generated by the centrifugal force. Once the flow is established, other mechanisms such as vorticity generation near a wall also occur, but it should be emphasised that buoyancy acts as the primary energy source for the flow.
2.1.2 Rotating flows and buoyancy effects

In flows subject to rotation it is often convenient to introduce a coordinate system that rotates with the domain of interest. When the Navier-Stokes equations are written in a rotating, non-inertial reference frame, two extra terms appear (assuming that the rotation rate is constant): a centrifugal force, which is irrotational and can be absorbed into the pressure gradient term, and the Coriolis force, which is responsible for many of the interesting phenomena observed in rotating flows. In this section the governing equations in a rotating reference frame are presented, and some general theories derived from them are discussed. Additionally, the Boussinesq approximation is formulated to account for buoyancy due to the effects of rotation.

**Governing equations in a rotating frame**

Letting \( \Omega = (0, 0, \Omega) \) denote the constant angular velocity of a system, the derivative of a given variable \( A \) in an inertial frame is related to the derivative in the rotating frame by,

\[
\left( \frac{dA}{dt} \right)_I = \left( \frac{dA}{dt} \right)_R + \Omega \times A. \tag{2.15}
\]

If the equation above is applied to the position vector \( r \), then the following relation is obtained between the velocities in the two frames,

\[
u_I = \nu_R + \Omega \times r. \tag{2.16}
\]

In the context of the laws of motion it is necessary to obtain a relation between the accelerations in the two frames, denoted by \( a \), which can be easily derived applying the operator 2.15 to Eq. 2.16,

\[
a_I = \left[ \left( \frac{d}{dt} \right)_R + \Omega \times \right] \left( \nu_R + \Omega \times r \right) = a_R + 2\Omega \times \nu_R + \Omega \times (\Omega \times r). \tag{2.17}
\]

It should be noted that the transformation presented above accounts for the total time derivative of the velocity, so that it must be applied to the material derivative of the velocity field in the Navier-Stokes equations, i.e., to the local rate of change in velocity and to the convective term. Using Eq. 2.17 and dropping the subscript from \( \nu_R \), the momentum equation in a rotating frame rotating at constant angular velocity \( \Omega \) is:

\[
\frac{\partial \nu}{\partial t} + \nu \cdot \nabla \nu = -\nabla \left( \frac{p}{\rho} \right) + \nu \nabla^2 \nu - 2\Omega \times \nu - \Omega \times (\Omega \times r). \tag{2.18}
\]

Since the Laplacian of \( \Omega \times r \) is zero, no extra term arises from the viscous part of the equation. The continuity equation for a rotating frame is identical to Eq. 2.1, with \( \nu \) being evaluated in the rotating frame.

The centrifugal acceleration can be written in gradient form, allowing for it to be combined with the pressure to form a reduced pressure \( p^* \):

\[
p^* = \frac{p}{\rho} - \frac{\left| \Omega \times r \right|^2}{2}. \tag{2.19}
\]
It is then convenient to rewrite the Navier-Stokes equations in terms of the reduced pressure, giving:

$$\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} = -\nabla p^* + \nu \nabla^2 \mathbf{u} - 2\Omega \times \mathbf{u}. \quad (2.20)$$

The fact that the centrifugal force can be absorbed into the pressure gradient term means that it does not play a dynamical role in the flow, and can be neglected when running a numerical simulation (only the pressure will be affected in that case). The Coriolis term $2\Omega \times \mathbf{u}$, on the other hand, is of great importance and is key to the explanation of many phenomena occurring in rotating systems. By the definition of the cross product, it is evident that the Coriolis force induces motion perpendicular to the direction of velocity and to the axis of rotation. Thus, for a cylindrical coordinate system rotating about its axis, fluid moving radially is deflected in the tangential direction, whereas fluid moving in the tangential direction is deflected radially. The direction of the deflection is such that angular momentum is conserved, so that for radially inward motion the tangential velocity increases, and reduces when the fluid moves radially outwards. Note that the motion induced by the Coriolis force will itself induce another motion, in the opposite sense of the original. This feature implies that the Coriolis force is able to sustain inertial waves, which have a significant impact on turbulent flows [5].

**Taylor-Proudman theorem**

One important result in the study of rotating flows is the Taylor-Proudman theorem, which states that rapidly rotating flows tend to be two-dimensional in the radial-tangential plane. The theorem can be demonstrated starting from the momentum equations in a rotating frame, Eq. 2.20, using the following assumptions:

i) the Coriolis force is much larger than the advective term;

ii) viscous effects can be neglected;

iii) steady motion is assumed.

Under these assumptions, Eq. 2.20 becomes:

$$-\nabla p^* = 2\Omega \times \mathbf{u}. \quad (2.21)$$

This result, often referred to as *geostrophic balance*, tells us that the Coriolis force is balanced by a pressure gradient when the three assumptions listed above are satisfied. Obviously, near a solid boundary the assumption of inviscid flow is not valid, but sufficiently away from the walls it is reasonable to neglect viscous effects, provided that the rotation rate is large. The hypothesis of steady motion might seem distant from reality, as turbulent flows are highly intermittent. However, experimental observations indicate that the results drawn from this assumption can be applied to rapidly rotating flows.

The Taylor-Proudman theorem is obtained, mathematically, by eliminating the pressure from Eq. 2.21 which can be achieved by taking the curl on both sides. Since the pressure is a scalar, $\nabla \times (\nabla p^*) = 0$. The remaining equation is then $\nabla \times (\Omega \times \mathbf{u}) = 0$. From vector calculus, this expression can be expanded into four
terms, but considering that the velocity field is solenoidal, and that the rotation vector $\Omega$ is constant, the following result is obtained:

$$(\Omega \cdot \nabla) u = 0.$$ 

(2.22)

Considering that the rotation vector is aligned with the axial direction $z$, one can conclude from Eq. 2.22 that the axial velocity does not change along $z$, i.e., $\partial_z u_z = 0$, meaning that the motion is two-dimensional. This result is remarkable and can be observed experimentally in many rotating flows. One classical example is the development of Taylor columns in a rotating cylindrical tank. If a round object is placed at the bottom of the tank, a vertical column spanning the entire axial length of the tank is formed. Now, if the object is moved along the bottom of the tank, the vertical column will move too, as if it was solid. This phenomenon can be explained by the Taylor-Proudman theorem, in the sense that the fluid inside the column cannot be strained axially.

For the problem of buoyancy-induced flow in a sealed rotating annulus, which is studied in Chapters 5 and 6, it will be shown that, in regions away from the end-wall disks (normal to the axis), there is very little variation in the flow field along the axial direction, which is consistent with the Taylor-Proudman theorem. If the theorem is valid even when no slip end-walls are present, one might wonder whether they play any significant role in a rotating system. As it has been shown in previous studies, end-walls induce precessing patterns in buoyant rotating flows, due to viscous effects in the near-wall region. Additionally, as shown in Chapter 5, in certain conditions a limit-cycle motion develops when the disks are present – such oscillatory motion in the rotating frame is entirely absent if periodicity is assumed along the axial direction.

Thus, it can be concluded that the Taylor-Proudman theorem provides valuable insight into rapidly rotating flows. However, near a solid wall, where the theorem is no longer valid, a boundary layer develops, and the phenomena it promotes are propagated throughout the entire axial extent of the cavity as a consequence of the Taylor-Proudman theorem.

**Ekman layer**

Near a rotating disk surface a boundary layer develops, and it can be shown analytically that it has different properties in comparison with boundary layers for non-rotating flows. Specifically, a transverse velocity component appears, and the angle of the velocity vector varies with the distance from the wall. This boundary layer is commonly referred to as an Ekman layer.

Starting again from Eq. 2.20, and restricting our attention to the boundary layer region, the following assumptions are made:

i) the Coriolis force is much larger than the advective term;

ii) the velocity is a function of the distance $z$ from the wall only;

iii) the normal velocity is zero;

iv) steady motion is assumed.
To obtain an analytical solution for the Ekman layer, the momentum equations are then written in component form, where \( x \) denotes the main flow direction, \( y \) the transverse direction parallel to the rotation surface, and \( z \) the direction normal to the disk. These coordinates are associated with the velocity components \( u \), \( v \) and \( w \), respectively. Under the assumptions above, the three components are:

\[
\begin{align*}
-2\Omega v &= -\frac{\partial p^*}{\partial x} + \nu \frac{\partial^2 u}{\partial z^2}, \\
2\Omega u &= -\frac{\partial p^*}{\partial y} + \nu \frac{\partial^2 v}{\partial z^2}, \\
0 &= -\frac{\partial p^*}{\partial z}.
\end{align*}
\tag{2.23}
\]

The third component provides the useful information that the pressure does not vary along \( z \). Sufficiently away from the wall, viscous effects can be neglected, the main velocity component is a constant, \( U \), and the transverse component is zero. Thus, the pressure gradient in \( x \) and \( y \) are,

\[
\begin{align*}
\frac{\partial p^*}{\partial x} &= 0, \\
\frac{\partial p^*}{\partial y} &= -2\Omega U,
\end{align*}
\tag{2.24}
\]

which can be used directly in Eqs. 2.23(a) and (b), giving:

\[
\begin{align*}
-2\Omega v &= \nu \frac{\partial^2 u}{\partial z^2}, \\
2\Omega (u - U) &= \nu \frac{\partial^2 v}{\partial z^2}.
\end{align*}
\tag{2.25}
\]

The system of equations above can be solved analytically considering a no-slip condition on the disk (\( u = v = 0 \) at \( z = 0 \)), and that the free stream velocity is \( U \) (\( u = U, \ v = 0, \ z \gg \lambda \), where \( \lambda \) is the boundary layer length scale). Seeking solutions in exponential form, the following expressions are obtained:

\[
\begin{align*}
u = U \left( 1 - e^{-z/\lambda} \cos \frac{z}{\lambda} \right), \\
v = U e^{-z/\lambda} \sin \frac{z}{\lambda}.
\end{align*}
\tag{2.26}
\]

The quantity \( \lambda = \sqrt{\nu/\Omega} \) is referred to as the Ekman depth. It is also relevant to define a boundary layer thickness as the distance \( \delta_E \) from the wall at which the transverse velocity \( v \) crosses the origin for the first time, i.e., at \( z = \pi \lambda \), thus giving \( \delta_E = \pi \lambda = \pi \sqrt{\nu/\Omega} \).

Since the magnitude of \( u \) and \( v \) change at different rates with \( z \), the direction of the velocity vector also changes with \( z \), giving rise to the so-called Ekman spiral.
CHAPTER 2

Figure 2.4: (a) Velocity profiles for an Ekman layer with $U = 1$, showing the Ekman depth at $z = \lambda$ and the thickness $\delta_E$ at $z = \pi \lambda$; (b) Ekman spiral.

Figure 2.4(a) illustrates typical profiles of $u$ and $v$, and Fig. 2.4(b) shows the shape of the Ekman spiral in the $u-v$ plane, where each point of the line forming the spiral is associated with a different value of $z$. For further details about Ekman layers, including extended formulations and comparisons with measurements of geophysical flows, please refer to Cushman-Roisin & Beckers [3].

Boussinesq approximation for rotating flows

In flows affected by buoyancy, it is customary to assume that the fluid is incompressible and that the density is variable in the buoyancy-generating terms only. This approach is simpler than considering the full system of governing equations in compressible form, and also more efficient, particularly for flows at low Mach number. The approximation, originally proposed by Boussinesq [10], is very frequently used in the numerical simulation of natural convection under gravity. The underlying formulation of the approximation for gravitational buoyancy can also be applied to flows dominated by centrifugal buoyancy, as it is shown below.

To present the formulation in a more general way, assume that gravity can also enter the equations, so that Eq. 2.18 becomes (note that the equation has been multiplied by the density $\rho$):

$$\rho \left( \frac{\partial u}{\partial t} + u \cdot \nabla u \right) = -\nabla p + \mu \nabla^2 u - 2\rho \Omega \times u - \rho \Omega \times (\Omega \times r) + \rho g. \quad (2.27)$$

In the Boussinesq approximation framework, it is assumed that the density $\rho$ is constant in the momentum equation, except in the terms that generate buoyancy, which, for the equation above, are the centrifugal and gravitational forces. It is also assumed that the density variation is small relative to a reference density, i.e., $\rho' \ll \rho_0$, and that it is a function of the temperature only, $\rho = \rho_0 + \rho' = \rho(T)$. To identify which terms generate buoyancy, it is necessary to check whether a given term can be written as the gradient of a scalar; as shown earlier, this happens for
the centrifugal force, and the gravitational force, which is constant, also has that property. Since density variations are assumed to be small, if a variable density were considered, for example, in the non-linear term, a term of the form $\rho'u \cdot \nabla u$ would appear, but it would compete with the term $\rho_0u \cdot \nabla u$, which is much larger. Thus, the density can be assumed to be constant in this term. The centrifugal and gravitational forces, on the other hand, do not play a dynamical role and are absorbed into the pressure gradient, thus the respective terms multiplied by $\rho'$ do not have a much larger counterpart to compete with. With these considerations, Eq. 2.27 becomes,

$$
\rho_0 \left( \frac{\partial u}{\partial t} + u \cdot \nabla u \right) = -\nabla p^\dagger + \mu_0 \nabla^2 u - 2\rho_0 \Omega \times u - \rho' \Omega \times (\Omega \times r) + \rho'g, \quad (2.28)
$$

where the parts of the centrifugal and gravitational forces multiplied by $\rho_0$ are absorbed into the reduced pressure $p^\dagger$ (note that in this case the definition of $p^\dagger$ differs slightly from Eq. 2.19). Finally, for small density variations with temperature, the density is $\rho = \rho_0 \left[ 1 - \alpha (T - T_0) \right]$, or $\rho' = -\rho_0 \alpha (T - T_0)$, with $T_0$ being a reference temperature and $\alpha$ the thermal expansion coefficient. Obviously, it is also necessary to specify a transport equation for the temperature, which is simply an advection-diffusion equation obtained from the energy conservation equations. Thus, the full system of equations is (dividing by the reference density $\rho_0$),

$$
\nabla \cdot u = 0, \quad (2.29a)
$$

$$
\frac{\partial u}{\partial t} + u \cdot \nabla u = -\nabla p^\dagger + \nu \nabla^2 u - 2\Omega \times u + \alpha (T - T_0) \Omega \times (\Omega \times r) - \alpha (T - T_0) g, \quad (2.29b)
$$

$$
\frac{\partial T}{\partial t} + u \cdot \nabla T = \kappa \nabla^2 T, \quad (2.29c)
$$

where $\kappa$ is the thermal diffusivity. The equations above provide a complete formulation to investigate problems involving buoyancy generated by gravity, centrifugal force, or both. The studies presented in Chapters 5, 6 and 7 are performed using this set of equations.

It is noteworthy that an extended version of the Boussinesq approximation has been proposed relatively recently by Lopez et al. [11], whose idea is to account for all sources of buoyancy with a formulation in the inertial reference frame, applying a variable density to the non-linear term. However, such an approximation introduces an inconsistency, which can be observed when the formulation is transferred to a rotating frame, as it is shown in Appendix A. For this reason, in this work the Boussinesq approximation is considered in its ‘traditional’ form.

### 2.2 Review of previous work

In this section a review of relevant studies is presented, split into three subsections. First, the problem of flow in a rotor/stator cavity is reviewed, considering some experimental studies and focusing on results obtained numerically. For buoyancy-induced flow in a sealed rotating annulus, reviews are presented for studies with focus on industrial applications and for those motivated by flows of geophysical
interest. In the last subsection the problem of flow in a heated rotating cavity with an axial throughflow is considered.

2.2.1 Flow in rotor/stator cavities

Rotor/stator configurations are often investigated considering an annular cylindrical system bounded by two plane, parallel disks. The inner cylindrical surface, or shaft, and one of the disks rotate at a rate $\Omega$, while the outer cylindrical surface, or shroud, and the remaining disk remain stationary. In some numerical and experimental studies a cylindrical cavity without a shaft is considered, in which case one of the circular disks rotates and the other disk and the shroud remain stationary. Referring to Fig. 1.3(a), the flow is governed by three main parameters: the aspect ratio $A = d/(b - a)$, the curvature parameter $R = (b + a)/(b - a)$, and the Reynolds number based on the velocity $\Omega b$ and on the outer radius $b$,

$$Re_\Omega = \frac{\Omega b^2}{\nu}.$$  \hfill (2.30)

Note that for the case without a shaft $a = 0$, so that $A = d/b$ and $R = 1$. Depending on the values of $A$ and $Re_\Omega$, different flow regimes might exist, as detailed below.

Daily & Nece [12] performed a theoretical and experimental investigation using a cylindrical cavity with $A$ ranging from 0.0127 to 0.217 and $Re_\Omega$ from $10^3$ to $10^7$. Depending on the combination of parameters, it was concluded that four flow regimes may occur as follows (where the boundary layers refer to those formed close to the disks):

i) laminar flow with merged boundary layers;

ii) laminar flow with distinct boundary layers;

iii) turbulent flow with merged boundary layers;

iv) turbulent flow with distinct boundary layers.

The most interesting of these regimes is regime (iv), although it should be noted that in many practical situations the rotor boundary layer is not fully turbulent, but it is clearly separated from the stator boundary layer. Regime (i) occurs for low Reynolds numbers only, on the order of $10^4$ for the cases studied in [12]. The experimental measurements revealed that regime (iii) occurred only for the lowest values of $A$, in which case the axial gap is so small that the boundary layers do not separate from each other. Therefore, for moderate values of $A$ and relatively high Reynolds numbers ($Re_\Omega > 10^6$), regime (iv) dominated the dynamics. In what follows, all flows are assumed to be in regime (iv), unless stated otherwise.

Itoh et al. [13] investigated experimentally a configuration similar to that of [12], using $A = 0.08$ and $Re_\Omega$ up to $10^6$. Velocity and turbulent stresses were measured at different radial stations, so that the behaviour of the rotor and stator boundary layers could be compared across different radii. For all values of $Re_\Omega$ investigated, a general flow pattern was identified, in which fluid is pumped outwards near the rotating disk, forming an Ekman-type boundary layer, and transported inwards near the stator surface, in a boundary layer referred to as a Bödewadt layer. These two
boundary layers are separated by a core region in which the radial velocity is zero and the tangential velocity is constant. Interestingly, the measurements revealed that the rotor boundary layer remained laminar in some cases, whereas the stator boundary layer was turbulent in all cases. The authors suggested that this was due to the fact that the flow near the rotor was accelerating, whereas that near the stator was decelerating, and thus was more unstable. The authors identified a critical local Reynolds number, based on the local radius $r$ instead of the disk radius $b$, for transition to turbulence near the rotor surface. The turbulent stress measurements revealed that the stresses were greater near the stator than near the rotor, and that they were significantly smaller in the cavity core. Just as in the flow near a flat plate, a viscous sub-layer and a log region were identified near both the rotor and the stator surfaces.

According to the analytical solution for a laminar Ekman layer presented in this chapter (Eq. 2.26), on the rotor surface the direction of the velocity vector is inclined by 45° relative to the tangential direction, i.e., the radial and tangential velocities have the same magnitude. As noted by Cushman-Rosin & Beckers [3], for turbulent flows this angle is smaller. In the experiments of Itoh et al. [13], plots of the tangential versus the radial velocity revealed that, for cases in which the rotor boundary layer was laminar, the profiles obtained were similar to those shown in Fig. 2.4(b), whereas for a turbulent boundary layer the values of the radial velocity were much smaller, thus reducing the angle between the velocity vector and the tangential direction.

While Itoh et al. [13] concluded that transition to turbulence on the rotor side depended on the local Reynolds number only, Cheah et al. [14] argued that the transition also depends on the Reynolds number based on the outer radius. Also, these authors suggested that the reason why the rotor boundary layer becomes turbulent at a higher local Reynolds number than the stator is due to upstream history, rather than to effects of acceleration and deceleration, as suggested by Itoh et al. [13]. Cheah et al. [14] hypothesised that, as the fluid impinges on the shroud, it is destabilised whilst being transported towards the stator, thus becoming turbulent. Although these experimental studies give different physical interpretations of the phenomena observed, they provide a clear picture of turbulent flows in rotor/stator cavities.

Early numerical investigations of rotor/stator flows were restricted to two-dimensional axisymmetric solutions using turbulence models. One obvious challenge rotor/stator flows pose to turbulence modelling is the possibility of coexistence of laminar, transitional and turbulent flow within the cavity, which requires complex modelling. Standard models such as the $k - \epsilon$ do not consider rotation effects on the turbulence, and fail to predict transition. Experimental data were reproduced with limited success using such models, e.g. in Chew & Vaughan [15], Morse [16] and Elena & Schiestel [17], where it was shown that using simple turbulence models it is possible to obtain satisfactory agreement with experimental data, although in some cases the boundary layer and core regions were not well predicted. Also, these studies indicated that the use of low-Reynolds number modelling approaches requires a case by case calibration of the model constants. Further studies using Reynolds stress models (RSM) obtained more success in predicting radial and tangential velocity profiles (Elena & Schiestel [17], Randriamampianina et al. [18], Poncet et al. [19]), as this family of models is able to account for rotation effects and anisotropy.
Still, the quality of the predictions improved mostly on the stator boundary layer, which was fully turbulent, using a RSM model. The rotor boundary layer, where regions of laminar flow may occur, proved to be challenging even for second-order closure models. Predictions of turbulent stresses were also not satisfactory using RSM models, although they were able to provide qualitatively correct profiles, in contrast with the $k-\epsilon$ model [17]. Still, it should be noted that results obtained from simulations using turbulence models were able to give sufficiently accurate predictions for engineering purposes, thus contributing to improve the understanding of industrial rotor/stator systems.

One feature of the boundary layers developed near both the rotor and the stator is that they are three-dimensional, in the sense that the direction of the velocity vector changes with the distance from the wall – note that this is observed even for a laminar Ekman layer. As noted by Lygren & Andersson [20], in three-dimensional boundary layers the shear stress vector is generally not aligned with the velocity gradient vector, which makes the isotropy hypothesis considered in many turbulence models unrealistic. Another characteristic of these boundary layers is that the ratio between the shear stress and the kinetic energy is significantly smaller than in two-dimensional boundary layers. Using DNS, Lygren & Andersson [20] investigated the flow between infinite rotating and stationary disks and compared their predictions with the measurements of Itoh [21], obtaining good agreement for the mean flow quantities, but rather overestimated values for the turbulent fluctuations. The authors attributed this discrepancy to the fact that the geometry in the experiments was finite. Analysing the characteristics of the turbulent boundary layers, Lygren & Andersson [20] concluded that the misalignment between the shear stress and the velocity gradient was modest, thus making the hypothesis of turbulent stress anisotropy plausible. However, in a later numerical study using LES, Séverac et al. [22] have shown that, when a finite geometry is considered, the angle between the shear stress and the velocity gradient is much larger than that obtained for infinite disks in [20], particularly near the shroud, thus providing further evidence of the presence of strong anisotropy effects in confined rotor/stator flows.

With advances in computing power, there was a shift in the scientific research community from the use of two-dimensional, RANS-based calculations, towards more accurate simulations using DNS and LES. Serre & Pulicani [23] and Raspo et al. [24] described the implementation of spectral methods in cylindrical coordinates for the simulation of both open and enclosed rotor/stator flows. The use of such methods has shown great potential due to their accuracy and convergence properties. Focusing on applying such methods to rotor/stator flows, Serre et al. [25] investigated in detail the onset of instabilities in an enclosed rotor/stator cavity using DNS at low Reynolds numbers ($Re_\Omega \sim 10^6$), showing the existence of annular and spiral patterns, as shown in Fig. 2.5. The existence of such patterns was confirmed experimentally by Czarny et al. [26] and Cros et al. [27], among others, suggesting that the use of three-dimensional unsteady CFD is imperative in order to reproduce the full range of features encountered in rotor/stator flows at these conditions. Again using DNS, Serre et al. [28] investigated flow in a rotor/stator system for $Re_\Omega$ up to $3 \times 10^5$, for which the stator boundary layer is turbulent and the rotor boundary layer has regions of laminar, transitional and turbulent flow. The authors observed spiral patterns near the stator surface, similar to those reported by Serre et al. [25]. The DNS results of [28] also revealed that the shroud
plays a crucial role in the transition to turbulence, as counter-rotating vortices are created when the flow transported along the shroud from the rotor impinges on the stator; these vortices are then propagated inwards along the stator. Poncet & Randriamampianina [29] presented experimental and DNS results for flow in a rotor/stator cavity with $Re_\Omega = 9.5 \times 10^4$, so that only the stator boundary layer was turbulent. The authors obtained similar conclusions to previous studies in terms of the behaviour of the three-dimensional boundary layers, and have shown that the DNS was able to predict the mean velocity profiles as well as the turbulent stresses obtained experimentally with good accuracy. Other numerical and experimental studies investigating flow structures in transitional rotor/stator flows include Poncet et al. [30], Serre [31] and Makino et al. [32].

Figure 2.5: Iso-surfaces of axial velocity fluctuation showing annular and spiral patterns near the stator and the rotor, respectively. Figure extracted from Serre et al. [25].

As a compromise between the full resolving capability of DNS and the use of RANS-based turbulence models, LES has emerged as an adequate level of modelling for rotor/stator flows. In a pioneering study, Lygren & Andersson [33] used LES to study the configuration considered by the same authors with DNS in [20]. Obviously, the computational cost of the LES was significantly lower than that for the DNS. Excellent predictions were obtained for the mean velocity profiles and for the nature of near-wall structures, although the results obtained for the turbulent stresses were less encouraging. Later, Séverac et al. [22] performed an experimental and numerical investigation using LES considering a finite cavity, in contrast with [20, 33]. This, along with the study of Pasquetti et al. [34], was the first study in which LES results were reported for turbulent flow in a rotor/stator cavity considering the presence of a shroud and an inner shaft. Very good agreement was obtained between the LES results and the experimental measurements, although anisotropy effects introduced by the relatively coarse LES grid were observed, causing a level of stress anisotropy much higher than that observed in the experiments. Séverac et al. [22] were able to obtain solutions for $Re_\Omega$ up to $10^6$, and observed that in the turbulent regime the near-disk structures are quasi-axisymmetric, in contrast with the spiral structures encountered at lower $Re_\Omega$. This supports the well-known fact that anisotropy effects become less pronounced as the Reynolds number is increased. LES was used in a number of other numerical investigations of flow in rotor/stator cavities, e.g. [35, 31, 36, 37, 32]. The review paper by Launder et al. [38] presents a comprehensive
overview of experimental and numerical studies of instability mechanisms, transition and fully turbulent flows in sealed rotor/stator cavities.

Besides the investigations described above for isothermal rotor/stator flows in confined domains, studies including buoyancy effects have been performed, e.g. Serre et al. [39], Poncet & Serre [40] and Tuliszka-Sznitko et al. [41]. In general, these studies concluded that buoyancy plays a secondary role in rotor/stator flows, as it was observed that it did not modify the velocity profiles in comparison with isothermal cases. Additionally, other investigators considered the presence of an impinging jet on the rotor – see Harmand et al. [42] for a review.

Conclusions

The review presented above shows that different levels of CFD modelling have been applied successfully in studies involving rotor/stator flows. While RANS-based models are able to give sufficiently accurate results for engineering purposes, with advances in computing power LES and DNS methods have been used to investigate fundamental aspects of these flows, such as the transition from laminar to turbulent flow that occurs in the rotor disk boundary layer. As a preliminary study to assess the capability of the spectral element formulation presented in Chapter 3 to simulate rotor/stator flows, results from an investigation using both DNS and LES are presented in Chapter 4. The rotational Reynolds numbers considered are \( \text{Re}_\Omega = 4 \times 10^5 \) and \( \text{Re}_\Omega = 10^6 \), so that in one case the rotor boundary layer is only partially turbulent whilst for the higher \( \text{Re}_\Omega \) it is fully turbulent.

2.2.2 Buoyancy-induced flow in sealed rotating cavities

In this section a review of studies of flow induced by centrifugal buoyancy in a sealed rotating annulus bounded by two side-wall disks is considered (see Figs. 1.3(a) and 2.6(a)). The main interest is on the case where the direction of the heat flux is parallel and opposed to the centrifugal force, i.e. the outer cylindrical surface is heated whilst the inner surface is cooled. This situation contrasts with cases in which the temperature gradient is orthogonal to the centrifugal force. The case when one disk is heated and the other is cooled, so that the heat flux of the base flow is parallel to the axial direction, is analogous to the problem of a differentially heated rectangular cavity with gravitational buoyancy. Although the orthogonal temperature gradient problem is not covered in this review, the reader may refer to some of the early theoretical (Homsy & Hudson [43]), experimental (Hudson et al. [44]) and numerical (Pustovalov & Sparrow [45], Chew [46]) studies performed for flows induced by centrifugal buoyancy in a cylinder without an inner shaft and with an axial temperature gradient. For a numerical study including an inner cylindrical surface, see Bohn et al. [47].

The review of previous work is presented in two parts. First, studies motivated by applications in turbomachinery are considered, and subsequently a review of more fundamental studies, mostly motivated by applications in geophysics, is given. While studies involving turbomachinery applications are generally focused on high-Reynolds number flows and on analysing heat transfer rates from the disks and cylindrical walls, the fundamental studies considered in the second part of the review tend to employ lower Reynolds numbers and to analyse the onset of convection, the presence of large-scale structures and the transition between different flow regimes.
Studies motivated by flows in turbomachinery internal air systems

Bohn & Gorzelitz [48] performed an experimental investigation of air flow in a sealed rotating annulus with insulated disks, cooled inner shaft and heated shroud. The geometry used is shown in Fig. 1.3(a). For a cavity rotating with angular velocity $\Omega$ around its axis, one can define a rotational Reynolds number $Re_\theta$ and a centrifugal Rayleigh number $Ra$ as,

$$Re_\theta = \frac{\Omega r_m (b - a)}{\nu},$$

$$Ra = \frac{\Omega^2 r_m \alpha \Delta T (b - a)^3}{\nu \kappa},$$

where it should be noted that the definition of $Re_\theta$ is different from that used for $Re_\Omega$ in Eq. 2.30. In the definitions above, $r_m = (b + a)/2$ is the mean radius, $(b - a)$ is the radial gap used as a characteristic length scale, $\alpha$ is the thermal expansion coefficient, $\Delta T$ is the temperature difference between the hot and cold surfaces and $\kappa$ is the thermal diffusivity. Another parameter of great importance in the analysis of buoyancy-induced flows is the Prandtl number $Pr$, defined as the ratio of the kinematic viscosity and the thermal diffusivity,

$$Pr = \frac{\nu}{\kappa}.$$  

A Grashof number $Gr$ is also commonly employed, and is related to $Ra$ and $Pr$ by $Ra = GrPr$. Bohn & Gorzelitz [18] considered Reynolds and Rayleigh numbers in the ranges $10^4 \leq Re_\theta \leq 10^7$ and $10^7 \leq Ra \leq 10^{12}$. The authors measured the heat transfer rate on the inner cylindrical wall for different cavity pressures, which did not affect the results obtained. A correlation for the Nusselt number, defined as the ratio between convective $\dot{q}$ and conductive $\dot{q}_c$ heat transfer,

$$Nu = \frac{\dot{q}}{\dot{q}_c},$$

was obtained, $Nu = 0.246Ra^{0.228}$. The exponent of $Ra$ in this correlation is rather low when compared to natural convection under gravity at high $Ra$, for which typically $Nu \propto Ra^{1/3}$ (e.g. Lloyd & Moran [19]). Bohn & Gorzelitz [18] attributed this reduction to the Coriolis force, which tends to suppress the fluid motion relative to the walls.

In a subsequent study, Bohn et al. [50] extended their previous work [18] and considered three different geometrical configurations, named A, B and C. In all cases the axial distance between the disks and the radius of the inner cylindrical surface were kept constant. The outer radius of geometry A was almost 50% larger than for geometries B and C, and for geometry C eight separation walls were inserted radially, thus forming eight sectors of $45^\circ$ each, whereas for cavities A and B no separation walls were inserted. Here again the Rayleigh number was varied in the range $10^7 \leq Ra \leq 10^{12}$, although a narrower range was considered for cavity C. The results were analysed in terms of correlations of the Nusselt number versus the Rayleigh number, and no significant differences were observed between cases A and B, thus indicating that the length of the radial gap does not have a strong influence on the heat transfer. More significant differences were observed between the results for cavities B and C, as the values of $Nu$ in case C were consistently higher than for.
case B, although with the same $Ra$ scaling. With the inclusion of radial walls (case C), effects arising from the Coriolis force are suppressed, thus enhancing the heat transfer rates. The authors also performed numerical simulations in both steady and unsteady form assuming that the flow was laminar for cavity C. For the steady calculations, the values of $Nu$ were higher than those obtained experimentally, but the $Nu - Ra$ scaling was consistent with the experimental correlation. For unsteady flow, a single value of the Rayleigh number was considered, $Ra = 4.86 \times 10^7$, and due to convergence problems, no reliable solution was obtained, although it was noted that the values of $Nu$ varied within a large range over time, and were significantly higher than the experimental value. Bohn et al. \cite{50, 51} also investigated numerically the case of axial heat flux allowing unsteady flow, and observed that after a short transient the calculation converged to a steady-state solution. This indicates that the situation with radial heat flux is considerably more challenging to study numerically, due to the inherent unsteady and unstable nature of the flow. Using a low-Reynolds number $k - \epsilon$ turbulence model and assuming steady flow, Bohn & Gier \cite{52, 53} concluded that the Nusselt number was around 10% higher than in the laminar case. However, it is hard to assess whether this difference is related to the flow physics or to an artefact of the turbulence model.

Sun et al. \cite{54} performed three-dimensional calculations using a compressible flow solver without any turbulence modelling on a geometrical configuration equivalent to cavity B in the experiments of Bohn et al. \cite{50}. Three Rayleigh numbers within the range $10^8 \lesssim Ra \lesssim 10^{10}$ were considered. Analysis of instantaneous snapshots in the radial-tangential plane revealed the presence of large-scale structures responsible for transporting hot (cold) fluid towards the inner (outer) part of the cavity, and the values of $Nu$ obtained were in excellent agreement with Bohn et al.’s \cite{50} correlation. It was shown that when a 45° sector was used instead of the full annulus, no difference in $Nu$ was observed, although the flow structure was affected, since the use of periodicity prevents the formation of large-scale structures.

Subsequently, King et al. \cite{55} performed two- and three-dimensional calculations using an incompressible formulation, in which buoyancy was accounted for using the Boussinesq approximation. These authors considered Rayleigh numbers up to $10^9$ and $10^6$ for the 2D and 3D calculations, respectively. In agreement with the results of Sun et al. \cite{54}, King et al. \cite{55} also observed large-scale structures in the $r - \theta$ plane, for both the two- and three-dimensional cases. However, the Nusselt numbers calculated were significantly higher than those obtained experimentally by Bohn et al. \cite{50}, but were in good agreement with correlations for natural convection under gravity obtained by Hollands et al. \cite{56} and Niemela et al. \cite{57}, in which $Nu \propto Ra^{1/3}$. One might wonder whether this difference comes from the fact that King et al. \cite{55} used an incompressible formulation, whereas Sun et al. \cite{54} solved the equations in compressible form; since the temperature differences considered by the authors were rather small, and since effects of viscous dissipation and density variation due to pressure changes can be neglected \cite{50}, this is unlikely to be the case. King et al. \cite{55} hypothesised that Sun et al. \cite{54} obtained better agreement with the experimental data because they considered a three-dimensional model (including the disks), which could be responsible for attenuating the heat transfer.

King et al. \cite{55} observed that the number of pairs of convection cells formed along the azimuthal direction changed over time and with the Rayleigh number. Owen \cite{58} then suggested that the system organises itself in a manner so as to
maximise entropy production, and proposed an analytical expression for the number of convection cells as a function of the inner and outer radii $a$ and $b$, assuming that the vortices formed were circular.

Recently, Tang & Owen \[59\] proposed a theoretical model for heat transfer in a sealed rotating annulus with a radial temperature gradient and compared their results with the correlations of Bohn et al. \[50\], obtaining overall good agreement. In the model laminar flow is assumed and a correlation for natural convection at low $Ra$, for which $Nu \propto Ra^{1/4}$, is used, together with considerations of compressibility in the core.

**Fundamental studies motivated by geophysics applications**

The first experimental investigations employing a differentially heated rotating annulus (Fig. 2.6) to study geophysical phenomena date from the 1950s. As noted by Hide \[60\], theoretical evidence suggested that the earth’s magnetic field was created by electric currents generated from fluid movement in the core, which is, in turn, strongly influenced by gravitational, rotational and thermal convection effects. To investigate these primary effects, the rotating annulus appeared as a good model due to its simplicity and because it contains the basic aforementioned ingredients, which can be varied individually in an experiment. Later, it was recognised that the same physical setting could be used to study the general circulation of the atmosphere, where the cold and hot surfaces model the poles and the equator, respectively, whilst the system rotation mimics that of the earth.

In the pioneering experiments performed by Hide \[60\] and Fowlis & Hide \[61\] (see also Hide & Mason \[62\] for a review of early studies), a cylindrical annulus filled with water was mounted on a turntable and both the rotation rate $\Omega$ and the temperature difference $\Delta T$ between the cylindrical surfaces could be varied. By visual observation of the flow, it was revealed that a number of regimes occurred when certain control parameters were varied. Referring to Fig. 2.6(a), a thermal Rossby number $\Theta$ and a Taylor number are generally defined as,

$$\Theta = \frac{g\alpha \Delta T d}{\Omega^2 (b-a)^2},$$  \hspace{1cm} (2.35)  

$$Ta = \frac{4\Omega^2 (b-a)^5}{\nu^2 d},$$ \hspace{1cm} (2.36)

where $g$ is the acceleration of gravity in the negative $z$ direction shown in Fig. 2.6(a). The thermal Rossby number is a measure of the ratio between inertial and Coriolis forces, whereas the Taylor number relates Coriolis and viscous effects. At this point it is important to note that the system shown in Fig. 2.6(a) is nearly identical to that introduced in Fig. 1.3(a), the main differences being that the rotating annulus used in experiments of geophysical interest is mounted such that the axis is in the vertical direction and that the centrifugal force is not necessarily large when compared to gravity, as explained below. In experiments motivated by turbomachinery applications, on the other hand, the system is mounted horizontally and the centrifugal force can be orders of magnitude larger than $g$. Additionally, in the vertical annulus the Rayleigh number is defined in terms of $g$ rather than $\Omega^2 r_m$,

$$Ra_g = \frac{g\alpha \Delta T (b-a)^3}{\nu \kappa}.$$  \hspace{1cm} (2.37)
Figure 2.6: (a) Geometry of a rotating annulus commonly considered in experiments of baroclinic and centrifugally-driven flows. The action of gravity in the negative $z$ direction can be neglected if the centrifugal acceleration is large enough. (b) $Ta - \Theta$ regime diagram in logarithmic scale obtained by Fowlis & Hide \cite{61}.

Figure 2.6(b) illustrates schematically the four flow regimes identified by Fowlis & Hide \cite{61}. Two of these regimes are characterised by axisymmetric flow, which reflects the symmetry of the physical model. One of them occurs at very low rotation rates (small $Ta$ and large $\Theta$) when the Coriolis force is not strong enough to destabilise the system in the tangential direction. The second axisymmetric regime is characterised by temperature differences that are so small that the symmetry is not broken. Of more relevance are the regimes of steady waves and irregular flow, as well as the vacillation regime which occurs during the transition between them. The appearance of waves along the azimuthal direction takes place via a baroclinic instability, in which the Coriolis force suppresses meridional motion and induces an azimuthal pressure gradient. Hide \cite{60} observed that the number of cells formed in the tangential direction varied as the rotation rate was increased. The steady waves regime is characterised by great regularity of the waves, and the experiments revealed that the pattern formed drifted along the azimuthal direction, the sense of the drift (slower or faster than the velocity of the solid walls) depending on whether the imposed temperature gradient was directed radially inwards or outwards. As the rotation rate was further increased in the experiments, the amplitude and shape of the waves started to vary over time, thus corresponding to a so-called vacillation regime. Increasing the rotation rate even more, irregular flow patterns were observed; such a regime is commonly referred to as geostrophic turbulence.

The literature on baroclinic flows is vast, and one may refer to, for example, Castrejón-Pita & Read \cite{63} and Vince et al. \cite{64} for recent experimental investigations, Randriamampianina et al. \cite{65}, Read et al. \cite{66} and Vincze et al. \cite{64} for numerical investigations using DNS, and Lewis & Nagata \cite{67} for a linear stability analysis using a model based on the early experiments of Hide \cite{60}. In general, the numerical studies were able to successfully reproduce the dynamics observed in the laboratory.
and have played a key role towards gaining better understanding of the vacillation regime, which is critical to understand the transition between the straightforward steady waves regime and geostrophic turbulence. Besides the thermal Rossby and Taylor numbers mentioned above, it has been shown that the Prandtl number is also of great importance in the characterisation of baroclinic flows (see, e.g. Lewis et al. [68]). While experiments are more easily realisable using fluids such as air ($Pr = 0.7$) and water ($Pr = 7$), geophysicists are also interested in fluids with much lower $Pr$ to understand the dynamics in the earth’s core, which is mostly formed of liquid metals. This reinforces the relevance of using direct numerical simulation to study these flows, since a range of Prandtl numbers can be investigated. For a thorough review of the phenomena observed in baroclinic flows and their relevance to study geophysical phenomena, please refer to Read et al. [69], and for a review of the amplitude vacillation regime, see Früh [70].

Although all the interesting dynamics described above occur as a result of the rotation of the system, and more specifically due to the action of the Coriolis force, in most experimental and numerical studies involving baroclinic flows the centrifugal acceleration $\Omega^2 r$ is small compared to the acceleration of gravity $g$. It is common to define a Froude number $Fr$ to relate the importance of the centrifugal force in comparison with gravity,

$$Fr = \frac{\Omega^2 r}{g},$$

(2.38)

where $r$ represents a local radial distance from the axis of rotation. The baroclinic instability can occur even when $Fr \ll 1$, as its dynamics is governed by the fact that the temperature gradient imposed is normal to the body force that causes buoyancy effects. When $Fr \sim 1$, the centrifugal and gravitational forces compete, thus changing the basic nature of the instability, since the centrifugal force is parallel to the temperature gradient.

Cases where $Fr \gg 1$ have also drawn the attention of geophysicists, for at least two reasons. First, it can be seen as a limiting case of both the lower symmetrical and irregular regimes shown in Fig. 2.6(b), possibly corresponding to a transition between the two. The irregular regime, or geostrophic turbulence, might thus be obtained without the presence of axial gravity. Second, if the body force in the system is effectively radial, the centrifugal force can be used to mimic the effects of radial gravity, a condition which is not feasible to obtain in laboratory experiments and is relevant to model convection in the liquid core of planets and in stars (Busse [71]). In these situations, the axis of rotation and the buoyancy force are not aligned, in contrast with models that use axial gravity to generate buoyancy.

Busse [71] investigated the onset of instabilities in a variety of rotating systems where the centrifugal force is the dominant body force. To simplify the analytical treatment of the equations, it was assumed that the distance between the cylinders, or gap, was much smaller than the mean radius, i.e. $(b - a) \ll r_m$ in the notation of Fig. 2.6(a). This allows for a Cartesian coordinate system to be introduced, but also eliminates curvature effects. The analysis demonstrated that, when the system was bounded by horizontal, no-slip surfaces, the onset of convection took place in the form of stationary columns aligned with the axis of rotation. Assuming that the end surfaces were inclined, so that the height of the annulus either increased or decreased with the distance from the rotation axis, convection occurred in the form
of drifting columns, also referred to as Rossby waves, associated with the imaginary part of the eigensystem solved in the stability analysis. Additionally, the use of inclined end-walls has a stabilising effect, thus requiring a higher critical Rayleigh number for convection to start in comparison with the case of flat end-walls. This tendency was confirmed experimentally by Busse & Carrigan [72], who performed visual observations of flow in a water-filled rotating annulus with a small gap at high rotation rates, achieving centrifugal accelerations up to seven times \( g \). The experimental observations were in good agreement with the theoretical results of Busse [71], thus indicating that gravitational effects could indeed be neglected when the centrifugal acceleration exceeded that of gravity.

In a subsequent experimental study by the same group, Azouni et al. [73] performed temperature measurements using mercury (low Prandtl number), and obtained quantitative data to complement the previous visual observations. Since the end-walls were conical, a drifting-wave pattern was obtained, which could be detected from the temperature measurements. In a series of papers, Busse and co-workers [74, 75, 76, 77] investigated analytically and numerically the onset and stability of Rossby waves considering conical end-walls with and without curvature for different Prandtl numbers, again assuming that the radial gap between the cylinders was very small. These investigations exploited the property that at high rotation rates the motion tends to become two-dimensional, as explained by Lin et al. [78], which simplifies the analytical treatment of the problem. In analogy with observations of the baroclinic instability reported by Hide [60], a vacillation regime was observed by Busse and co-workers [75, 76, 77] when the Rayleigh number was increased, characterised by irregularity of the solutions over time.

Relaxing the hypothesis of a small radial gap, Pino et al. [79] investigated numerically the stability of Rossby waves considering different radius ratios \( a/b \) and assuming that gravity was directed radially inwards and that \( T_a > T_b \). Conical end-walls were assumed, and the analysis was two-dimensional using the procedure described by Lin et al. [78]. In addition to the columnar rolls of convection obtained when the gap was small, Pino et al. [79] have shown the existence of spiral rolls, which are the preferred mode of convection depending on the ratio \( a/b \) and on the rotation rate. Employing a non-linear analysis in a system with a finite gap and inclined end-walls, Chen & Zhang [80] observed the existence of vacillating and chaotic flow for Rayleigh numbers above the onset of convection. Due to the competition between the stabilising effect of rotation and strong non-linearity at high \( Ra \), the authors observed an increase on the size of convection cells as \( Ra \) was increased, and thus a decrease of the dominant azimuthal wavenumber. The transition to vacillating flow using a finite gap was also investigated by Net et al. [81] using two-dimensional simulations, i.e. assuming that the annulus was infinitely long in the axial direction. Those authors suggested a transition scenario formed of a sequence of bifurcations containing steady, periodic, quasi-periodic and chaotic solutions.

While the studies cited above considered conical end-walls with or without curvature, as this configuration gives a fair representation of spherical shells, less work was devoted to the case of flat, parallel end-walls. In the early work of Busse [71], in which the small-gap approximation was used, it was shown that when the end-walls were flat the convection cells were stationary. Alonso et al. [82] considered the case of a finite gap and flat end-walls with radial gravity, but assuming stress-free condi-
tions, using linear stability analysis, and have shown that convection first occurs in the form of stationary columns for sufficiently high Taylor numbers, in agreement with the results of Busse [71]. However, the subsequent stability analysis of Alonso et al. [83] including no-slip end-walls and a finite gap revealed that the convection cells were not stationary, but drifted in the rotating frame, just like the Rossby waves observed in the rotating annulus with conical end-walls. This happens because the presence of a no-slip wall induces viscous effects which break the constraint of the Taylor-Proudman theorem. Therefore, near the solid end-walls the Coriolis force is not balanced by the azimuthal pressure gradient, thus causing the drift of the convection columns.

Conclusions

From the review presented for studies motivated by buoyancy-induced flows occurring in gas turbines internal air systems, it is clear that the investigations performed so far were focused on extracting global features of the flow. Thus there is a lack of analyses focusing on the unsteady behaviour of the flow, as well as on time-averaged quantities. This motivates the present work, in which the flow structure is analysed in greater detail, investigating, for example, the behaviour of the Ekman-type boundary layers formed near the rotating disks, the shroud boundary layer, components of the Reynolds stress tensor, and unsteady features of the flow. As the level of understanding of the flow physics required to achieve improvements in design increases, the use of high-fidelity methods to obtain detailed insight into the flow behaviour becomes increasingly relevant.

Studies motivated by applications in geophysics focus on more fundamental aspects of the flow, such as the transition between steady and irregular flow, but are restricted to relatively low Rayleigh numbers. In the present research aspects of the onset of convection and transition to turbulence are analysed, and the irregular regime, or geostrophic turbulence, is investigated in more detail. While situations where the baroclinic instability dominates the dynamics are also of interest and could be studied using the numerical tool employed in this research, they are out of the scope of this thesis, although results for two test cases are presented for numerical verification purposes.

2.2.3 Buoyancy-induced flow in rotating cavities with an axial throughflow

The geometry of a rotating cavity with an axial throughflow is shown in Fig. 1.3(b). Besides the rotational Reynolds number $Re_\theta$ and the centrifugal Rayleigh number $Ra$, when an axial throughflow is present two new non-dimensional parameters are introduced: an axial Reynolds number, $Re_z$, defined in terms of the axial velocity at the inlet, $W$, and of the hydraulic diameter $2(a - r_s)$, and a Rossby number, $Ro$, which relates the strength of the axial jet relative to the rotational speed at the inner radius. In some studies a central shaft is not considered, in which case $Re_z$ is defined in terms of the inlet pipe diameter $2a$. These parameters are given by,

$$Re_z = \frac{2W(a - r_s)}{\nu}, \quad (2.39)$$
Farthing et al. [84, 85] performed an experimental investigation of flow in a rotating cavity with an axial throughflow of cooling air without a central shaft. For cases without heat transfer, the authors observed that the axial jet induces a toroidal vortex inside the cavity for large Rossby numbers, i.e. when the jet velocity is much larger than a characteristic angular velocity. Reducing the Rossby number, vortex breakdown in both axisymmetric and non-axisymmetric forms was observed. When the vortex breakdown is non-axisymmetric, the axial jet precesses and most of the inflow enters the cavity radially. It was shown in the experiments that the precession amplitude reaches a maximum for $Ro = 21$. For values of $Ro$ lower than 20, the flow in most of the cavity consisted of solid body rotation, which means that vortex breakdown effects are not significant and there is no secondary recirculation inside the cavity. It should be noted that in gas turbine applications the Rossby number is of order unity or less, therefore the isothermal experiments suggested that, in the absence of heat transfer, no vortex breakdown would occur at engine operating conditions. The results reported in [84] are in agreement with the earlier experiments of Owen & Pincombe [86] for an isothermal rotating cavity, where it was also observed that vortex breakdown occurs only in a limited range of the Rossby number.

Farthing et al. [84, 85] also conducted non-isothermal experiments, in which the disks were heated using different radial temperature distributions. When the disks had a constant or radially increasing temperature profile, the flow was observed to be buoyancy-induced and non-axisymmetric for the entire range of parameters investigated. The flow pattern observed by Farthing et al. [84] is shown schematically in Fig. 2.7 and has become an accepted mechanism to describe flows in rotating cavities with axial throughflow. Counter-rotating vortices are formed and have an angular velocity lower than that of the walls. Such vortices are separated by radial arms, which transport cold fluid outwards, whilst the radially inwards transport occurs in the Ekman layers formed near the disks. Although the Rossby number had a great influence on the flow structure, heat transfer measurements indicated that it had a weak effect on the disk Nusselt number distributions.

![Diagram](image-url)  
**Figure 2.7:** Physical mechanism for flow in a heated rotating cavity with axial throughflow, proposed by Farthing et al. [84].

Long et al. [87] performed experiments in a multi-cavity rig in which the shroud...
was heated and including a central shaft - this is a more realistic configuration in comparison with gas turbine applications. Similarly to what was reported in ref. [84], the authors observed evidence of a structure consisting of counter rotating vortices. For the largest radial clearance, i.e. the distance between the shaft and the inner part of the disk, velocity measurements revealed that in the inner part of the cavity the tangential velocity was larger than that of the disks, which suggests that this region is strongly affected by the axial jet. At outer radii, on the other hand, the flow was close to solid body rotation with an angular speed lower than that of the disks, in agreement with the results of Farthing et al. [84]. The authors concluded that buoyancy effects dominated the dynamics near the shroud. Interestingly, when the radial clearance was decreased keeping the Rossby number approximately constant, buoyancy effects seemed to dominate even at lower radii, since in this case the axial throughflow had a weaker interaction with the flow inside the cavity.

Using the same experimental apparatus of Long et al. [87], Long & Childs [88] investigated the shroud heat transfer behaviour for a multi-cavity rig with a central shaft. The authors observed that the shroud Nusselt number depended weakly on the axial Reynolds number and on the geometrical parameters varied. One interesting conclusion from this work is that the shroud heat transfer is governed by free convection, and that existing correlations for convection under gravity can be adapted to predict the Nusselt numbers on the shroud of a rotating cavity.

In the recent experimental investigation of Puttock-Brown et al. [89] it has also been reported that correlations for natural convection under gravity can be adapted to predict the shroud Nusselt number. As in the work of Long & Childs [88], in [89] a multi-cavity rig with a central shaft was considered, and the measurements suggested that the shroud heat transfer is governed by free convection, in the sense that it is insensitive to the axial Reynolds number. Although the magnitude of the shroud Nusselt number measured is consistent with correlations for turbulent natural convection, the slope of $Nu$ versus the Grashof number matched that of laminar, or low Grashof number, convection.

Table 2.1 shows the range of parameters considered in the experimental studies described above. For reference, in engine operating conditions the following representative values can be considered: $Re_z \approx 10^5 - 10^6$, $Re_\theta \approx 10^7$, $Ro \approx 0.1 - 1.0$, $Gr \approx 10^{13}$.

From other experimental observations [90, 91, 92] it is possible to conclude that the Rossby number is a key parameter that will determine how much influence the axial jet has on the flow structure and disk heat transfer. In general, for small Rossby numbers the flow is strongly affected by buoyancy, the Nusselt number on the downstream disk increases radially, and the relative tangential velocity of the flow is negative. For these cases a large proportion of the throughflow fluid is transported into the cavity due to the rapid rotation of the system. For larger $Ro$, on the other hand, the axial jet dominates and a lower proportion of the axial throughflow is transported towards the outer part of the cavity. Additionally, the Nusselt number on the downstream disk reduces with increasing radius due to jet impingement.

It should be noted that different dynamics can be obtained for the same Rossby number when the axial and rotational Reynolds numbers and the centrifugal Rayleigh number are changed. As shown by Atkins & Kanjirakkad [93], when the Rossby number is kept constant and the tangential and axial Reynolds numbers are varied, the disk temperatures are strongly affected. Still, the measurements show that for
smaller $Ro$ the disk temperatures at large radii are reduced when compared to larger $Ro$ cases.

The flow features of buoyant flows in rotating cavities with an axial throughflow observed experimentally have been reproduced with some success using CFD. Long & Tucker [91] solved the unsteady Navier-Stokes equations in laminar form, and observed the existence of counter-rotating vortices and radial arms, in qualitative agreement with the observations of Farthing et al. [84]. Tian et al. [95, 96] investigated numerically the flow in a rotating cavity with axial throughflow where the shroud is heated and the disks are adiabatic. It was concluded that the flow pattern inside the cavity can be split in two regions: a Rayleigh-Bénard-like zone where natural convection dominates, located near the shroud, and a forced convection-dominated region near the cavity centre, where fluid enters and leaves the cavity.

Sun et al. [97] used RANS and LES models to investigate buoyant flows in rotating cavities with axial throughflow using Ansys Fluent. The authors used sectors of $45^\circ$, $90^\circ$, $120^\circ$ and a full $360^\circ$ model to investigate periodicity in the azimuthal direction, and concluded that only the heat transfer results obtained with the $120^\circ$ sector agreed well with the ones obtained with the full $360^\circ$ model. The Nusselt number predictions were compared with the experimental results presented in [98], and although the values of $Nu$ obtained with LES slightly underestimated the measurements, they followed the trend observed in the experiments. Temperature contours revealed the presence of radial arms inside the cavity, and instantaneous velocity contours confirmed that unsteady large-scale structures dominate the flow pattern. When compared to RANS, the results obtained with LES were more consistent with experimental observations and turbulent flow phenomena. Tan et al. [99] compared the performance of LES and steady and unsteady RANS to predict the flow features and Nusselt numbers reported by Bohn et al. [91]. Whilst with steady RANS the global flow structure could not be captured, similar results in terms of Nusselt number and presence of large-scale structures were observed with both the LES and unsteady RANS approaches. In a later study, Tan et al. [100] applied a discontinuous Galerkin (DG) method using a transitional turbulence model and obtained even better agreement with the experimental data. The authors reported that using the high-order DG method the numerical errors were significantly lower than for a low-order finite volume method. Puttock-Brown et al. [89] used an unsteady RANS model to investigate the flow structure considering a multi-cavity setup, and observed the presence of large scale structures as well as of radial arms, in agreement with previous studies. Additionally, the authors detected coherent structures in the form of counter rotating cells along the shroud, which contribute to form streaks elongated along the azimuthal direction.

For a summary of the operating conditions considered in the numerical studies discussed, see Table 2.2.

In addition to the experimental and computational studies presented above, there have also been recent developments in theoretical models of buoyancy-induced flows in rotating cavities with axial throughflow. Tang et al. [101] used Bayesian statistics to compute the inverse solution of the fin equation, from which it is possible to compute the radial distribution of the Nusselt number on compressor disks from temperature measurements. Owen & Tang [102] developed a theoretical model to predict temperature and heat flux distributions on rotating cavity disks assuming that the Ekman layers are laminar, axisymmetric and steady. In the model it is
fluctuations. This is particularly useful to investigate the interaction between the one can identify regions of the cavity where the flow is strongly affected by unsteady containing second-order quantities, such as rms profiles for temperature and velocities, disk boundary layers are in fact laminar, as suggested by Owen & Tang [102]. By obtaining time- and circumferentially-averaged velocity and temperature profiles, for example, it is possible to verify whether the flow field. The numerical studies available in the literature mostly consist of RANS or relatively coarse LES calculations, and are focused on reporting heat transfer coefficients and temperature profiles for the rotating disks, and on providing insight into the instantaneous behaviour of the flow. The studies reviewed above indicate that there is a substantial amount of knowledge about the physics of buoyant flows in rotating cavities with an axial throughflow of cooling air. While the amount of information about the flow structure available from experiments is limited, unsteady numerical simulations can provide more details about the flow field. The numerical studies available in the literature mostly consist of RANS or relatively coarse LES calculations, and are focused on reporting heat transfer coefficients and temperature profiles for the rotating disks, and on providing insight into the instantaneous behaviour of the flow. The theoretical model is overall able to reproduce the experimental and extrapolated data with good accuracy, which supports the hypothesis that the Ekman layers are laminar, at least in the time-average sense, even though the flow structure within the cavity is strongly unsteady and three-dimensional.

Conclusions

The studies reviewed above indicate that there is a substantial amount of knowledge about the physics of buoyant flows in rotating cavities with an axial throughflow of cooling air. While the amount of information about the flow structure available from experiments is limited, unsteady numerical simulations can provide more details about the flow field. The numerical studies available in the literature mostly consist of RANS or relatively coarse LES calculations, and are focused on reporting heat transfer coefficients and temperature profiles for the rotating disks, and on providing insight into the instantaneous behaviour of the flow.

In this work the objective is to obtain more accurate quantitative data from numerical simulations using LES. By obtaining time- and circumferentially-averaged velocity and temperature profiles, for example, it is possible to verify whether the disk boundary layers are in fact laminar, as suggested by Owen & Tang [102]. By obtaining second-order quantities, such as rms profiles for temperature and velocities, one can identify regions of the cavity where the flow is strongly affected by unsteady fluctuations. This is particularly useful to investigate the interaction between the

### Table 2.1: Range of parameters for the most relevant experimental data reported in the literature.

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>$Re_x$</td>
<td>$2.0 \times 10^9$ – $1.6 \times 10^{10}$</td>
<td>$2.0 \times 10^9$ – $1.0 \times 10^{10}$</td>
<td>$2.0 \times 10^3$ – $4.4 \times 10^5$</td>
<td>$2.0 \times 10^9$ – $1.0 \times 10^{10}$</td>
<td>$1.2$ – $5.2 \times 10^9$</td>
</tr>
<tr>
<td>$Re_y$</td>
<td>$2.0 \times 10^9$ – $1.6 \times 10^{10}$</td>
<td>$2.0 \times 10^9$ – $1.0 \times 10^{10}$</td>
<td>$2.0 \times 10^3$ – $4.4 \times 10^5$</td>
<td>$2.0 \times 10^9$ – $1.0 \times 10^{10}$</td>
<td>$1.2$ – $5.2 \times 10^9$</td>
</tr>
<tr>
<td>$Ro$</td>
<td>$2.0 \times 10^5$ – $5.0 \times 10^7$</td>
<td>$2.0 \times 10^5$ – $8.0 \times 10^5$</td>
<td>$4.0 \times 10^6$ – $5.0 \times 10^6$</td>
<td>$8.0 \times 10^5$ – $3.0 \times 10^6$</td>
<td>$1.5$ – $6.6 \times 10^5$</td>
</tr>
<tr>
<td>$Gr$</td>
<td>$1.5 \times 10^5$ – $1.0 \times 10^{11}$</td>
<td>$1.7 \times 10^{11}$</td>
<td>$3.8 \times 10^7$ – $4.5 \times 10^9$</td>
<td>$1.7 \times 10^9$ – $9.0 \times 10^9$</td>
<td>$5.9 \times 10^9$ – $4.4 \times 10^{10}$</td>
</tr>
<tr>
<td>Heated shroud</td>
<td>no</td>
<td>no</td>
<td>yes</td>
<td>yes</td>
<td>yes</td>
</tr>
<tr>
<td>$a$</td>
<td>45 mm</td>
<td>120 mm</td>
<td>0.319 b</td>
<td>71 mm</td>
<td>71 mm</td>
</tr>
<tr>
<td>$b$</td>
<td>426 mm</td>
<td>400 mm</td>
<td>b</td>
<td>220 mm</td>
<td>220 mm</td>
</tr>
<tr>
<td>$d$</td>
<td>59 mm</td>
<td>80 mm</td>
<td>0.195 b</td>
<td>42.9 mm</td>
<td>42.9 mm</td>
</tr>
<tr>
<td>$r_s$</td>
<td>0</td>
<td>102 mm</td>
<td>0.272 b</td>
<td>52 mm</td>
<td>52 mm</td>
</tr>
</tbody>
</table>

### Table 2.2: Range of parameters for the most relevant numerical studies available in the literature.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Long &amp; Tucker [94]</th>
<th>Sun et al. [97]</th>
<th>Tian et al. [96]</th>
<th>Tan et al. [99, 100]</th>
</tr>
</thead>
<tbody>
<tr>
<td>$Re_x$</td>
<td>$2.2 \times 10^9$</td>
<td>$4.4 \times 10^4$ – $1.5 \times 10^6$</td>
<td>$1.3 \times 10^3$ – $1.1 \times 10^5$</td>
<td>$2.0 \times 10^9$</td>
</tr>
<tr>
<td>$Re_y$</td>
<td>$1.3 \times 10^4$</td>
<td>$1.0 \times 10^6$ – $2.0 \times 10^6$</td>
<td>$1.3 \times 10^4$ – $1.5 \times 10^5$</td>
<td>$2.0 \times 10^5$ – $8.0 \times 10^5$</td>
</tr>
<tr>
<td>$Ro$</td>
<td>8.0</td>
<td>$0.75$ – $5.0$</td>
<td>$0.83$ – $84$</td>
<td>$0.14$ – $0.56$</td>
</tr>
<tr>
<td>$Gr$</td>
<td>$1.4 \times 10^5$ – $7.4 \times 10^9$</td>
<td>$2.8 \times 10^7$ – $6.9 \times 10^9$</td>
<td>$2.4 \times 10^9$ – $3.8 \times 10^{10}$</td>
<td></td>
</tr>
<tr>
<td>Heated shroud</td>
<td>no</td>
<td>yes</td>
<td>yes</td>
<td>yes</td>
</tr>
<tr>
<td>$a$</td>
<td>10.8 mm</td>
<td>0.319 b</td>
<td>40 mm</td>
<td>120 mm</td>
</tr>
<tr>
<td>$b$</td>
<td>108 mm</td>
<td>b</td>
<td>175 mm</td>
<td>400 mm</td>
</tr>
<tr>
<td>$d$</td>
<td>28.8 mm</td>
<td>0.195 b</td>
<td>40 mm</td>
<td>80 mm</td>
</tr>
<tr>
<td>$r_s$</td>
<td>0</td>
<td>0.272 b</td>
<td>0</td>
<td>102 mm</td>
</tr>
</tbody>
</table>
axial jet and the flow at the outer part of the cavity. The numerical results will also be compared to the case of a sealed cavity, which will give insight into the effect of the axial throughflow on specific regions of the cavity, such as the disk and shroud boundary layers.
Chapter 3
Numerical techniques

In this chapter the numerical method employed in this thesis is described. First, a general description of spectral element methods is given, and a few examples and comparisons with low-order schemes are presented. The presentation focuses mainly on aspects relevant to the implementation of the code Semtex [104], which was used in this research. Secondly, the technique of linear stability analysis is described, as well as the numerical method used to solve the resulting eigenvalue problem. Finally, in the last part of the chapter the modifications implemented in Semtex are discussed and comparisons with well-established results are given for numerical verification purposes.

3.1 Spectral and spectral element methods

Discretisation schemes such as finite differences, finite volume, and standard finite element methods are characterised by having a fixed error decay rate when the number of grid points is increased. In discretisations utilising spectral methods, on the other hand, the rate of convergence is not fixed, but exponential if the solution is smooth. Additionally, spectral methods present lower dispersion and diffusion errors than low-order schemes, which are desirable properties in the context of numerical solutions of the unsteady Navier-Stokes equations [4].

Numerical schemes based on global spectral methods consist of approximating the solution $u(x)$ of a given problem using a Fourier-series expansion, if the solution is periodic, or a certain family of high-order polynomials, if the solution is not periodic. These approximations, denoted by $u^{\delta}(x)$, are given, respectively, by,

$$ u^{\delta}(x) = \sum_{k=-N}^{N} \hat{u}_k e^{ikx}, \quad (3.1) $$

$$ u^{\delta}(x) = \sum_{k=0}^{N} \hat{u}_k T_k(x). \quad (3.2) $$

One common choice for $T_k$ is the family of Chebyshev polynomials, $T_k(\cos \theta) = \cos k\theta$, which are in practice obtained from the recurrence relation $T_0(x) = 1$, $T_1(x) = x$, $T_{n+1} = 2xT_n(x) - T_{n-1}(x)$. Besides providing good accuracy, the equations resulting from the approximations above can be implemented efficiently using fast Fourier transform (FFT) algorithms, and derivatives, which must be evaluated...
Figure 3.1: (a) Infinity norm of the error in the evaluation of the derivative of \( e^{\sin(x)} \) using Fourier expansions (black) and a fourth-order finite difference scheme (blue). (b) Error in the approximation of the derivative of \( e^{-x^2} \) using Chebyshev differentiation. In both cases the errors of the spectral approximations decay faster than \( e^{-N} \).

numerically in any method to solve differential equations, can be evaluated in a straightforward manner in Fourier space. To illustrate the convergence properties of spectral methods, Fig. 3.1 shows how the infinity norm of the error in the calculation of the first derivatives of two functions decays as the number of degrees of freedom is increased. In part (a) of the figure, the periodic function \( f(x) = e^{\sin(x)} \) is considered, and the solution obtained with spectral differentiation using Fourier series expansions is compared with a fourth-order finite difference solution. As expected, using finite differences the error decays following a power-law trend, whereas using the Fourier transform the error decays exponentially, faster than \( e^{-N} \), and quickly reaches the round-off level. In Fig. 3.1(b) a non-periodic function is considered, \( f(x) = e^{-x^2} \), and the solution is obtained using Chebyshev differentiation. Exponential convergence is again observed for this case.

One major limitation of global spectral methods is that they are only able to accommodate simple geometries, since the expansion basis used to approximate the solution – Fourier series or Chebyshev polynomials, for example – spans the entire domain along a given direction. Such methods can be successfully employed, for instance, to study flow through a rectangular channel, where the streamwise and spanwise directions are approximated with Fourier series expansions, whereas high-order polynomials are used as the expansion basis for the wall-normal direction. Another limitation of global spectral methods is that they do not allow local mesh refinement. Chebyshev points used for polynomial interpolation are clustered near the ends of the domain, making this technique a good candidate to discretise the wall-normal direction of a domain between two solid walls. In other cases, such as the radial coordinate of a circular pipe, if Chebyshev points are used the resolution would be unnecessarily fine near the axis, thus reducing the efficiency of the
discretisation. To extend the concept of spectral methods to complex geometries, the *spectral element* method is employed. This technique consists of formulating a Galerkin problem from the governing equations, as in a finite element method, and using high-order polynomials as expansion bases. The remainder of this chapter focuses on spectral element methods, and global spectral methods are not further discussed. For details of the formulation of global spectral methods, the reader may refer to, e.g., Trefethen [105], Boyd [106] and Canuto et al. [107].

### 3.1.1 Spectral element method

In a standard, low-order finite element formulation, a given differential equation is multiplied by a weight function $v$, the solution $u$ is approximated locally by a linear function, and the resulting equation is integrated over the domain $\Omega$, which is split into a number of non-overlapping elements. Mesh refinement then consists of decreasing the element size $h$. In a spectral element formulation, in contrast, the functions used to approximate the solution are high-order polynomials, thus allowing for the resolution to be refined by either decreasing the element size $h$ or by increasing the polynomial order $p$. For this reason these methods are also referred to as spectral/$hp$ element. Here only a brief outline of spectral element discretisations is presented, and the reader may refer to Karniadakis & Sherwin [4] and Henderson [108] for further details.

#### Galerkin formulation

Consider a one-dimensional Poisson problem,

$$\frac{d^2u}{dx^2} + f(x) = 0,$$

where $f$ is a known function and $a \leq x \leq b$, subject to Dirichlet and Neumann boundary conditions at $x = a$ and $x = b$, respectively,

$$u(a) = g_D, \quad \frac{du}{dx}(b) = g_N.$$  \[3.3\]  \[3.4\]

Multiplying Eq. $3.3$ by a test function $v(x)$ and integrating over the domain, one obtains:

$$\int_a^b v(x) \frac{d^2u}{dx^2} dx + \int_a^b v(x) f(x) dx = 0.$$  \[3.5\]

The next step of the formulation consists of integrating the first term by parts, so that the Neumann boundary conditions appear explicitly,

$$\int_a^b \frac{dv}{dx} \frac{du}{dx} dx = \int_a^b v(x) f(x) dx + \left[ v(x) \frac{du}{dx} \right]^b_a.$$  \[3.6\]

Now, the Galerkin formulation consists of approximating both the test function $v(x)$ and the exact solution $u(x)$ using a given set of basis functions $\phi_p(x)$, so that $u^\delta(x)$ represents an approximation of $u(x)$. To impose Dirichlet boundary conditions, it is convenient to write the solution $u^\delta(x)$ as the sum of a homogeneous solution and a solution satisfying the Dirichlet boundary conditions,
\[ u^\delta = u^H + u^D, \]  
(3.7)

with

\[ u^H(a) = 0, \quad u^D(a) = g_D. \]  
(3.8)

When the Galerkin formulation is introduced by approximating the test function and the solution in terms of a set of basis functions, the infinite-dimensional problem becomes discrete and can be solved using a computer. In matrix form, Eq. 3.6 can be written as,

\[ Lu = f + \text{boundary terms}, \]  
(3.9)

where \( L \) is the Laplacian matrix, whose components are given by,

\[ L_{pq} = \int_a^b \frac{d\phi_p}{dx} \frac{d\phi_q}{dx} dx, \]  
(3.10)

and \( f \) is,

\[ f_p = \int_a^b \phi_p f(x) dx. \]  
(3.11)

The solution for \( u^\delta \) is then obtained by inverting \( L \) to obtain the vector \( \hat{u} \), which contains the coefficients \( \hat{u}_p \) of the expansion of \( u^\delta \) in terms of the basis \( \phi_p \). To fully specify the problem, it is necessary to specify the set of basis functions \( \phi_p \), and it is at this stage that spectral element discretisations differ from the standard finite element method.

**Expansion bases and the standard elemental region**

In the context of spectral element methods the expansion bases \( \phi_p \) are invariably polynomials. If \( p \) varies within \( 0 \leq p \leq P \), then the solution is approximated by the following sum,

\[ u^\delta = \sum_{p=0}^{P} \hat{u}_p \phi_p. \]  
(3.12)

Many choices are available for \( \phi_p \), and they are generally classified as either **modal** or **nodal** expansion bases. In modal bases there is a hierarchy in the construction of each basis, as \( \phi_{p-1} \) is contained within \( \phi_p \). If \( P = 5 \), for example, then \( \phi_0 \) is a polynomial of order one, \( \phi_1 \) of order two, and so on. Note, however, that the expansion bases are defined in a way that, within an interval \([a, b]\), \( \phi_0(a) = 1 \), \( \phi_0(b) = 0 \), \( \phi_P(a) = 0 \) and \( \phi_P(b) = 1 \), while \( \phi_p(a) = \phi_p(b) = 0 \) for \( 1 \leq p \leq P - 1 \), so that continuity between multiple elements is satisfied. For this reason \( \phi_P \) is a first-order polynomial which satisfies the conditions stated above. Modal expansion bases are constructed in terms of Jacobi polynomials \( J^{\alpha,\beta}_p(x) \), as they can be computed efficiently and are able to deliver mass matrices with a high degree of orthogonality. When nodal expansion bases are used, the \( P + 1 \) \( \phi_p \) bases are constructed using Lagrangian interpolants, which are defined on a set of nodal points. With this choice, all polynomials \( \phi_p \) are of order \( P \), and therefore nodal bases are
not hierarchical. Nodal expansion bases are described in more detail below, as the code used throughout this research employs a basis of this type.

A set of Lagrange polynomials \( h_p(\xi) \), with \( 0 \leq p \leq P \), is constructed from a given set of \( P + 1 \) nodal points \( \xi_q \), and each \( h_p \) is defined to be one at \( \xi = \xi_p \) and zero for \( \xi = \xi_q, q \neq p \),

\[
h_p(\xi) = \prod_{q=0,q\neq p}^{q=P} \frac{\xi - \xi_q}{\xi_p - \xi_q}.
\]

(3.13)

In nodal spectral element formulations it is customary to define the set \( \xi_q \) to be the roots of \((1 - \xi^2)L'_P(\xi) = 0\), where \( L_P(\xi) \) is the Legendre polynomial of degree \( P \). Legendre polynomials are a special case of Jacobi polynomials \( J^{\alpha,\beta}_P(x) \) with \( \alpha = \beta = 0 \), and the roots of \((1 - \xi^2)L'_P(\xi) = 0\) are within the interval \(-1 \leq \xi \leq 1\). As noted by Henderson [108], this choice of nodal points, which is referred to as Gauss-Lobatto-Legendre (GLL), is convenient for many reasons. First, the expansion of any smooth function using Lagrangian interpolants at the GLL points converges exponentially fast; second, from the definition of the Lagrangian interpolants it is clear that the expansion coefficients \( \hat{u}_p \) correspond to the approximation of the solution at the nodal points, \( u_\delta(\xi_p) = \hat{u}_p \). Additionally, the GLL points are associated with a set of weights \( w_q \) which allow for efficient numerical integration using Gauss quadrature. Finally, the family of Jacobi polynomials can be evaluated by recursive algorithms in an efficient way, which means that the GLL points, the associated weights and the Lagrangian interpolants can be evaluated efficiently in the pre-processing stage of a given calculation, without relying on tabulated values. Figure [3.2] shows the shape of the Lagrangian interpolants of degree 7 using the GLL points. Note that for each curve the interpolant is unity at one nodal point and zero on the others. In practice, then, when this particular set of basis functions is used the solution of a given problem is approximated as a combination of the curves shown in Fig. [3.2], each of which is multiplied by a coefficient \( \hat{u}_p \). When the Lagrange polynomials are calculated considering the GLL nodes, they can be written as,

\[
h_p(\xi) = \begin{cases} 
1, & \xi = \xi_p, \\
\frac{(1 - \xi^2)L'_P(\xi)}{P(P + 1)L_P(\xi_p)(\xi - \xi_p)}, & \text{otherwise}.
\end{cases}
\]

(3.14)

In spectral element formulations a standard elemental region is defined as \( \Omega_{st} = \{ \xi \mid -1 \leq \xi \leq 1 \} \), and operations such as integration and differentiation are performed locally, within the standard region. In practice, the mapping from an arbitrary region to the standard region is performed considering the Jacobian of the transformation to evaluate integrals and derivatives. When more than one element is present, the full system is assembled taking all the elemental contributions into account via a technique called direct stiffness summation, which results in global matrices with high sparsity which can thus be computed and stored efficiently. For further details, the reader is referred to Karniadakis & Sherwin [4].

Elemental operations

As mentioned earlier, the operations of integration and differentiation are performed at an elemental level. For numerical integration, Gauss quadrature rules are used,
Figure 3.2: Shape of the Lagrangian interpolants of order $P = 7$ defined on the Gauss-Lobatto-Legendre nodes, which are represented by black circles.

which require a set of $Q$ points $x_q$ and associated weights $w_q$ to integrate a given function,

$$\int_{-1}^{1} f(x) \, dx = \sum_{q=0}^{Q-1} w_q f(x_q).$$

(3.15)

In spectral element formulations it is convenient to use the GLL points and weights to perform numerical integration. In nodal discretisations, this choice leads to a diagonal elemental mass matrix $M^e$ if $Q = P + 1$ points are used for the integration,

$$M^e = \int_{-1}^{1} h_p(\xi) h_q(\xi) \, d\xi = \sum_{i=0}^{P} w_i h_p(\xi_i) h_q(\xi_i) = w_p \delta_{pq},$$

(3.16)

where the property $h_p(\xi_i) = \delta_{pi}$ was considered, with $\delta$ denoting the Kronecker delta. The numerical integration of other matrices and operators are performed using the procedure described above. When a derivative is involved, it is only necessary to calculate the derivative of the expansion basis $h_p(\xi)$ on a set of nodal points,

$$\frac{du(\xi)}{d\xi} = \sum_{i=0}^{Q-1} u(\xi_i) \frac{dh_i(\xi)}{d\xi} = \sum_{j=0}^{Q-1} d_{ij} u(\xi_j),$$

(3.17)
where \( d_{ij} \) are the entries of a differentiation matrix \( \mathbf{D} \), which can be derived analytically for the GLL polynomials.

**Two-dimensional expansion bases**

The description provided above considered a one-dimensional formulation only, but it covers the fundamental aspects of spectral element discretisations. In two dimensions, if the discretisation is limited to rectangular elements, then a two-dimensional expansion basis is constructed from the tensor product of two bases,

\[
\phi_{pq}(\xi_1, \xi_2) = h_p(\xi_1)h_q(\xi_2),
\]

where the normalised coordinates \( \xi_1 \) and \( \xi_2 \) form a standard region such that \( \Omega_{st} = \{-1 \leq \xi_1, \xi_2 \leq 1\} \). In the implementation used in the present work, this expansion basis is used and the polynomial order is the same along the two directions, although it is possible to formulate the basis with distinct values of \( P \) for \( \xi_1 \) and \( \xi_2 \). It should be noted that two-dimensional expansion bases can also be formulated using triangular elements \([4]\), which is a convenient option to accommodate more complex geometries. Also, in the context of quadrilateral elements, it is possible to use non-conforming elements \([4,108]\), which allow better local mesh refinement, but such techniques are not considered in this work.

### 3.1.2 Semtex spectral element code

In this research the spectral element code *Semtex* \([104]\) was employed, as well as an extension which considers an advection-diffusion equation for a passive scalar. Centrifugal buoyancy terms were implemented so that the code could be used to investigate buoyancy-induced flows in rotating cavities. In the code a spectral element-Fourier formulation, originally proposed by Karniadakis \([109]\), is used. Using Fourier expansions in one direction is an efficient way to investigate problems where one direction is homogeneous, such as in channel and pipe flow, or if the geometry is axisymmetric in cylindrical coordinates, where the azimuthal direction is naturally homogeneous. The spectral element discretisation is obtained using a nodal expansion basis; in particular, Lagrange polynomials interpolated at the GLL nodes are considered, just as explained above. Only quadrilateral elements are permitted, so that the two-dimensional expansion basis is constructed from the tensor product of one-dimensional bases. Therefore, the three-dimensional expansion basis used in *Semtex* can be written as,

\[
\phi_{pqr}(\xi_1, \xi_2, \xi_3) = h_p(\xi_1)h_q(\xi_2)e^{i\beta \xi_3},
\]

where \( \beta = \frac{2\pi}{L_{\xi_3}} \) is the wavenumber, \( L_{\xi_3} \) being the domain length along the homogeneous direction. The use of Fourier expansions along one direction allows for efficient evaluation of derivatives in Fourier space, and FFT algorithms are used to transfer from/to physical space. Due to the linearity of the Fourier transform operator, the full three-dimensional problem is reduced to two-dimensional problems, one for each Fourier plane. This separation provides a natural way to parallelise the calculations in a straightforward and efficient way, and constitutes the basis of the parallelisation strategy used within *Semtex*. Note, however, that this limits the
number of processors that can be used to the number of Fourier modes used in a given calculation.

The time integration is based on the semi-implicit, stiffly stable velocity-correction scheme proposed by Karniadakis et al. [110]. Assuming a unity density $\rho = 1$, the Navier-Stokes equations can be written as,

$$\frac{\partial \mathbf{u}}{\partial t} = -\nabla p - \mathbf{N}(\mathbf{u}) + \nu \mathbf{L}(\mathbf{u}), \quad \nabla \cdot \mathbf{u} = 0,$$

(3.20)

where $\mathbf{N}(\mathbf{u})$ and $\nu \mathbf{L}(\mathbf{u})$ are the advective and diffusive terms, respectively. With this notation, the scheme proposed in [110], which is used in Semtex, is,

$$\hat{\mathbf{u}} - \mathbf{u}^n \quad \Delta t = \sum_{q=0}^{J_e-1} \beta_q \mathbf{N}(\mathbf{u}^{n-q}),$$

(3.21a)

$$\hat{\mathbf{u}} - \hat{\mathbf{u}} \quad \Delta t = -\nabla p^{n+1},$$

(3.21b)

$$\mathbf{u}^{n+1} - \hat{\mathbf{u}} \quad \Delta t = \nu \sum_{q=0}^{J_i-1} \gamma_q \mathbf{L}(\mathbf{u}^{n+1-q}),$$

(3.21c)

where $\hat{\mathbf{u}}$ and $\hat{\mathbf{u}}$ are intermediate velocity fields and $\bar{p}$ is the pressure field at the time level $n + 1$ which ensures that the velocity is divergence-free. In the equations above $J_e$ and $J_i$ are the orders of integration for the evaluation of the non-linear and diffusive terms, respectively, whereas $\beta_q$ and $\gamma_q$ are the corresponding integration coefficients, which can be found in [110]. Taking the divergence of Eq. 3.21(b), and considering that $\nabla \cdot \hat{\mathbf{u}} = 0$, a Poisson equation for the pressure is obtained, which is solved considering a high-order Neumann boundary condition [110],

$$\frac{\partial \bar{p}^{n+1}}{\partial n} = \mathbf{n} \cdot \left[ \sum_{q=0}^{J_e-1} \beta_q \mathbf{N}(\mathbf{u}^{n-q}) + \nu \sum_{q=0}^{J_i-1} \gamma_q \nabla Q^{n+1-q} + \nu \sum_{q=0}^{J_e-1} \beta_q \left( -\nabla \times (\nabla \times \mathbf{u})^{n-q} \right) \right],$$

(3.22)

with $Q \equiv \nabla \cdot \mathbf{u}$ and $\mathbf{n}$ denoting the direction normal to the boundary. The updated velocity field $\mathbf{u}^{n+1}$ is obtained by solving a Helmholtz equation, Eq. 3.21(c), where $\hat{\mathbf{u}}$ enters as a forcing term. Note that the viscous term is solved implicitly, while the non-linear term is evaluated explicitly. It is convenient to evaluate the non-linear terms in a skew-symmetric form, $\mathbf{N}(\mathbf{u}) = 0.5 [\mathbf{u} \cdot \nabla \mathbf{u} + \nabla \cdot (\mathbf{uu})]$, as this form reduces aliasing errors. In Semtex, specifically, the non-linear terms are evaluated in an alternating skew-symmetric form, so that either the conservative or non-conservative form is evaluated in alternating time steps.

In the cases considered in this research, the implementation of Semtex in cylindrical coordinates was considered, which is described in detail by Blackburn & Sherwin [104]. One challenge in the use of cylindrical coordinates is the appearance of a geometrical singularity on the axis, which can be treated by multiplying the governing equations by the radius $r$, in which case the singular terms can be zeroed without loss of generality, since they do not contribute to the integral formulation of the problem [104]. With this formulation the expansion basis $h_p(\xi)$ can be used along the radial and axial directions while preserving the spectral convergence of the scheme.
To solve the governing equations in a rotating reference frame and to account for centrifugal buoyancy, it is necessary to add the Coriolis acceleration and the buoyancy term presented in Chapter 2 to the momentum equations. This is achieved by adding these terms directly to the non-linear term in physical space, so that these extra forces are evaluated explicitly at each time step.

**Spectral vanishing viscosity (SVV)**

When performing numerical simulations of turbulent flows, a direct numerical solution (DNS) of the Navier-Stokes equations is usually not feasible due to the large number of grid points and small size of the time step required to obtain a converged and stable solution. For this reason, a turbulence model is generally employed, such as a RANS-based model, or a LES technique, in which the filtered Navier-Stokes equations are solved using a given sub-grid scale (SGS) model.

As an alternative to SGS modelling in LES, it is also possible to stabilise a numerical solution without applying a filter to the governing equations. In the context of spectral methods, one technique of this kind consists of adding a controlled amount of dissipation to the governing equations for wavenumbers higher than a given threshold. The justification for this is that, if the resolution is not fine enough, then the energy tends to accumulate at the small scales (large wavenumbers), which can compromise the stability of the solution. Thus, adding an artificial viscosity to the large wavenumbers may dissipate more energy and stabilise the solution, without affecting the behaviour of the larger scales, which are still solved directly, i.e., without the use of an augmented viscosity. Since the artificial viscosity is only applied to a specific region of the spectrum, this method is called spectral vanishing viscosity (SVV). It can be interpreted as an implicit LES method, since no sub-grid model is utilised, or as an under-resolved DNS technique.

Karamanos & Karniadakis [111] have applied a SVV method to the one-dimensional Burgers equation and to a turbulent channel flow. As mentioned, artificial dissipation is only added for wavenumbers larger than a threshold, say, $M$. Therefore, the dissipation will be zero for $k \leq M$, whilst for $M < k \leq N$, where $N$ is the number of modes, the amount of dissipation will be controlled by an amplitude $\epsilon$ and by a kernel $\hat{Q}_k$. In [111], the authors have employed the kernel given by Eq. 3.23, which is shown for different values of $M$ in Fig. 3.3. This kernel provides a smooth increase of the artificial viscosity, and the figure shows that the smaller the value of $M$, the bigger the region of the spectrum where the dissipation is added. Karamanos & Karniadakis [111] have shown that the convergence properties of a spectral element formulation are preserved when the SVV kernel is applied, and that the additional computational cost is only 1% higher than for the case without the SVV operator. These two features make the SVV a very attractive method to perform under-resolved calculations involving turbulent flows. It should be noted that for a classical LES technique, such as the Smagorinsky model, which is based on the eddy-viscosity hypothesis, the increase in computational effort is more significant since it is necessary to evaluate the SGS tensor. For a turbulent channel flow simulation at $Re_\tau = 180$, Karamanos & Karniadakis [111] employed two meshes, one corresponding to $2.82 \times 10^4$ degrees of freedom and a finer mesh with $1.76 \times 10^5$. The results were compared with experimental data and with published DNS results obtained on a grid with $3.96 \times 10^6$ degrees of freedom. Even with the coarse mesh the rms quantities were in good agreement with the DNS results, and the authors
have shown how the use of SVV improves the results in comparison with the case without artificial dissipation.

\[ \hat{Q}_k = \exp \left( -\frac{(k - N)^2}{(k - M)^2} \right), \quad M < k \leq N. \]  

(3.23)

![Figure 3.3: SVV kernel $\hat{Q}_k$ (Eq. 3.23) for different values of $M$.](image)

Pasquetti [112] investigated the influence of the SVV parameters on the numerical solution of turbulent flow past a circular cylinder, and concluded that the choice of $M$ and $\epsilon$ does not have a major influence on the solution, except in the wake region very close to the cylinder, where the use of a large $M$ and small $\epsilon$ yielded better results compared to experimental data. Analysing one-dimensional energy spectra, the author observed, as expected, that when large values of $M$ and small values of $\epsilon$ are employed more high-frequency content is contained in the spectrum.

The SVV kernel given by Eq. 3.23 is implemented in *Semtex*, and details about the implementation are given by Koal et al. [113]. In the Fourier direction, the inclusion of SVV is straightforward, since the only modification in the code consists of augmenting the viscosity in the Fourier modes with wavenumber $k > M$. To illustrate this, consider a one-dimensional diffusion equation, Eq. 3.24,

\[ \frac{\partial u}{\partial t} = \nu \frac{\partial^2 u}{\partial x^2} \text{ for } x \in [0, 2\pi]. \]  

(3.24)

Applying a Fourier transform in $x$ to this equation, we obtain equations for the Fourier modes $k$:

\[ \frac{\partial \hat{u}_k}{\partial t} = -\nu k^2 \hat{u}_k \text{ for } -N \leq k \leq N. \]  

(3.25)

It is at this stage that the SVV term is added. As mentioned, this is accomplished by augmenting the viscosity by $\epsilon \hat{Q}_k$, as shown in Eq. 3.26. Here we note that it is the kernel $\hat{Q}_k$ that controls which modes will be affected by the artificial dissipation.

\[ \frac{\partial \hat{u}_k}{\partial t} = -(\nu + \epsilon \hat{Q}_k) k^2 \hat{u}_k \text{ for } -N \leq k \leq N. \]  

(3.26)

As noted in [113], a different treatment is required to apply the SVV filter to the non-homogeneous directions, i.e., the two directions of the domain discretised.
CHAPTER 3

with a nodal spectral element method. This is accomplished by applying a discrete polynomial transform to the differentiation matrix $D$, which is used to evaluate the diffusion operator. In polynomial space, the SVV filter can be applied in a manner analogous to that employed in the Fourier direction. Koal et al. [113] employed the symmetric form of the SVV operator proposed by Xu & Pasquetti [114], given by Eq. 3.27

$$D^{SVV} = T^{-1} \text{diag}(1 + \frac{\xi}{\nu} \hat{Q}_k) TD,$$  \hspace{1cm} (3.27)

where $T$ is the transform matrix from physical to modal (polynomial) space, given by Eq. 3.28 (Canuto et al. [107]),

$$T_{ij} = \frac{L_i(\xi_j)w_j}{\gamma_i}, \quad \gamma_i = \begin{cases} 
\frac{1}{k + 0.5}, & i < N, \\
\frac{N}{2}, & i = N.
\end{cases}$$ \hspace{1cm} (3.28)

where $L_i$ represents the $i$-th Legendre polynomial, and $\xi_j$ and $w_j$ are the GLL nodes and associated weights, respectively. Finally, the modifications described above are applied to the differentiation operators of the viscous term during preprocessing, and for this reason the computational cost of a simulation with or without the SVV dissipation is the same.

Moura et al. [115] have shown recently that the use of a constant coefficient $\epsilon$ and of the standard kernel $\hat{Q}_k$ has certain limitations. Specifically, adding SVV dissipation in the form described above tends to introduce non-smooth features in regions strongly dominated by advection or diffusion. The authors have then proposed the use of a variable SVV amplitude depending on the local mesh spacing and local advection speed, and also a ‘power’ kernel instead of the traditional exponential kernel. However, this modified SVV operator was only applied to simple one-dimensional advection-diffusion problems, and therefore it is not clear whether it would be robust enough for three-dimensional Navier-Stokes problems. Additionally, such approaches would increase the computational cost and implementation complexity, since it would be necessary to take into account the local mesh spacing, and also the characteristic local advection speed, a quantity which would be potentially difficult to evaluate accurately. For these reasons, in this work we employ the standard (exponential) SVV kernel with a constant amplitude. It should be kept in mind, though, that there is no universal rule for the choice of SVV parameters, therefore the parameters are generally chosen to provide a stable calculation without degrading the solution accuracy.

3.2 Linear stability analysis

In the study of fluid dynamics it is often of interest to investigate the effect of imposing infinitesimal disturbances to a certain base flow, as this can provide useful insight into the onset of three-dimensional motion and transition to turbulence. In general, linear stability analysis problems result in a linearised set of equations, from which an eigensystem is obtained. The eigensystem solution, in terms of its eigenvalues and corresponding eigenvectors, provides information about the stability of the base flow and about the instability modes. In this research a time-stepper
method, described, e.g. by Tuckerman & Barkley [116], is employed to obtain the eigensystem, and an iterative procedure is used to extract the most unstable eigenvalues and eigenvectors only, rather than the full spectrum. When analysing flow stability, only the most unstable modes are of interest, since they drive the transition to a new, unstable state. The use of time-stepper methods for stability analyses is particularly convenient since a code used for DNS calculations can be modified to integrate the linearised perturbation equations rather than the non-linear Navier-Stokes equations. Additional algorithms are then required to extract the eigenvalues and eigenvectors of the system.

Decomposing the velocity field $\mathbf{u}$ as the sum of a base flow $\mathbf{U}$ and a small perturbation $\mathbf{u}'$, $\mathbf{u} = \mathbf{U} + \mathbf{u}'$, and also assuming an analogous decomposition for the pressure, $p = P + p'$, the momentum equations can be written as (assuming $\rho = 1$ for convenience),

$$\frac{\partial (\mathbf{U} + \mathbf{u}')}{\partial t} + (\mathbf{U} + \mathbf{u}') \cdot \nabla (\mathbf{U} + \mathbf{u}') = -\nabla (P + p') + \nu \nabla^2 (\mathbf{U} + \mathbf{u}'), \quad (3.29a)$$

$$\nabla \cdot (\mathbf{U} + \mathbf{u}') = 0. \quad (3.29b)$$

Expanding the equations above, and neglecting the quadratic term $\mathbf{u}' \cdot \nabla \mathbf{u}'$, separate equations are obtained for the evolution of the base flow and of the perturbation. The base flow equations are identical to the Navier-Stokes equations, while the perturbation equations become linear with respect to $\mathbf{u}'$, assuming the following form:

$$\frac{\partial \mathbf{u}'}{\partial t} + \mathbf{u}' \cdot \nabla \mathbf{U} + \mathbf{U} \cdot \nabla \mathbf{u}' = -\nabla p' + \nu \nabla^2 \mathbf{u}', \quad (3.30a)$$

$$\nabla \cdot \mathbf{u}' = 0. \quad (3.30b)$$

Since in an incompressible flow the pressure can be treated as a constraint to satisfy incompressibility of the velocity field, it is convenient to eliminate it from (3.30a) to obtain,

$$\frac{\partial \mathbf{u}'}{\partial t} = \mathbf{L(U)} \mathbf{u}', \quad (3.31)$$

where $\mathbf{L(U)}$ is a linear operator and a function of the base flow $\mathbf{U}$. Using the decomposition $\mathbf{u}'(\mathbf{x}, t) = \mathbf{q}'(\mathbf{x}) e^{\lambda t}$, where $\lambda = \lambda_r + i\lambda_i$ is a complex eigenvalue, and substituting into Eq. [3.31] the following eigensystem is obtained:

$$\mathbf{L(U)} \mathbf{q}' = \lambda \mathbf{q}'. \quad (3.32)$$

When the eigensystem above is solved, it is possible to tell whether the flow is stable or unstable to infinitesimal perturbations by looking at the real part of the eigenvalues. If one eigenvalue has a positive real part, then the flow is unstable, and the shape of the disturbance is obtained from the corresponding eigenvector. In a stability analysis one is usually interested in finding the critical value of a certain parameter, such as the Reynolds or Rayleigh number, for which the leading eigenvalue has a zero real part, where the leading eigenvalue is that with largest real part.
3.2.1 Iterative solution procedure

In practice, assembling the matrix $L(U)$ and calculating all the eigenvalues is impracticable. Fortunately, using iterative methods it is not necessary to construct the $L(U)$ operator, but only its repeated projection on an initial condition, in which case only a subset of the full spectrum is obtained. The simplest of these methods is the power method, in which an initial condition $u'_0$ is used to obtain a sequence of the form $u'_n = L^n u'_0$, which converges to the dominant eigenvector, which is associated with the dominant eigenvalue, i.e. that with largest magnitude. One generalisation of the power method is the Arnoldi, or block power, method, allowing more than one eigenvalue to be calculated. The idea is to generate a Krylov subspace from the repeated applications of $L$ on $u'_0$,

$$u'_0, Lu'_0, L^2 u'_0, \ldots, L^{K-1} u'_0,$$

where $K$ is the Krylov subspace dimension which corresponds to the number of eigenvalues required. The Krylov sequence is then orthonormalised using the Gram-Schmidt process to obtain a basis $v_1, v_2, \ldots, v_K$, or $V(i, k) = v_k(i)$. When the orthonormalisation is performed, an upper triangular matrix $R$ is also obtained, such that $L = VR$. Subsequently, a $K \times K$ Hessenberg matrix is calculated as $H = V^T LV$. When $H$ is diagonalised, its eigenvalues approximate the $K$ dominant eigenvalues of $L$ (see Tuckerman & Barkley [116] and Barkley et al. [117] for further details). Note that, since $V$ is orthonormal, $V^T L = R$ is used to calculate $H$, and therefore the entire procedure described above can be performed without explicitly constructing the operator $L$.

The issue with obtaining the dominant eigenvalues (i.e. those with largest magnitude) of $L$ is that, in the context of the Navier-Stokes equations, they have negative real part and are therefore quickly damped [116]. In a stability analysis one is interested in obtaining the leading eigenvalues, i.e. those with the largest real part. For any matrix $A$, the leading eigenvalues can be obtained by calculating the dominant eigenvalues of $e^{\Delta t A}$ for $\Delta t > 0$ [116]. Conveniently, the general solution of Eq. 3.31 can be written as (assuming that the base flow is steady),

$$u'_\tau = e^{\tau L} u'_0,$$

where the subscript $\tau$ represents the time the solution is integrated for. Thus, the leading eigenvalues of $L$ are obtained by applying the Arnoldi method to $e^{\tau L}$. The eigenvectors obtained for $e^{\tau L}$ are directly equivalent to those of $L$, but its eigenvalues $\mu_j$ are related to $\lambda_j$ by $\mu_j = e^{\lambda_j \tau}$. Again, the operator does not need to be constructed explicitly, but instead only its projection on the initial condition is used, which is effectively the solution of the linearised equations 3.30 for a time $\tau$. The Krylov sequence is then obtained by performing the integration $K$ times.

3.2.2 Stability analysis using Semtex

As mentioned above, using time-stepper methods one can adapt existing DNS codes to perform stability calculations. An extension of Semtex called Dag (direct optimal growth) has the procedure described above implemented. In this research only the stability of steady base flows is considered, but within the code it is possible to analyse the stability of time-periodic flows via Floquet analysis as well as the optimal
growth of perturbations. Further details about these functionalities are described by Barkley et al. [117] and Blackburn et al. [118].

The implementation is restricted to two-dimensional base flows, which can have either two or three velocity components. The perturbation, however, can be three-dimensional, as Fourier expansions are used along the homogeneous direction, in which case the flow stability for each wavenumber $\beta$ is analysed separately.

In Chapter 5 the linear stability analysis code is used to investigate the onset of convection in a rotating cavity. To perform this analysis it was necessary to modify the implementation used in Dog to include the energy equation for the temperature perturbation and the buoyancy terms in the linearised momentum equations.

### 3.3 Numerical verification test cases

The base version of Semtex is well validated, and throughout this research results for classical test cases of turbulent flow, such as channel and pipe flow, were reproduced successfully. In a preliminary study to assess the performance of the code for natural convection flows, a test case of turbulent flow in a differentially heated tall cavity of aspect ratio 4:1 was studied. The results are published in [119], and are presented in Appendix [3]. In the paper it is shown that the results obtained with Semtex agree well with the benchmark DNS results of Trias et al. [120], in terms of velocity profiles in different vertical positions, as well as of second-order statistics. For comparison purposes, the same test case was investigated using ANSYS Fluent without a turbulence model, on a mesh with approximately the same number of degrees of freedom as that considered in Semtex. While the results obtained with Fluent were able to predict the Nusselt number and the velocity profiles in the centre of the cavity, where turbulent effects are weaker, poor predictions were obtained for the second-order quantities and even for the velocity profiles near the top of the cavity. This preliminary investigation encouraged the use of Semtex for flows involving natural convection.

As mentioned earlier, the centrifugal buoyancy terms are not implemented in the base distribution of Semtex, and the energy equation is not implemented in Dog. These modifications were implemented in the two codes, and in the next subsections test cases used for numerical verification are presented.

#### 3.3.1 Centrifugal buoyancy term in Semtex

The modifications required to account for centrifugal buoyancy in a rotating reference frame are straightforward to implement. Inclusion of the Coriolis force requires two new components in the radial and azimuthal directions, whereas the centrifugal buoyancy term is added in the radial direction.

The implementation was verified by comparisons with the DNS results reported by Read et al. [66] for a baroclinic flow, where the Froude number is increased and the flow changes from being dominated by baroclinic to centrifugal effects. In this case buoyancy effects arise from both the gravitational acceleration in the axial direction and the centrifugal force. The geometry considered is the annulus shown in Fig. 2.6(a), with $a = 0.0348$ m, $b = 0.0602$ m and $d = 0.1$ m, and air is used as the working fluid, with $\nu = 1.697 \times 10^{-5}$ m$^2$/s and $\kappa = 2.400 \times 10^{-5}$ m$^2$/s, thus giving $Pr = 0.707$. The inner cylinder is cooled and the outer is heated, so
that $\Delta T = T_b - T_a$, while the horizontal disks are thermally insulated. The flow is governed by the Prandtl number $Pr$, the Taylor number $Ta$, the gravitational Rayleigh number $Ra_g$, and by the thermal Rossby number $\Theta$, all of which were defined in Chapter 2 and in the Nomenclature. Additionally, the influence of the local Froude number, $Fr_r = \Omega^2 r/g$, is considered. The mesh used in the calculations has 225 spectral elements with polynomial order 7, and 192 planes in the azimuthal direction, thus corresponding to approximately $2.1 \times 10^6$ degrees of freedom. For comparison, Read et al. [66] used a Chebyshev-Chebyshev-Fourier spectral method with mesh sizes varying between $4.92 \times 10^5$ and $3 \times 10^6$ degrees of freedom. Note that the authors also considered cases with Taylor number higher than those presented here.

Figure 3.4 shows temperature maps at the cavity mid-height for three values of $Ta$, both those obtained with Semtex (top row) as well as those extracted from Read et al. [66] (bottom row). The temperature is normalised so that it ranges from -0.5 to 0.5, and the gray and black lines correspond to negative and positive values, respectively. At $Ta = 2.35 \times 10^5$ (Figs. 3.4(a) and (d)), the Froude numbers based on the inner and outer radii are $Fr_a = 0.57$ and $Fr_b = 0.98$, respectively, thus indicating that the gravitational acceleration dominates over the entire radial extent of the domain. For this case the temperature field is rather tilted, and the cold (hot) fluid does not penetrate far into the hot (cold) region. For $Ta = 5 \times 10^5$, $Fr_a = 1.2$ and $Fr_b = 2.1$, therefore centrifugal buoyancy dominates, and as shown the temperature map is less tilted than for $Ta = 2.35 \times 10^5$. Finally, for $Ta = 1.5 \times 10^6$, $Fr_a = 3.6$ and $Fr_b = 6.27$, and centrifugal buoyancy effects are even stronger. At this condition any tilt in the flow structure is not distinguishable, and the pattern is analogous to that of Rayleigh-Bénard convection at relatively low Rayleigh number, where the jets ejected from the hot and cold walls are transported towards the opposite wall.

It should be noted that the patterns shown in Fig. 3.4 drift relative to the walls, and since they correspond to instantaneous snapshots, any differences in phase are irrelevant. Note that in Fig. 3.4(f) the inner radius seems to be smaller than in the other figures. The figure was extracted from the paper by Read et al. [66], and the inner radius was kept constant throughout the study, therefore the difference is probably due to an issue that occurred when the figure was generated.

The simulations performed were able to capture the transition from gravitational to centrifugal convection, which demonstrates that the corresponding terms are implemented correctly.

### 3.3.2 Stability analysis with heat transfer

**Gravitational buoyancy**

The stability analysis solver was modified to account for both gravitational and centrifugal buoyancy, in both Cartesian and cylindrical coordinates. In Cartesian coordinates, a straightforward way to verify the implementation is to consider the problem of Rayleigh-Bénard convection for two infinitely long flat plates. In that case, the critical Rayleigh number is 1708, regardless of the value of the Prandtl number, which was reproduced successfully.

For a finite cavity, various calculations were performed and the results were compared with those reported by Mizushima [121]. The setup consists of a rectangular cavity heated from below and cooled from above, with insulated side-walls. Varying
the cavity width/height ratio, the goal is to find the critical Rayleigh number $Ra_c$, for which the leading eigenvalue has a zero real part. Figure 3.5 shows that $Ra_c$ decreases as the aspect ratio increases, approaching the asymptotic value $Ra_c = 1708$ for an infinitely long cavity. In the figure, the crosses correspond to values of $Ra_c$ obtained from the linear stability analysis code, while the solid line is a trend line. In part (b) of Fig. 3.5, the values obtained are compared with those reported by Mizushima [121]. In all cases the difference between the values of $Ra_c$ is on the order of 0.01% or less.

For cylindrical coordinates, a test case of natural convection in a cylinder heated from below without rotation with insulated side-walls was considered, and the results were compared with those of Wanschura et al. [122]. At the onset of two-dimensional convection, the base flow is simply a conduction temperature profile and the critical Rayleigh number does not depend on the Prandtl number $Pr$. The eigenmodes are axisymmetric, i.e. with azimuthal wavenumber $\beta = 0$. For an aspect ratio (cylinder radius divided by its height) of 1, we obtained the value $Ra_c = 2260.48$. The corresponding value was not explicitly reported by Wanschura et al. [122], but it could be inferred from a plot as approximately $Ra_c = 2266$, which is within less than 1% in comparison with our computed value. For the onset of three-dimensional motion, it is first necessary to calculate the base flow separately, from the DNS
Figure 3.5: Critical Rayleigh number for the onset of convection in a rectangular enclosure heated from below with insulated side-walls. The aspect ratio refers to the cavity width/height ratio. Comparison with the results of Mizushima [121].

code, since in those cases the base flow is no longer stratified, and then perform the stability analysis for different $\beta > 0$, in which case the eigenmodes are three-dimensional and the critical Rayleigh number depends on $Pr$. The results obtained with the implementation used throughout this research are in agreement with those reported by Wanschura et al. [122]. For a Prandtl number $Pr = 0.02$ and aspect ratio 1, for instance, we obtained $Ra_c = 2460.63$, whereas Wanschura et al. [122] reported $Ra_c = 2463$. For $Pr = 1$ and still with an aspect ratio of 1, our value is $Ra_c = 3004.5$, while in ref. [122] $Ra_c = 3017$.

**Gravitational and centrifugal buoyancy**

As in this study the interest is on flows induced by centrifugal buoyancy, the code was further evaluated considering the stability analysis results of Lewis & Nagata [67], who investigated the onset of baroclinic instabilities in a rotating annulus accounting for both gravitational and centrifugal buoyancy. Consider, again, the rotating annulus shown in Fig. 2.6(a) where gravity acts along the negative $z$ direction, and the dimensions are $a = 0.0348$ m, $b = 0.0602$ m and $H = 0.05$ m. The cavity is filled with water with $\nu = 1.01 \times 10^{-6}$ m$^2$/s, $\alpha = 1.41 \times 10^{-7}$ m$^2$/s, thus giving $Pr = 7.16$. The inner and outer cylindrical walls are kept at constant temperatures $T_a$ and $T_b$, respectively, where $\Delta T = T_b - T_a > 0$. The horizontal disks are thermally insulated and a no-slip condition for the velocities is used on all walls. It should be noted that, when performing the stability analysis, the boundary conditions are set for the perturbed quantities, which vanish in the cases where a Dirichlet condition is used for the base flow. Therefore, $T'_a = T'_b = 0$ on the cylindrical walls, whereas $\partial T'/\partial z = 0$ for the disks. The perturbed velocities $u'$ are zero on all walls.

Convection initially sets in in axisymmetric form, therefore the two-dimensional base flow must be computed with the non-linear solver before performing the stability analysis. Having calculated the base flow, we are interested in a thermal Rossby number versus Taylor number ($\Theta - Ta$) diagram, as shown schematically in Fig. 2.6(b), as well as on the corresponding critical wavenumbers for the onset of three-
dimensional motion. Figure 3.6(a) shows a comparison between the current results and those reported by Lewis & Nagata [67]. The critical wavenumber changes as one moves along the stability curve, ranging from 3 to 8, also in agreement with Lewis & Nagata [67]. The level of agreement between the results is excellent, which shows that the code can be used to investigate the onset of three-dimensional convection in buoyancy-induced rotating flows. For comparison purposes, Lewis & Nagata used a $20 \times 20$ two-dimensional mesh with non-uniform spacing, and solved the equations using a second-order finite difference scheme. Here, a mesh with 25 spectral elements and polynomial order 7 was used, which corresponds to approximately three times more degrees of freedom than used by Lewis & Nagata [67]. For illustration purposes, Figs. 3.6(b,c,d) show the shape of the eigenmode associated with $\Theta = 0.3$ and $Ta = 5.39 \times 10^5$, for which the critical wavenumber is 6. The figures represent positive (red) and negative (blue) perturbations of temperature and radial and tangential velocity, respectively.

As noted earlier, the output of greatest interest from a stability analysis using Dog is the growth rate of the leading eigenvalues. This means that the critical values shown in Fig. 3.6 are not a direct output, but instead it is necessary to perform a zero-search for the growth rate of the most unstable eigenvalue. This was achieved by fixing the thermal Rossby number $\Theta$ and then searching for the critical value of $Ta$ associated with a leading eigenvalue of zero real part. Note that this had to be done for different azimuthal wavenumbers, to check which one was associated with the lowest critical value of $Ta$.

### 3.4 Conclusion

In this chapter an overview of the spectral element formulation used throughout this research was presented. High-order methods have favourable properties in the context of numerical simulation of turbulent flows, and are generally preferred over low-order methods when high spatial and temporal accuracy is crucial, as it is the case in direct numerical simulations.

The spectral element-Fourier code Semtex [104] was adapted to account for centrifugal buoyancy. Additionally, the linear stability analysis tool Dog was extended to study buoyant flows. Results obtained with both codes compare favourably with results from the literature, thus showing that these tools are adequate to conduct the numerical simulations considered in this thesis.
Figure 3.6: (a) $\Theta - Ta$ diagram for the onset of three-dimensional instabilities in a rotating annulus. Comparison between results obtained with the linear stability solver used in this research (markers and solid line) and those reported by Lewis & Nagata [67] (dashed line). (b,c,d) eigenmode showing the temperature and radial and tangential velocity perturbations, respectively, for $\Theta = 0.3$ and $Ta = 5.39 \times 10^8$. The critical wavenumber is 6, and positive (negative) perturbations are represented in red (blue).
Chapter 4

Flow in a rotor/stator cavity

In this chapter the problem of turbulent flow in a rotor/stator cavity is investigated using both DNS and LES, considering the stabilisation technique of spectral vanishing viscosity presented in Chapter 3. A canonical configuration which has been previously investigated both experimentally and numerically is considered, and the results are compared with available experimental data. First, DNS results are presented, and subsequently a relatively coarse mesh is used for the LES calculations. The results are compared with both the experimental and DNS data. Since the use of LES is desirable to reduce the computational cost, one of the points analysed in this chapter is how coarse a mesh can be while still preserving good accuracy.

4.1 Problem description

The physical model considered corresponds to that investigated experimentally by Séverac et al. [22], and is shown in Fig. 4.1. The dimensions of the cavity are such that the aspect ratio is \( A = d/(b - a) = 0.2 \) and the curvature parameter is \( R = (b + a)/(b - a) = 1.8 \). We recall that the Reynolds number is defined in terms of the tangential velocity at \( r = b \) and of the outer radius \( b \),

\[
Re_\Omega = \frac{\Omega b^2}{\nu}.
\]

The shaft and the rotor rotate at an uniform angular velocity \( \Omega \), while the stator and the shroud remain stationary. In the experiments the cavity was filled with water, although the value of the kinematic viscosity is arbitrary as long as the Reynolds number remains the same, since the flow is incompressible and isothermal, so that the Prandtl number is not a relevant parameter. The problem is formulated

Some of the results included in this chapter are published in [123]:

in an inertial cylindrical coordinate system, and the boundary conditions are,

\[
\begin{align*}
    u(a, \theta, z) &= (0, \Omega a, 0), \\
    u(b, \theta, z) &= (0, 0, 0), \\
    u(r, \theta, 0) &= (0, \Omega r, 0), \\
    u(r, \theta, d) &= (0, 0, 0),
\end{align*}
\]

where \( u = (u_r, u_\theta, u_z) \). The simulations are performed for \( Re_\Omega = 4 \times 10^5 \) and \( Re_\Omega = 10^6 \), so that the results can be compared to the experimental data available. For reference, in jet engine applications the Reynolds number is on the order of \( 10^7 \), however many experimental rigs operate at lower \( Re_\Omega \).

4.1.1 DNS and LES grids

The numerical investigation was started considering a mesh with \( N_{el} = 400 \) spectral elements of polynomial order \( P = 9 \) and using \( N_\theta = 128 \) azimuthal planes. To reduce the resolution requirements along \( \theta \), a sector of \( 30^\circ \) was used, rather than the full annulus. Initially only the case \( Re_\Omega = 4 \times 10^5 \) was considered, and the calculation was initialised by obtaining a 2D axisymmetric solution, and then projecting it to 3D and adding small-amplitude random perturbations to the velocity field. This is necessary to excite all the azimuthal Fourier modes in \( \theta \). From this preliminary calculation, the friction velocity \( u_\tau \) was calculated on all the surfaces considering the total wall shear stress. As mentioned in Chapter 2, an estimation of the friction velocity is important for the near-wall mesh design. Having obtained estimates for \( u_\tau \), two refined grids were designed such that the first elements are located within \( \Delta r^+ < 10 \) (shaft and shroud) and \( \Delta z^+ < 10 \) (rotor and stator). These grids are shown in Figs. 4.2(a) and (b), and are referred to as D1 and D2, respectively. Note that for grid D2 there are more elements near the stator surface, as the flow is expected to be fully turbulent in this region even for \( Re_\Omega = 4 \times 10^5 \).

It is important to note that the element sizes in wall units referred to above correspond to where the first spectral element is located, which means that the corresponding region is discretised with a high-order polynomial expansion basis. For low-order schemes, in contrast, it is customary to use \( \Delta y^+ \approx 1 \), where \( y \) denotes a generic wall-normal coordinate. Due to the non-uniform distribution of the GLL points used within each element, as shown in Fig. 3.2 using \( \Delta y^+ = 10 \) for the first element near the wall there will be at least one point at \( \Delta y^+ < 1 \) for the polynomial orders considered in this study.

The solutions obtained on the meshes D1 and D2 are considered as a reference, and much coarser meshes L1 and L2 with \( N_{el} = 180 \) and \( N_{el} = 210 \), respectively, are
used for the (implicit) LES calculations. These meshes are shown in Figs. 4.2(c,d). For the L2 mesh, the near-wall elements still respect $\Delta z^+ < 10$ near the rotor and the stator, whereas in the L1 grid the near-wall elements with $\Delta z^+ < 10$ are removed, so that the first elements near the rotor and the stator are located within $\Delta z^+ < 25$. For both LES grids the core of the domain is discretised using considerably fewer elements. This relaxation in the resolution is justified considering that the velocity gradients occur in the near-wall region, while the core is characterised by uniform radial and tangential velocities in the time-average sense.

### 4.2 DNS results - $Re_\Omega = 4 \times 10^5$

In this section reference results are presented for $Re_\Omega = 4 \times 10^5$, which are later compared with LES predictions. First, a grid-independence study is carried out, where it is shown that first- and second-order flow statistics compare favourably with the experimental data available, and that only minor differences are observed between the results obtained on three meshes with significantly different numbers of degrees of freedom. Then, results are presented considering time history data for probes located inside the rotor and stator boundary layers, and flow statistics at different radial positions are discussed.

Figure 4.2: Spectral element meshes used for the (a,b) DNS and (c,d) LES calculations.
Table 4.1: Mesh parameters considered for the grid-independence study performed for $Re_\Omega = 4 \times 10^5$.

<table>
<thead>
<tr>
<th>Grid</th>
<th>$N_{el}$</th>
<th>$P$</th>
<th>$N_\theta$</th>
<th>DOF</th>
</tr>
</thead>
<tbody>
<tr>
<td>D1-6-192</td>
<td>840</td>
<td>6</td>
<td>192</td>
<td>$5.8 \times 10^6$</td>
</tr>
<tr>
<td>D1-8-192</td>
<td>840</td>
<td>8</td>
<td>192</td>
<td>$10.3 \times 10^6$</td>
</tr>
<tr>
<td>D2-9-256</td>
<td>1300</td>
<td>9</td>
<td>256</td>
<td>$27.0 \times 10^6$</td>
</tr>
</tbody>
</table>

4.2.1 Grid-independence

A grid-independence study was performed considering the case $Re_\Omega = 4 \times 10^5$ without applying the SVV method for stabilisation. Three grids were considered, as shown in Table 4.1 where DOF indicates the number of independent degrees of freedom for each mesh. To assess grid-independence, profiles of first- and second-order quantities were analysed and compared with the experimental data of Sèverac et al. [22]. Before collecting time-averages, the solutions were monitored in different regions of the rotor and stator boundary layers, as well as in the core, to ensure that the solution was statistically steady. Then, averages were collected for about 10 disk revolutions, and the time-averaged profiles were also averaged along the $\theta$ direction.

Figure 4.3 shows profiles obtained at the mid-radial position $r^* = 0.5$, where $r^* = (r-a)/(b-a)$. The velocity fluctuations are given by $u_{\theta,rms} = (\bar{u}_\theta^2 - \bar{u}_\theta \bar{u}_\theta)^{1/2}$ and $u_{r,rms} = (\bar{u}_r^2 - \bar{u}_r \bar{u}_r)^{1/2}$. All the velocities and velocity fluctuations are normalised by the local disk speed $\Omega_r$, and the axial coordinate is normalised by the cavity width $d$. In the figures only the meshes D1-6-192 and D1-8-192 are considered, since the results obtained on the D2-9-256 mesh are nearly indistinguishable from those obtained on D1-8-192. As shown in parts (a) and (b) of Fig. 4.3, the tangential and radial velocity profiles obtained on the two meshes are almost identical, and are in good agreement with the experimental data. For the velocity fluctuations, on the other hand, more significant differences, but still small, are observed, particularly in the stator boundary layer region (recall that the stator is at $z^* = 1$). Using the D2-9-256 grid, which is more refined in the stator region, the results match those obtained on the D1-8-192 mesh, therefore it can be assumed that this grid is adequate for the present investigation. It should be noted that the radial locations $r^* = 0.3$ and $r^* = 0.7$ were also analysed and compared with the experimental data, but in these regions the differences between the results obtained on the meshes D1-6-192 and D1-8-192 were insignificant, therefore only comparisons for $r^* = 0.5$ are shown here.

To analyse the effect of the three grids on a global quantity, the moment coefficient $C_m$ on the rotor disk was considered, which is defined as [12] [124],

$$C_m = \frac{M_t}{0.5 \rho \Omega_r^2 b^5},$$

where $M_t$ is the moment on the rotating disk, given by,

$$M_t = \int_a^b 2\pi r^2 \tau_t dr,$$

with $\tau_t$ denoting the tangential wall shear stress. As shown in Table 4.2, the differences between the values of $C_m$ obtained on the three grids is rather small considering
that this quantity is quite sensitive to the near-wall resolution. Also, the values are within approximately 10% of the correlation obtained experimentally by Daily & Nece [12], given by,

$$C_m = 0.051(d/b)^{0.1} Re^{0.2}_{\Omega},$$  \hspace{1cm} (4.4)$$

where $d$ is the inter-disk spacing. Considering that the correlation was obtained for a cavity with an inner shaft much smaller than the disk outer radius, the results obtained numerically are in reasonable agreement with the correlation.

The time step was chosen considering the maximum value of $\Delta t$ such that the solution was numerically stable. In contrast with fully implicit time discretisations, the semi-implicit scheme used in this study naturally imposes a constraint on the time step. Halving the time step used, no difference was observed in the statistics, therefore it is considered that the maximum time step giving a stable solution is within the asymptotic limit.
Another important consideration is the sector size used, of 30°. At sufficiently high Reynolds numbers, the flow is expected to be dominated by small-scale structures, therefore the use of a small sector is justified. Note that if one is interested in studying the spiral structures that arise during transition at lower $Re_\Omega$, as in Fig. 4.4(a), the use of a small sector would be inadequate, as it would prevent the formation of azimuthally-elongated structures. To ensure that the choice of a 30° sector does not influence the results presented here, a calculation using a 60° sector was run on the D1 mesh with polynomial order $P = 8$ and using $N_\theta = 384$. This mesh is then equivalent to D1-8-192, except that the number of Fourier modes is doubled to preserve the resolution in the azimuthal direction. Again, the results obtained are identical to those of the 30° sector, so this value was used in all the cases considered here.

### 4.2.2 Results and discussion

*Instantaneous fields*

As mentioned in the literature review, in rotor/stator flows the stator boundary layer becomes turbulent at a lower Reynolds number than the rotor boundary layer, and for this reason situations where laminar, transitional and turbulent flow appear are common. To illustrate the flow behaviour qualitatively, iso-surfaces of the $Q$-criterion are shown in Fig. 4.4 for $Re_\Omega = 4 \times 10^5$ and $Re_\Omega = 10^6$, where in each case views from the rotor and stator sides are on the left- and right-hand sides, respectively. The $Q$-criterion is a scalar quantity obtained from the symmetric $S$ and antisymmetric $\Omega$ parts of the velocity gradient tensor $\nabla \mathbf{u}$, which are given, respectively, by, $S_{ij} = \frac{1}{2}(u_{i,j} + u_{j,i})$ and $\Omega_{ij} = \frac{1}{2}(u_{i,j} - u_{j,i})$. Note that $\Omega$ should not be confused with the angular velocity vector. Denoting the norm of these two tensors by $\|S\| = \text{tr}(SS^T)^{1/2}$ and $\|\Omega\| = \text{tr}(\Omega\Omega^T)^{1/2}$, the quantity $Q$ is given by (see Jeong & Hussain [125]),

$$Q = \frac{1}{2} \left( \|\Omega\|^2 - \|S\|^2 \right).$$

$Q$ provides a balance between shear strain rate and vorticity in the flow. For the lower $Re_\Omega$ considered, a substantial part of the rotor layer is dominated by large-scale structures, thus suggesting that the flow is laminar in this region, whereas it becomes turbulent at larger radii, where much smaller structures form. When $Re_\Omega = 10^6$, the rotor region is almost entirely dominated by small-scale structures, although relatively large structures can be seen at low radii, which suggests that a small transitional region still exists even for this high value of the Reynolds number. It should be noted that in the core region the structures are significantly larger than near the boundary layers, which is expected if we consider that vorticity is generated

<table>
<thead>
<tr>
<th>Grid</th>
<th>$C_m$</th>
<th>$C_m$ [12]</th>
</tr>
</thead>
<tbody>
<tr>
<td>D1-6-192</td>
<td>$2.802 \times 10^{-3}$</td>
<td></td>
</tr>
<tr>
<td>D1-8-192</td>
<td>$2.859 \times 10^{-3}$</td>
<td>$3.182 \times 10^{-3}$</td>
</tr>
<tr>
<td>D2-9-256</td>
<td>$2.897 \times 10^{-3}$</td>
<td></td>
</tr>
</tbody>
</table>

Table 4.2: Moment coefficient $C_m$ obtained with the three DNS grids for $Re_\Omega = 4 \times 10^5$, compared with the experimental correlation proposed by Daily & Nece [12].
near the walls, whereas in the core turbulent fluctuations are weaker. This is further evidenced by the mean profiles shown in Figs. 4.3 and 4.7.

Figure 4.5 shows time series of the normalised radial velocity \(u_r^* = u_r/(\Omega r)\) for one disk revolution, where \(t^* = \Omega t/(2\pi)\), inside the rotor boundary layer at \(z^* = 0.01\) and the stator boundary layer at \(z^* = 0.99\), where \(z^* = z/d\). The plots clearly illustrate the different nature of the two boundary layers. In particular, in the rotor boundary layer at \(r^* = 0.3\) the motion occurs at a rather small frequency, and the value of \(u_r^*\) varies weakly over time. It is interesting to note that for \(r^* = 0.7\) the rotor layer contains a broader frequency content, but a clear signature of lower-frequency motions is still present, in contrast with the signals obtained for the stator boundary layer. To analyse the energy distribution across the frequency spectrum, normalised power spectral density (PSD) plots are presented in Fig. 4.6, where the PSD values are obtained from a Fourier transform of radial velocity time signals collected over eight disk revolutions. For the probes located in the rotor boundary layer (top row of the figure), an inertial sub-layer is not observed, even at \(r^* = 0.7\), which suggests that transitional effects are strong in this region. The spectrum for the \(r^* = 0.3\) probe is more characteristic of laminar flows, as virtually all the energy is contained in the low frequencies. The PSD curves obtained on the stator boundary layer are characteristic of turbulent flows, with the spectrum decreasing monotonically with the frequency and without any energy pile-up at high frequencies. It should be noted, however, that the presence of an inertial sub-range, indicated by the \(f^{-5/3}\) curve shown in red, is not very evident, and seems to comprise only a narrow range of the spectrum.

**Time-averaged fields**

First- and second-order statistics were collected for approximately 10 disk revolutions, and compared with the experimental data available at three radial stations, \(r^* = 0.3\), \(r^* = 0.5\) and \(r^* = 0.7\). Figure 4.7 shows normalised profiles for \(r^* = 0.3\) and \(r^* = 0.7\), while the results for \(r^* = 0.5\) are presented in Fig. 4.3. We recall that the results presented were obtained on the D1-8-192 grid. Overall, the profiles are in good agreement with the experimental measurements at the three radial locations. There are, however, differences in the radial velocity profiles \(u_r^*\) near the stator, par-
Figure 4.5: Time series of normalised radial velocity $u_r^*$ over one disk revolution at three radial stations, for (a) $z^* = 0.01$ (rotor) and (b) $z^* = 0.99$ (stator). $Re_\Omega = 4 \times 10^5$.

Figure 4.6: Power spectral densities of radial velocity obtained from time series near the rotor at $z^* = 0.01$ (top row) and the stator at $z^* = 0.99$ (bottom row), for different radial positions. The red curves represent a $f^{-5/3}$ slope, characteristic of the inertial sub-range in turbulent flows. $Re_\Omega = 4 \times 10^5$. 
particularly for $r^* = 0.3$ and $r^* = 0.7$. As mentioned in the grid-independence study subsection, these profiles were unaffected when the number of degrees of freedom was varied from $5.8 \times 10^6$ to $27.0 \times 10^6$, therefore we assume that the differences observed are not related to the resolution employed in the computations.

The mean velocity profiles indicate that the flow is of Batchelor type, i.e. it consists of two boundary layers separated by a core with uniform tangential velocity and zero radial velocity, in the time-average sense. Also, the turbulent fluctuations are weaker in the core region than in the boundary layers, as shown in the $u^*_{\theta,rms}$ and $u^*_{r,rms}$ plots. Comparing the profiles for $r^* = 0.3$ and $r^* = 0.7$, the numerical results indicate that velocity fluctuations near the rotor are rather weak at $r^* = 0.3$, but become comparable to those near the stator for $r^* = 0.7$. This result corroborates the qualitative behaviour presented in Fig. 4.4(a), showing finer structures associated with chaotic motion at the outer part of the rotor boundary layer. Note that this behaviour is less evident from the experimental results, as the maximum values of the fluctuations are comparable in the rotor and stator regions, even for $r^* = 0.3$.

Although the numerical results provide good predictions for the fluctuations, in some cases they overestimate or underestimate the peak values of the measurements considerably. It should be noted that the fluctuations are rather small in comparison with the respective mean velocities, as low as around 1% of the local disk speed $\Omega r$ in the core region. Additionally, the experimental uncertainty for the second-order quantities reported by Sèverac et al. [22] is of ±5% of the fluctuations.

Figure 4.7: Normalised profiles of tangential and radial velocity and corresponding fluctuations at two radial stations. The DNS results (solid lines) are compared with the experimental data of Sèverac et al. [22] (circles). $Re_{\Omega} = 4 \times 10^5$.

Figure 4.8(a) shows plots of radial versus tangential velocity at three radial locations. The analytical solution for a laminar Ekman layer, given by Eq. 2.26 is
presented for the $r^* = 0.3$ case, and is represented by dashed lines. From the rotor side, starting on the right-hand side of the plots, it is evident that the spiral converges more rapidly to its centre when $r^*$ is increased, thus having a more significant departure from the analytical solution. Still, the angles of the spirals are rather close to that of the laminar Ekman layer on the rotor surface, which is of $45^\circ$. One clear feature of the laminar solution is that it predicts a large overshoot of the tangential velocity before converging to the centre, while in the numerical solutions this overshoot is negligible. On the stator side, the angle of the spirals is much smaller than that of the analytical solution in all cases, which is a consequence of the turbulent nature of the stator boundary layer. The shape of the spiral shown in Fig. 4.8(a) can also be obtained from two-dimensional simulations utilising RANS models, as shown, for instance, by Chew [126], who used a mixing length model and obtained profiles similar to those shown in Fig. 4.8(a).

Figures 4.8(b) and (c) show tangential velocity profiles in wall units at the mid-radial position $r^* = 0.5$ near the rotor and the stator, respectively. The wall distance is given by $z^+ = z u_r / \nu$ and the velocity is normalised by $u_r$, where $u_r$ is the friction velocity calculated from the local tangential wall shear stress. In both cases the profiles scale linearly with the wall distance until about $z^+ \approx 5$, thus comprising a viscous sub-layer region. In contrast with turbulent boundary layers of pipe and channel flow (see Fig. 2.3(a)), a buffer layer is not observed, but instead a logarithmic region $(a_1 + a_2 \log z^+)$ appears immediately at the end of the viscous sub-layer. While in a pipe flow the velocity increases monotonically from the wall to the centre of the pipe, for the rotor and stator layers there is a velocity overshoot, and the velocity converges to a uniform value. The profiles suggest that this behaviour pushes the logarithmic layer closer to the wall, in such a way that it is effectively merged with the buffer layer.

4.3 LES results

4.3.1 $Re_\Omega = 4 \times 10^5$

The LES results were obtained using the L1 spectral element mesh shown in Fig. 4.2(c), which has $N_d = 180$, with a polynomial order $P = 7$ and using $N_\theta = 96$, thus giving approximately $8.47 \times 10^5$ degrees of freedom. We refer to this grid using the notation L1-7-96. For $Re_\Omega = 4 \times 10^5$, the time step used is three times larger than that used in the DNS calculation with the D1-8-192 grid, therefore the computational effort required to perform the LES is approximately 3% of that required for the DNS. Using the L2 spectral element mesh shown in Fig. 4.2(d) rather than L1, no difference was observed in the mean profiles, and for this reason only results obtained on the L1 mesh are presented. As noted earlier, the only difference between the L1 and L2 grids is that for the latter the near-wall resolution for the rotor and stator surfaces is the same as in the DNS grids, whereas for the former the first wall-normal elements near each surface are removed. It is therefore interesting that this significant decrease in the near-wall resolution does not affect the quality of the predictions. Additionally, noting that the time-averaged velocity profiles are constant in the cavity core, and that changes in the profiles from the inner towards the outer part of the cavity are not very significant, it is reasonable to relax both the core and wall-tangential resolutions.
As explained in Chapter 3 when using the spectral vanishing viscosity technique to stabilise under-resolved calculations it is necessary to specify the amount of artificial viscosity added, which is controlled by the parameter $\epsilon$, and the cut-off wavenumber $M$ for both the Fourier direction and the polynomial basis. Although there is no universal rule for the choice of these parameters, it is common to use a multiple of $\nu$ for $\epsilon$ and half the number of Fourier modes for $M$ (e.g. Koal et al. [113]). Here, we use $\epsilon = 5\nu$ for both the Fourier and polynomial basis, $M_\theta = 24$ for the cut-off in Fourier space, and $M_P = 4$ for the cut-off in polynomial space.

Time-averaged profiles obtained using the mesh and parameters detailed above are shown in Fig. 4.9 for three radial locations, and are compared with the DNS and experimental results. For the velocity profiles, the level of agreement between the DNS and LES predictions is remarkably good considering the difference in the number of degrees of freedom used to obtain the solutions. In terms of the tangential velocity, the largest difference observed between the solutions is at $r^* = 0.3$, where the LES predicts a core velocity 4% higher than the DNS. The radial velocity profiles also match well, although for $r^* = 0.7$ the LES predictions slightly overestimate the DNS results in the stator region. LES predictions of the velocity fluctuations are

![Figure 4.8](image-url)
also in good agreement with the DNS, although in some cases there are differences in the peak values. At $r^* = 0.3$, for instance, the LES results are in better agreement with the experimental data than the DNS results. This, however, does not imply that the LES results are more accurate, since the DNS results were obtained on a much finer mesh, and are considered to be more reliable. It is important to stress the fact that at $r^* = 0.3$ the flow near the rotor is transitional, thus being inherently sensitive to vibration, which may cause more significant experimental errors than a laminar or fully turbulent flow. Considering that the LES results can be obtained using significantly less computational time, its use is preferred over DNS to obtain accurate predictions for first- and second-order statistics.

The moment coefficient predicted by the LES is $C_m = 2.608 \times 10^{-3}$, or around 10% lower than the DNS values. This indicates a strong dependence of $C_m$ on the resolution, which is likely due to the large element sizes used along the radial direction in the LES grid in comparison with the DNS one. This underprediction, however, has only a minor impact on the velocity and velocity fluctuation profiles, as discussed above.

To investigate the effectiveness of the SVV method in stabilising the LES calculations and improving the solution accuracy, a calculation was run on the L1-7-96 grid without applying the SVV stabilisation. Surprisingly, it was not necessary to reduce the time step to obtain a numerically stable solution. Figure 4.10 shows a comparison between the solutions with and without the SVV stabilisation for $r^* = 0.5$. Note that both approaches constitute a form of implicit LES (iLES), as no sub-grid scale model is used, however for convenience the calculation without SVV stabilisation is referred to as iLES, whereas that with SVV is referred to as LES. The only remarkable difference between the two solutions occurs for the prediction of the azimuthal velocity fluctuation, which is significantly over-predicted when the SVV stabilisation is turned off. This suggests that the SVV stabilisation is able to effectively damp the excessive oscillations of the azimuthal velocity that occur due to the coarse resolution in the core region, while it does not play a major role in the boundary layer regions, where the resolution is sufficiently fine, thus reducing the numerical errors without relying on artificial dissipation. Still, in these well-resolved regions the SVV method does not degrade the quality of the predictions, which indicates that the method is appropriate to stabilise under-resolved calculations.

4.3.2 $Re_\Omega = 10^6$

LES calculations have also been performed for the $Re_\Omega = 10^6$ case on the D1, D2, L1 and L2 grids. Due to the higher Reynolds number, the near-wall resolutions obtained are different in comparison with the $Re_\Omega = 4 \times 10^5$ case, but still $\Delta z^+ < 15$ for the first spectral elements for the D1, D2 and L2 grids, in which case there is one grid point below $z^+ = 1$. For the coarser grid L1, the first elements are within $\Delta z^+ < 35$. As only minor differences were observed between the results obtained on all these grids, we present the averaged profiles obtained on L1-7-96 only, which is the same grid used for $Re_\Omega = 4 \times 10^5$.

As shown in Fig. 4.11, the LES results are in good agreement with the experimental measurements, although differences are observed in the profiles near the stator at $r^* = 0.5$ and $r^* = 0.7$. Additionally, the numerical values of $u_\theta^*$ in the core region are not as close to the experimental data as they were for the lower
Figure 4.9: LES predictions of normalised profiles of tangential and radial velocity and corresponding fluctuations at three radial stations. The DNS (D1-8-192 grid) and LES (L1-7-96 grid) results are represented by black and blue solid lines, respectively, whereas the circles correspond to the experimental data of Séverac et al. [22]. $Re_\Omega = 4 \times 10^5$.

Figure 4.10: Comparisons between LES predictions with (black) and without (blue) SVV stabilisation, for $r^* = 0.5$ and $Re_\Omega = 4 \times 10^5$. 
value of $Re_\Omega$ investigated. In comparison with the $Re_\Omega = 4 \times 10^5$ case, no major differences are observed in the velocity and velocity fluctuation profiles, even though for $Re_\Omega = 10^6$ nearly the entire rotor boundary layer is turbulent. The fact that good predictions were obtained using the L1 mesh suggests that an even coarser mesh could be employed for the simulations with $Re_\Omega = 4 \times 10^6$, especially because it was shown that the wall-normal resolution can be reduced without degrading the accuracy of the predictions. Still, using the L1 grid the predictions can be obtained within a few hours on a small cluster, whereas using the more refined D1 and D2 grids several days are needed to obtain the solutions.

Differences between the results obtained on the different LES grids are more evident when the moment coefficient is considered. For $Re_\Omega = 10^6$, the correlation of Eq. 4.4 gives $C_m = 2.649 \times 10^{-3}$, whereas with the grids D1, D2, L1 and L2 the values are, respectively, $2.274 \times 10^{-3}$, $2.357 \times 10^{-3}$, $1.717 \times 10^{-3}$ and $1.742 \times 10^{-3}$. While the values obtained on the grids D1 and D2 are in reasonable agreement with the correlation, those given by the grids L1 and L2 are considerably underestimated, which is due to the coarse resolution near the walls employed in these grids. Again, it is interesting to note that the quality of the results for the velocities and velocity fluctuations is only weakly affected.

### 4.4 Conclusion

In this chapter the problem of flow in a rotor/stator cavity was investigated using DNS and LES, and the results were compared with experimental data available in the literature. A qualitative analysis of the flow structure revealed that, for $Re_\Omega = 4 \times 10^5$, a significant part of the rotor boundary layer is either laminar or transitional. Analysis of the radial velocity over time at $r^* = 0.7$ and $z^* = 0.01$, i.e. inside the rotor boundary layer, revealed that, although the power spectral density contains high-frequency content, a clear signature of low-frequency motion is still present, which is in contrast with the behaviour observed across the stator boundary layer. For $Re_\Omega = 10^6$, both the rotor and the stator boundary layer are dominated by small-scale structures.

The cost of the LES calculations for $Re_\Omega = 4 \times 10^5$ is only approximately 3% of that of the DNS, and only modest differences are observed between the velocity and velocity fluctuation results, while greater sensitivity was observed for the disk moment coefficient. Using the SVV stabilisation technique, the excessive oscillations of the azimuthal velocity in the core region occurring due to the coarse resolution are damped, thus improving the quality of the predictions in comparison with a case where the SVV is not used. Also, the results reveal that the near-wall resolution can also be reduced in the rotor and stator boundary layers without degrading the solution accuracy.

Finally, we note that the use of a spectral element-Fourier formulation using the SVV technique for numerical stabilisation provides an effective framework for the investigation of enclosed rotor/stator flows in the transitional and turbulent regimes. In comparison with a global spectral method, the spectral element method has the advantage of allowing local mesh refinement in the boundary layers while using a rather coarse resolution in the core region, where gradients are weaker and thus less resolution is required.
Figure 4.11: LES predictions of normalised profiles of tangential and radial velocity and corresponding fluctuations at three radial stations, using the L1-7-96 grid. Comparison between LES results and the experimental data of Sèverac et al. [22]. Re_Ω = 10^6.
Chapter 5

Onset of convection induced by centrifugal buoyancy in a rotating cavity

In this chapter linear stability analysis and DNS are used to investigate the onset of convection in a rotating cavity, where gravitational effects are assumed to be negligible in comparison with the centrifugal acceleration. While in practical applications involving turbomachinery internal air systems the Rayleigh number is typically several orders of magnitude larger than the critical value of $Ra$ for the onset of convection, a study of the linear regime provides information about the instability mechanisms that trigger the transition. At low $Ra$ the buoyant force is too weak to overcome the viscous dissipation, therefore convection does not occur. Above a critical value of $Ra$, convection starts in a relatively well-organised manner, and when $Ra$ is further increased, chaotic motions occur. Here, linear stability analysis is used to calculate the critical Rayleigh number for the onset of convection for different gap sizes, as well as the corresponding critical azimuthal wavenumbers. Using DNS, insight into the non-linear regime is gained, and cases representative of well-organised as well as turbulent convection are considered.

The chapter is organised as follows. The problem is stated in the next subsection, and boundary conditions for the linear analysis as well as for the DNS are presented. Details of the meshes employed are also provided, as well as a grid-independence study. In section 5.2 results of the linear stability analysis are discussed, and in section 5.3 the DNS results are considered. Conclusions are presented in the end of the chapter in section 5.4.

5.1 Problem description

5.1.1 Geometry and boundary conditions

Consider an incompressible fluid bounded by a cylindrical annulus with flat end-walls, rotating about its axis at a constant angular speed $\Omega$, cf. Fig. 5.1(a). The centrifugal force induced by rotation is assumed to be the leading-order body force.

The results included in this chapter are published in [127]:
therefore the effect of gravity is neglected, i.e. we assume that $Fr \gg 1$, where $Fr = \frac{\Omega^2 r}{g}$ is the Froude number. The inner and outer cylindrical walls are kept at constant temperatures, $T_a$ and $T_b$, respectively, where $T_b > T_a$, whilst the two disks are adiabatic. Since the whole system rotates at the same angular speed, the flow is generated purely by buoyancy effects. The physical dimensions of the cavity are chosen to match the experimental set-up employed by Bohn et al. [50], who used $a = 0.125 \text{ m}$, $b = 0.24 \text{ m}$, $d = 0.12 \text{ m}$, and air as the working fluid, giving $Pr \equiv \frac{\nu}{\kappa} = 0.7$, where $Pr$ is the Prandtl number. Although the limited data obtained from the experiments does not allow for a comparison against our results, the choice of this set-up was motivated by its relevance as a typical industrial configuration. The dimensions were kept fixed in the direct numerical simulations, but for the linear analysis of the onset of convection we have varied the radius ratio, $\eta = \frac{a}{b}$, within the interval $0.2 \leq \eta \leq 0.99$. While the analysis is focused on the case with no-slip end-walls, for some cases we include results obtained using periodicity along the axial direction.

In order to be consistent with the nomenclature adopted in the turbomachinery community, the non-dimensional numbers used here are the Rayleigh and Reynolds numbers,

\begin{equation}
Ra = \frac{\Omega^2 r_m \alpha \Delta T (b - a)^3}{\nu \kappa},
\end{equation}

\begin{equation}
Re = \frac{\Omega r_m (b - a)}{\nu},
\end{equation}

where $\alpha$ is the thermal expansion coefficient, $\Delta T = T_b - T_a$ and $r_m = (b + a)/2$. The kinematic viscosity has a constant value in all cases, $\nu = 1.5 \times 10^{-5} \text{ m}^2/\text{s}$. Here we restrict our attention to the cases where the temperature difference is constant, so $\alpha \Delta T$ is kept at a fixed value, $\alpha \Delta T = 0.1$. The Reynolds and Rayleigh numbers then change simultaneously when the angular speed $\Omega$ is varied, and for this reason our results are presented in terms of the Rayleigh number only. For reference, the two cases analysed in more detail are $Ra = 10^5$, giving $Re = 1.5 \times 10^3$ and $Ta = 3.5 \times 10^6$, and $Ra = 10^8$, giving $Re = 4.8 \times 10^4$ and $Ta = 3.5 \times 10^9$, where $Ta$ is the Taylor number defined in Eq. 2.36. Figures 5.1(b) and (c) show examples of instantaneous temperature contours obtained in the non-linear regime for $Ra = 10^5$ and $Ra = 10^8$, respectively, at the mid-axial plane. As discussed in more detail later, for $Ra = 10^5$ the flow is dominated by an initially-perturbed azimuthal mode and its harmonics, while for $Ra = 10^8$ many modes contribute significantly to the total energy.

For the DNS calculations, we solve the Navier–Stokes–Boussinesq equations, which were presented in Chapter 2, Eqs. 2.29. The boundary conditions for the velocities $u = (u_r, u_\theta, u_z)$ and temperature $T$ are,

\begin{equation}
u(a, \theta, z) = u(b, \theta, z) = u(r, \theta, 0) = u(r, \theta, d) = 0,
\end{equation}

\begin{equation}
T(a, \theta, z) = T_a,
\end{equation}

\begin{equation}
T(b, \theta, z) = T_b,
\end{equation}

\begin{equation}
\partial_z T(r, \theta, 0) = \partial_z T(r, \theta, d) = 0.
\end{equation}
Figure 5.1: (a) Geometry of a rotating annulus with end-walls. The entire system rotates at a constant angular speed $\Omega$, the temperatures of the cylindrical surfaces are such that $T_b > T_a$ and the two disks are adiabatic. Instantaneous temperature contours at the mid-axial position in the non-linear regime for (b) $Ra = 10^5$ and (c) $Ra = 10^8$.

For the linear stability analysis, the perturbation equations are solved for a given base flow, and the perturbed variables $u'$ and $T'$ are set to zero on all boundaries.

### 5.1.2 Details of the computations

For the linear stability analysis, a mesh with 50 spectral elements was employed. The polynomial order $P$ of the two-dimensional expansion basis was chosen based on two convergence studies using cases representative of the calculations performed in this work. In the first case the smallest Rayleigh number $Ra_c$ giving an unstable solution was calculated, as well as the critical azimuthal wavenumber $k_c$, for different values of $P$. As illustrated on the left-hand side of Table 5.1, the results are very similar for all values of $P$. Note that, since the domain is cylindrical, only integer values of $k_c$ are considered. The second convergence study was carried out perturbing the base flow at the highest value of the Rayleigh number considered in this work, $Ra = 10^8$, and evaluating the growth rate $\lambda_r$ and the angular frequency $\lambda_i$ of the leading, most unstable eigenvalue. The values obtained from this analysis for different $P$ are shown on the right-hand side of Table 5.1 where no significant differences are observed for $P \geq 7$. Based on this study, we adopted $P = 7$ to carry out all the linear stability analyses reported in this study.

The non-linear analyses are performed using a mesh with 252 quadrilateral spectral elements in the $r - z$ plane. To perturb the motionless basic state, a series of sinusoidal waves with an amplitude of 0.001% of the mean temperature are added to the temperature field. Using this approach, one can follow the evolution of disturbances of a given wavenumber $k$, and analyse how they grow in time and how
Onset of convection

Table 5.1: Effect of the polynomial order on quantities obtained from linear stability analysis. The mesh used has 50 quadrilateral spectral elements.

<table>
<thead>
<tr>
<th>$P$</th>
<th>$k_c$</th>
<th>$Ra_c(k_c)$</th>
<th>$P$</th>
<th>$\lambda_r$</th>
<th>$\lambda_i$</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>5</td>
<td>3040.04</td>
<td>5</td>
<td>9.3056</td>
<td>6.3607 $\times 10^{-5}$</td>
</tr>
<tr>
<td>7</td>
<td>5</td>
<td>3040.06</td>
<td>7</td>
<td>9.2632</td>
<td>1.0743 $\times 10^{-4}$</td>
</tr>
<tr>
<td>9</td>
<td>5</td>
<td>3040.07</td>
<td>9</td>
<td>9.2590</td>
<td>1.1089 $\times 10^{-4}$</td>
</tr>
<tr>
<td>11</td>
<td>5</td>
<td>3040.06</td>
<td>11</td>
<td>9.2597</td>
<td>1.1015 $\times 10^{-4}$</td>
</tr>
</tbody>
</table>

they behave during and after the transition. In Fig. 5.2(a) the temporal evolution of the modal kinetic energy, $E(k)$, given by,

$$E(k) = \frac{1}{2d(b-a)} \int_{\Omega_{rz}} \hat{u}_k \cdot \hat{u}_k^* d\Omega_{rz},$$

(5.5)

of disturbances with azimuthal wavenumbers ranging from 4 to 6 is presented for a mesh with $P = 9$ and $N_\theta = 144, 288$ and 576, for $Ra = 10^8$ (note that the energy corresponding to $k = 0$, i.e., energy of the mean flow, is not shown). In the equation above, $d(b-a)$ is the area of the domain in the $r-z$ plane $\Omega_{rz}$, and $\hat{u}_k$ and $\hat{u}_k^*$ represent the velocity field in Fourier space and its complex conjugate, respectively. As the curves obtained with the three meshes collapse during the initial stage, the figure only shows the behaviour of $E(k)$ after the saturation takes place.

The non-dimensional time used is given by $t^* = \Omega t/2\pi$, so it corresponds to the number of revolutions of the system. For $N_\theta = 144$, the energy of mode 6 starts to decay and is overtaken by mode 5 at $t^* \approx 14$, when the flow becomes turbulent. For the other two values of $N_\theta$, the behaviour of the modes is qualitatively similar, with mode 6 remaining as the most energetic. A plot of the modal kinetic energy versus the azimuthal wavenumber is presented in Fig. 5.2(b) for the three meshes, again excluding the mean flow energy. The energy for each mode was obtained by time-averaging the modal energies in the interval $10 \leq t^* \leq 20$, where the flow is turbulent. The range of resolved scales increases significantly when $N_\theta$ is increased.

Note that in all cases there is a small increase of the energy at high $k$, which is due to aliasing effects. Despite this, the behaviour of the low $k$ modes is not affected by the resolution when $N_\theta$ is increased from 288 to 576, cf. Fig. 5.2(a). For the purposes of this work, we decided to use $N_\theta = 288$ in the calculations, as the behaviour of the disturbances is similar to that observed for $N_\theta = 576$, and also because Fig. 5.2(b) shows that a sufficiently broad range of scales is resolved. Repeating the calculations using $P = 7$ no differences were observed, so $P = 7$ was adopted for the non-linear calculations.
5.2 Linear stability analysis

5.2.1 Onset of convection

To analyse the onset of convection we consider a base flow consisting of solid-body rotation for the velocity components and a pure conduction temperature profile,

\[ u = 0, \]
\[ T(r) = T_a + (T_b - T_a) \frac{\log(r/a)}{\log(b/a)}. \]

This base state is perturbed for different integer azimuthal wavenumbers \( k \), and we look for the critical values of the Rayleigh number, \( Ra_c \), that give a zero growth rate to the leading eigenvalues. This procedure of finding \( Ra_c \) was repeated for 18 values of the radius ratio \( \eta \) within the interval \( 0.2 \leq \eta \leq 0.99 \), keeping the axial length constant.

As the radius ratio \( \eta \) is increased, the critical Rayleigh number for the onset of convection decreases, and the critical azimuthal wavenumber increases, as shown in Fig. 5.3. As expected, in the asymptotic limit \( \eta \to 1 \), the value of \( Ra_c \) approaches that corresponding to natural convection under gravity heated from below, for which \( Ra_c \approx 1708 \), as indicated by a dashed line in Fig. 5.3(a). The calculations also reveal that the conductive state loses its stability to modes formed of pairs of convection cells whose azimuthal length are very close to the gap dimension. Assuming that, at the onset of convection, the length of a cell at \( r = r_m \) is exactly equivalent to the gap width, the critical wavenumber is given by

\[ k_c = \frac{\pi r_m}{b - a} = \frac{\pi}{2} \frac{1 + \eta}{1 - \eta}. \]
This expression was employed by Owen [58] to estimate the number of pairs of cells in a rotating cavity in the non-linear regime. Figure 5.3(b) shows that Eq. 5.7 is an excellent approximation to the critical wavenumber, even for relatively small values of $\eta$. Note that Eq. 5.7 returns non-integer values, and that is the reason why it is slightly off the calculated values of $k_c$ for low $\eta$. The fact that Owen [58] has suggested that Eq. 5.7 is a suitable approximation in the non-linear regime suggests that the basic structure that arises on the onset of convection persists even in the turbulent state, albeit in a less pronounced manner.

![Figure 5.3](image)

Figure 5.3: (a) Critical Rayleigh number $Ra_c$ for the onset of convection as a function of the radius ratio $\eta$. The dashed line corresponds to the critical Rayleigh number for flow between parallel plates heated from below. The extra point at $\eta = 0.52$ corresponds to the geometry considered in more detail throughout the chapter. (b) Critical azimuthal wavenumber $k_c$ versus $\eta$.

We now focus on the case with $a = 0.125$ m and $b = 0.24$ m, or $\eta = 0.52$, as employed in the experiment by Bohn et al. [50]. Figure 5.4(a) shows the critical Rayleigh numbers obtained for 12 azimuthal wavenumbers. From this figure, it can be seen that the lowest value of $Ra_c$ for the onset of convection corresponds to a critical wavenumber $k_c = 5$. The shape of the eigenmode obtained setting $k = 5$ is illustrated in Figs. 5.4(b) and (c), for the temperature and azimuthal velocity component, respectively. These indicate that the flow structure at the onset of convection consists of pairs of counter rotating cells, akin to convection under a gravitational field. Although the mode shapes seem to be invariant along the axial direction, a plot of the amplitude of the temperature perturbation in Fourier space in two dimensions reveals that the modes are tilted near the end-walls of the cavity, as shown in Fig. 5.4(d). This tilt is caused by the presence of the disks with no-slip boundary conditions, which forces the flow to be three-dimensional. For higher Rayleigh numbers, the effect of the disks becomes less significant and confined to thin boundary layers.
5.2.2 Linear growth of unstable solutions

When the Rayleigh number is higher than $Ra_c$, the flow is unstable and each azimuthal wavenumber is associated with a certain growth rate. Figure 5.5 shows how the growth rate $\lambda$ varies with the wavenumber for four values of the Rayleigh number. Although the curves are qualitatively similar, the results indicate that the wavenumber of the most unstable modes increases as $Ra$ is increased. Also, at high $Ra$ the peak of the growth rate constitutes a plateau and decreases at a small rate at high $k$.

Also shown in Fig. 5.5 are the growth rates obtained without the end-walls, i.e., using a periodicity condition along the axial coordinate, as indicated by dashed lines. The results reinforce the fact that the effect of the end-walls becomes less pronounced as $Ra$ increases. When the end-walls are present, in all cases the imaginary part of the eigenvalues is non-zero, albeit small (see Table 5.1). This imaginary part is associated with the drift frequency of the waves, meaning that the convection rolls are moving in the azimuthal direction in the rotating frame. In the range of $Ra$ analysed here, the pattern travels slower than the walls, i.e., in the retrograde direction. Alonso et al. [83] showed that the convection modes are associated with
a non-zero frequency when the flow is three-dimensional, which is the case when end-walls are present. Near the disk boundary layers the Coriolis force is no longer balanced by the azimuthal pressure gradient due to viscous effects, which generates a relative motion between the convection cells and the solid end-walls. Using periodicity along the axial direction, on the other hand, the eigenvalues obtained are real, i.e., the waves do not drift relative to the rotating frame.

Figure 5.5: Growth rate $\lambda_r$ as a function of the azimuthal wavenumber $k$ for $Ra = 10^5$, $10^6$, $10^7$ and $10^8$. The dashed lines indicate solutions obtained without end-walls, i.e., considering periodicity along the axial direction.

As the Rayleigh number is increased, the boundary layers adjacent to the disks become thinner. Figure 5.6 shows how the 99%-thicknesses of the thermal ($\delta_T$) and Ekman ($\delta_E$) boundary layers vary with the Rayleigh number, considering $k = 5$, as well as a power-law fit with 95% confidence intervals for the exponents of the two curves. We note that these thicknesses are normalised with respect to the cavity axial length and that they are evaluated at the mid-radial position. As shown qualitatively in Fig. 5.4(d), the boundary layer thickness varies along the radial direction, specially near the cylindrical walls. In regions away from the mid-radial location, significant departures from the scalings shown in Fig. 5.6 were obtained, presumably because of the greater influence of the cylindrical surfaces. The results show that the Ekman layer scales according to $\delta_E \propto Ra^{-0.255\pm0.022}$, which is consistent with the expected scaling $\delta_E \propto \Omega^{-1/2} \propto Ra^{-1/4}$ for an Ekman layer (recall that the Rayleigh number is evaluated in terms of the centrifugal acceleration $\Omega^2 r_m$). The thermal boundary layer scales as $\delta_T \propto Ra^{-0.208\pm0.039}$, which is in agreement with the scaling $\delta_T \propto Ra^{-1/5}$ obtained from a scale analysis and confirmed experimentally by Rossby [128], for natural convection over a stationary plate with a horizontal temperature gradient. Rossby [128] has shown that this scaling is valid for the plate itself and also for the vertical side-wall used in the experiments, which was adiabatic and therefore equivalent to the disks in the present study. The scaling obtained by Rossby also holds for rotating horizontal convection in some regions of the parameter space (see, e.g., Sheard et al. [129]), but it should be noted that in such problems the body force is acting normal to the disks, and not parallel to them, as is the case here. Finally, we note that the uncertainty of the exponent of $Ra$ for the $\delta_T$ is rather significant, mainly because there might be a change of slope for $Ra > 10^7$. 

91
Figure 5.6: Normalised thicknesses of the thermal ($\delta_T$) and Ekman ($\delta_E$) boundary layers as the Rayleigh number is increased, obtained from linear stability analysis for $k = 5$. The markers indicate computed solutions and the solid lines represent power-law extrapolations.

5.3 Transition to the non-linear regime

Above the critical Rayleigh number $Ra_c$, infinitesimal perturbations grow exponentially in time, up to a point where the energy level saturates and the flow transitions to a different state, which can be either quasi-periodic in time or turbulent, depending on the value of $Ra$. To understand how modes with different azimuthal wavenumbers interact before and during the transition process, we perturb the base flow given by Eq. 5.6 adding small-amplitude sinusoidal perturbations of the form $\sin(k\theta)$ to the temperature field. Figure 5.7 illustrates the behaviour of modal energies when perturbations of wavenumber ranging from one to six and one to four are added to the base flow. When six modes are perturbed, the energies of modes three to six grow exponentially with a constant growth rate until the energy level saturates. As indicated by the markers, the growth rates observed for each mode match those calculated from the linear stability analysis. Modes one and two, on the other hand, experience a sudden change in their growth rates before the transition takes place. To investigate why this happens, a case in which only modes one and two are perturbed was run. For that case, no change in the growth rate was observed before the transition for modes one and two. This indicates that the changes in growth rate arise from the non-linear interactions between the modes. Specifically, mode $k = 1$, for instance, interacts with pairs of modes with wavenumbers $(n, n+1)$, where $n$ is arbitrary, whilst mode $k = 2$ interacts with pairs $(n, n+2)$. These triadic interactions are responsible for the behaviour observed for modes one and two in Fig. 5.7(a). Mode $k = 1$ is mostly affected by modes $k = 5$ and $k = 6$, as these have the greatest energies, and its growth rate at $t^* = 5$ is roughly given by $\lambda_r(k = 6) + \lambda_r(k = 5)$ (this relation is not exact since the growth rate is also affected by interactions between lower-energy modes). Analogously, mode $k = 2$ is mostly affected by modes $k = 6$ and $k = 4$, and that is why its growth rate after $t^* = 5$ is slightly smaller than that for $k = 1$. To illustrate this behaviour further, Fig. 5.7(b) shows the evolution of modal energies when modes one to four
are perturbed. This excludes not only the triad formed by modes $k = 2$, $k = 4$ and $k = 6$ but also the lowest possible triad for $k = 2$, i.e., that with modes $k = 3$ and $k = 5$, while retaining a number of possible triads with $k = 1$. Note that in this case mode $k = 1$ interacts with modes $k = 4$ and $k = 3$ and its growth rate increases, whilst no change is observed for mode $k = 2$. This shows that the energy growth observed for the lower modes comes from the triadic interactions, and not from a resonance involving sub-harmonics. It is important to note that although we have observed a significant effect of triadic interactions during transition, we have not identified triadic resonances. Triadic resonances have been observed in experiments of the baroclinic annulus (e.g. Früh & Read [130], and Früh [70] and references therein) and in other rotating systems, such as precessing cylinder flow (Albrecht et al. [131]).

![Figure 5.7: Behaviour of the modal energies when the solution is initialised adding sinusoidal perturbations of wavenumber (a) one to six and (b) one to four, for $Ra = 10^8$. The solid lines represent the energies obtained from the DNS, and the markers indicate the growth rate obtained from linear stability analysis.](image)

The mode shape associated with each of the perturbed azimuthal modes is similar to that shown in Figs. 5.4(b) and (c), where the number of pairs of cells changes with the mode number. These modes are all associated with modes that have no change of sign in the radial direction, as there is a single cell spanning the radial extent of the cavity. When triadic interactions occur and affect the growth rate of a specific mode, the original mode with a single cell in the radial direction is replaced, as shown in Fig. 5.8 for the case where modes one to six are perturbed. In part (a) of this figure, the shape of mode $k = 1$ at $t^* = 1$ is shown, and part (b) shows the mode shape at $t^* = 5.4$. These mode shapes were obtained from a Fourier decomposition of the temperature field and are evaluated at the mid-axial position. The modified shape has one change of sign along the radial direction, and the position of the convection cells is shifted when compared to Fig. 5.8(a). A similar behaviour is observed for mode $k = 2$, as shown in Figs. 5.8(c) and (d). Both the phase shift and the appearance of an extra cell in the radial direction are due to the form of the advection term of the energy equation, $\mathbf{u} \cdot \nabla T$. Recalling that $\sin(\theta_1)\sin(\theta_2) = \frac{\cos(\theta_1 - \theta_2) - \cos(\theta_1 + \theta_2)}{2}$, we see that when two modes interact they give rise to a new wave which is shifted by 90 degrees relative to
the original waves. Therefore, the phase shift is $\pi/2k$ radians, which is observed in Fig. 5.8. This means that both interacting waves have the same phase, which is expected since their drift velocities are small and they were initialised with the same phase. The appearance of an extra cell in the radial direction comes directly from the multiplication of $u$ by the temperature gradient, as the initial ‘half-wave’ is multiplied by its derivative, which will also be shifted by 90 degrees.

![Temperature contours at the mid-axial position](image)

Figure 5.8: Temperature contours at the mid-axial position for $Ra = 10^8$, (a) $k = 1$ at $t^* = 1$, (b) $k = 1$ at $t^* = 5.4$, (c) $k = 2$ at $t^* = 1$, (d) $k = 2$ at $t^* = 6.5$, (e) $k = 5$ at $t^* = 1$ and (f) $k = 5$ at $t^* = 65$ (turbulent regime). (g) Temperature in Fourier space as a function of the radial position for $k = 5$ at $t^* = 1$, $t^* = 65$ and $t^* = 80$ (the curves are normalised with respect to the maximum values of $\hat{T}$). (h) Temperature in Fourier space for $k = 5$ averaged in the interval $60 \leq t^* \leq 90$.

After the transition for $Ra = 10^8$, all the initially perturbed modes retain a certain magnitude and the modal energies decrease as $k$ increases, as shown in Fig. 5.2(b). Even though a detailed analysis of the turbulent regime is not intended in this work, we have obtained solutions for a few dozen revolutions in the non-linear regime to observe the behaviour of the flow pattern. The calculations revealed that at $t^* \approx 40$ the mode $k = 5$ became the dominant mode, despite the fact that its amplification was initially lower than that of mode $k = 6$, cf. Fig. 5.9(a). As shown by the linear stability analysis, the onset of convection occurs for a critical azimuthal wavenumber $k_c = 5$. Even though the flow is fully turbulent after the transition at $Ra = 10^8$, the structure of mode $k = 5$ is well-defined, consisting of two cells attached to the two cylindrical surfaces and separated by a nearly isothermal core, as shown in Fig. 5.8(f). Figure 5.8(g) shows the temperature profile for mode $k = 5$ in Fourier space as a function of the normalised radial coordinate $r^*$ at three instants. At $t^* = 1$ (Fig. 5.8(e)) the profile consists of a single convection cell, whilst at $t^* = 65$ (Fig. 5.8(f)) there are two cells. At different time instants in the
turbulent regime, the mode shape is qualitatively similar, as can be observed by comparing the profiles for $t^* = 65$ and $t^* = 80$. Figure 5.8(h) shows a time average, still for $k = 5$, obtained within the interval $60 \leq t^* \leq 90$. This mode shape provides a good qualitative representation of the expected behaviour of the temperature in the turbulent regime, where the gradients are confined to thin boundary layers near the cylindrical surfaces, with little activity in the cavity core.

It is instructive to compare the flow behaviour observed in the non-linear regime for $Ra = 10^8$ with a case of lower $Ra$. Figure 5.9(a) shows how the energy of mode $k = 6$ behaves after the transition for $Ra = 10^5$. It is clear from this figure that at this Rayleigh number the flow is quasi-periodic, in contrast with the chaotic behaviour observed for $Ra = 10^8$. Note that Fig. 5.9(a) also shows, for comparison purposes, the corresponding behaviour of modes $k = 5$ and $k = 6$ for $Ra = 10^8$. Another striking difference between the solutions is that for $Ra = 10^5$ modes $k = 1$ to $k = 5$ are damped and the harmonics of the dominant mode $k = 6$ are amplified, reaching significant amplitudes. This behaviour is shown in Fig. 5.9(b), which was obtained from a time-average comprising several revolution periods, and is in contrast with that observed in Fig. 5.2(b) for $Ra = 10^8$ (note that the calculation for $Ra = 10^5$ was run using $N_\theta = 144$). In this case it is clear that the initial forcing affects the observed periodic behaviour. When the flow is turbulent, on the other hand, the modes interact more intensely and effects related to the initial disturbances tend to disappear more quickly over time.

![Figure 5.9](image)

Figure 5.9: (a) Behaviour of the most energetic modes after the transition for $Ra = 10^5$ and $Ra = 10^8$. (b) Energy spectrum for $Ra = 10^5$.

To further illustrate the behaviour of the solutions obtained for $Ra = 10^5$ and $Ra = 10^8$, Fig. 5.10 shows azimuth-time plots of the temperature at the mid-axial and mid-radial position. For $Ra = 10^5$ the flow pattern clearly consists of six cells in the azimuthal direction, which are drifting relative to the rotating frame in a direction opposite to that of the cavity rotation. The quasi-periodic nature of the solution is also revealed by the oscillations occurring over time, which indicate that the convection cells move back and forth along the azimuthal direction. As mentioned, the drift of the cells is caused by the end-walls (disks), which force the flow to be three-dimensional. To verify whether the limit-cycle oscillation observed for $Ra = 10^5$ is affected by the disks, a calculation was run using a periodicity condi-
tion along the axial direction. The corresponding azimuth-time diagram obtained is shown in Fig. 5.10(b), which reveals that no drift occurs, as expected, and also that the azimuthal oscillations of the convection cells are quickly damped after the solution saturates. When $Ra = 10^8$ the pattern is much less clear and is dominated by high-frequency oscillations, and the cells drift at a faster rate when compared to the lower $Ra$ case. Still, the figure reveals that a similar pattern is repeated five times, which is consistent with the fact that $k = 5$ is the dominant mode at $t^* = 100$. For both values of $Ra$, the drift rates obtained from the linear stability analysis are much smaller than those observed in the non-linear regime, which indicates that non-linear effects contribute significantly to the motion of the convection cells relative to the walls.

![Space-time plots of temperature at mid-radius and mid-axial position](image)

Figure 5.10: Space-time plots of temperature at mid-radius and mid-axial position for (a) $Ra = 10^5$ with end-walls; (b) $Ra = 10^5$ without end-walls; (c) $Ra = 10^8$ with end-walls. The temperatures range from $T_a$ to $T_b$.

Figure 5.11 shows that the onset of convection initially follows a similar route for both $Ra = 10^5$ and $Ra = 10^8$. As the flow loses stability, the hot (cold) jet penetrates radially towards the inner (outer) part of the cavity. For $Ra = 10^5$ the solution quickly becomes a limit-cycle and little mixing occurs, whilst for $Ra = 10^8$ the flow becomes turbulent rather abruptly. The initially thin and laminar jets of cold fluid moving outwards lose their stability and become much thicker and chaotic, as shown in Fig. 5.11(i), before a fully turbulent state with strong mixing is attained, cf. Figs. 5.11(k) and (l). Note that in Fig. 5.11(l), which corresponds to $t^* = 60$, $k = 5$ has become the dominant mode. Figures 5.11(e) and (f) correspond to two extrema of the limit cycle observed for $Ra = 10^5$; in the first case the cold jets are close to the hot jets located on their right, and in the other extreme the cold jets have moved leftwards. This oscillatory behaviour is repeated indefinitely after the transition, cf. 5.10(a).

### 5.4 Conclusion

The problem of convection induced by centrifugal buoyancy in a sealed rotating annulus bounded by horizontal disks was investigated numerically using linear stability analysis and direct numerical simulation. From the linear analysis predictions were obtained for the critical azimuthal wavenumber and critical Rayleigh number for different radius ratios. The assumption that the azimuthal length of convection cells at the onset of the instability is equal to the radial gap leads to a simple expression which gives an excellent prediction for the critical wavenumber as a function of
Figure 5.11: Temporal evolution of the temperature field at the mid-axial position for (a)-(f) $Ra = 10^5$ and (g)-(l) $Ra = 10^8$. For $Ra = 10^5$ the snapshots are for $t^* = 8, 9, 10, 12, 24.6$ and $27.7$, respectively, and for $Ra = 10^8$ $t^* = 6, 7, 8.7, 10, 15$ and $60$, respectively, so that the instants before, during and after the saturation are covered.

the radius ratio. Although the disks have only a small effect on the growth rates, specially at large $Ra$, they prevent the flow from being two-dimensional and allow the convection columns to drift, which is associated with the eigenvalues having a non-zero imaginary part. The drift rates observed in the non-linear regime for $Ra = 10^5$ and $Ra = 10^8$ are much larger than those obtained from the linear analysis. The linear analysis also revealed that the scaling obtained for the Ekman layer is consistent with the expected behaviour $\delta_E \propto \Omega^{-1/2}$, and that the thermal boundary layer scales approximately as $\delta_T \propto Ra^{-1/5}$, which could have an impact on disk heat transfer predictions for cases in which the disks are not thermally insulated.

Our non-linear analyses consisted of superposing a number of small-amplitude sinusoidal perturbations to the initially stratified temperature field. While some of the modes corresponding to the perturbed wavenumbers grow exponentially with a constant growth rate up to the point where the energy saturates, lower modes are affected by triadic interactions and are more amplified than the other, non-interacting modes. For $Ra = 10^5$ the solutions obtained in the non-linear regime were shown to be dependent on the initial conditions for the time period considered, as the flow structure is dominated by the initially fastest growing mode and its harmonics. The solution in this case develops limit-cycle oscillations and consists of well-defined convection columns which drift relative to the rotating frame. The columns span the entire axial extent of the cavity, and are slanted near the disks. For $Ra = 10^8$, effects of the initial perturbation disappear over time and the dominant mode is associated with an azimuthal wavenumber consistent with the critical wavenumber obtained from the linear stability analysis. Although this agreement is reasonable, it is necessary to verify whether it holds at different $Ra$ in the turbulent regime and if it persists even at very long integration periods. The fact that the flow is nearly invariant along the axial direction suggests that the use of a two-dimensional model could be appropriate to investigate the flow dynamics in the turbulent regime with a reduced computational cost, although such approach would exclude features such as the drift of the convection columns from the analysis. For $Ra = 10^5$, the calculations revealed that the presence of the disks not only causes the convection
columns to drift, but it also triggers a limit-cycle oscillation, which is not observed in the calculation without the disks.

Read et al. [66] have shown that when the Taylor number is increased progressively while keeping the temperature difference constant, the transition to chaos follows a different route depending on whether the flow is dominated by the baroclinic instability or by convection due to the centrifugal force. While for the baroclinic instability the transition occurs via a sequence of low-dimensional states, for $Fr > 1$ the large-scale motion develops small-scale oscillations when $Ta$ is increased. In this work we have isolated the effect of centrifugal buoyancy by neglecting the gravitational acceleration, and have shown the mechanism for the onset of convection and the existence of solutions dominated by a single azimuthal wavenumber and its harmonics, as well as a solution in which the flow is turbulent but still has a clear signature of a certain wavenumber ($k = 5$). A natural extension of this work will consist on studying the route to chaos via a gradual increase of the centrifugal Rayleigh number, and seeing whether the transition to chaos occurs directly via the excitation of small scales, as suggested by Read et al. [66], or if low-dimensional mechanisms can also occur. Neglecting the effect of gravity is an effective way to study such a transition mechanism, since the flows can be analysed at low Taylor numbers while still neglecting effects arising due to the baroclinic instability.
Chapter 6

Buoyancy-induced flow in a sealed rotating cavity

The study presented in Chapter 5 focused on how the basic state becomes unstable at a certain value of the Rayleigh number and how different azimuthal modes interact during the transition to the non-linear regime. In the present chapter the analysis is extended to study flow statistics in the non-linear regime for three values of the Rayleigh number, $Ra = 10^7$, $10^8$ and $10^9$. While the previous chapter provides information of fundamental interest about the transitional regime, here the focus is on understanding the nature of the thermal and kinetic boundary layers that form near the disks and the shroud. Proper understanding of the disk boundary layer behaviour may allow the development of simplified models to describe buoyancy-induced flows, and a description of the thermal boundary layers is of practical interest since they are related to the wall heat transfer rate. As mentioned in the literature review presented in Chapter 2, previous studies focused on analysing global features of buoyancy-induced flows in rotating cavities, such as instantaneous temperature and velocity fields and shroud Nusselt number. In this study time- and circumferentially-averaged statistics for these flows are reported and insight into the disk and shroud boundary layers is gained.

6.1 Problem description

6.1.1 Geometry and boundary conditions

The geometry adopted is identical to that used in the non-linear calculations reported in Chapter 5 and is shown in Fig. 5.1. The inner and outer radii of the cavity are $a = 0.125$ m and $b = 0.24$ m, and the width is $d = 0.12$ m. These dimensions correspond to those used in the experiments of Bohn et al. [50]. The flow is governed by the Rayleigh and Reynolds numbers given by,

$$Ra = \frac{\Omega^2 r_m \alpha \Delta T (b - a)^3}{\nu \kappa},$$

(6.1)

The results included in this chapter have been accepted for publication in: Pitz, D. B., Chew, J. W. & Marxen, O. (2018). Large-eddy simulation of buoyancy-induced flow in a sealed rotating cavity. ASME Turbo Expo 2018
Figure 6.1: Disk temperature profile used in the calculations where the disks are heated.

\[ Re = \frac{\Omega r_m (b - a)}{\nu}. \]  

(6.2)

In Chapter 5 the value of \( \alpha \Delta T \) was fixed at 0.1, whereas here we assume a fixed relation between \( Ra \) and \( Re \) given by,

\[ Re = 1.441 Ra^{0.557}, \]

as this was used in the experiments of ref. [50]. Using this equation, the value of \( Re \) is calculated from a given \( Ra \), from which the rotational speed \( \Omega \) can be evaluated, and substituting into the equation for \( Ra \) the value of \( \alpha \Delta T \) is obtained.

All walls rotate at a rate \( \Omega \), so that the velocity boundary condition is \( u = (u_r, u_\theta, u_z) = (0, 0, 0) \) on all boundaries. The inner and outer cylindrical surfaces are kept at temperatures \( T_a \) and \( T_b \), i.e. \( T(a, \theta, z) = T_a \) and \( T(b, \theta, z) = T_b \), where \( T_b > T_a \). The disks are either adiabatic, in which case \( \partial_z T(r, \theta, 0) = \partial_z T(r, \theta, d) = 0 \), or have a temperature profile varying from \( T_a \) at \( r = a \) to \( T_b \) at \( r = b \) as shown in Fig. 6.1. This profile was obtained by fitting a third-order polynomial to the experimental data reported by Atkins & Kanjirakkad [93]. Although their experiments included an axial throughflow of cooling air, a similar profile is adopted here so that it is representative of an experimental condition.

The results are presented in terms of non-dimensional quantities. Temperatures and temperature fluctuations are normalised so that \( T^* = (T - T_a)/(T_b - T_a) \), whereas the velocities and corresponding fluctuations are calculated as \( u^*_i = u_i/\Omega r \), unless stated otherwise. The dimensionless radial and axial coordinates are \( r^* = (r - a)/(b - a) \) and \( z^* = z/d \), respectively.

### 6.1.2 Details of the computations

In Rayleigh-Bénard convection, which is fundamentally similar to the problem of buoyancy-induced flow in a rotating cavity, it is common to analyse the flow statistics in terms of fluctuating quantities, since the time-averaged velocity components tend to approach zero for long integration periods. Although it is shown later that a weak mean flow exists near the rotating disks, fluctuating quantities are used to assess grid-independence. For a given variable \( T \), the corresponding time-averaged rms (root-mean-square) quantity is defined as \( T_{rms} = (\overline{T^2} - \overline{T}^2)^{1/2} \). Since in this
Table 6.1: Effect of the polynomial order $P$, number of azimuthal planes $N_{\theta}$ and use of SVV on $T_{\text{core}}^*$, $\delta_{\text{rms}}^*$, $\lambda_{\text{rms}}^*$ and $Nu$ for $Ra = 10^8$ on a mesh with $N_{el} = 160$ spectral elements.

<table>
<thead>
<tr>
<th>$P$</th>
<th>$N_{\theta}$</th>
<th>SVV</th>
<th>$T_{\text{core}}^*$</th>
<th>$\delta_{\text{rms}}^*$</th>
<th>$\lambda_{\text{rms}}^*$</th>
<th>$Nu$</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>128</td>
<td>no</td>
<td>0.6387</td>
<td>0.01459</td>
<td>0.01802</td>
<td>25.63</td>
</tr>
<tr>
<td>7</td>
<td>128</td>
<td>no</td>
<td>0.6430</td>
<td>0.01451</td>
<td>0.01732</td>
<td>26.25</td>
</tr>
<tr>
<td>9</td>
<td>128</td>
<td>no</td>
<td>0.6413</td>
<td>0.01451</td>
<td>0.01723</td>
<td>25.96</td>
</tr>
<tr>
<td>7</td>
<td>256</td>
<td>no</td>
<td>0.6590</td>
<td>0.01459</td>
<td>0.01984</td>
<td>24.58</td>
</tr>
<tr>
<td>7</td>
<td>576</td>
<td>no</td>
<td>0.6561</td>
<td>0.01476</td>
<td>0.02063</td>
<td>24.60</td>
</tr>
<tr>
<td>7</td>
<td>128</td>
<td>yes</td>
<td>0.6419</td>
<td>0.01459</td>
<td>0.01967</td>
<td>25.08</td>
</tr>
</tbody>
</table>

study the main interest is to understand the nature of the disk and shroud boundary layers, grid-independence is carried out evaluating the thickness of these layers on different grids. Following the criterion described by Ahlers et al. [132], the thermal and kinetic boundary layer thicknesses are defined as the distance from the wall where the corresponding $rms$ values reach their maxima. Additionally, the value of the core temperature $T_{\text{core}}$ is also evaluated.

Considering the case $Ra = 10^8$ as a baseline, a mesh with 160 spectral elements was designed, and it has been verified a posteriori that there were 10 degrees of freedom to resolve the shroud thermal boundary layer and 10 degrees of freedom within the disk kinetic boundary layer when a polynomial of order $P = 7$ was used. Other researchers, e.g. Kunnen et al. [133] for rotating RB convection, have used meshes with similar resolution in the boundary layer region. Table 6.1 shows the influence of the polynomial order $P$, of the number of azimuthal planes $N_{\theta}$, and of the use of SVV stabilisation on the normalised core temperature $T_{\text{core}}^*$, on the normalised disk kinetic boundary layer thickness $\delta_{\text{rms}}^*$ based on $u_{\text{rms}}^*$, on the normalised shroud thermal boundary layer thickness $\lambda_{\text{rms}}^*$ based on $T_{\text{rms}}^*$, and on the shroud Nusselt number $Nu$. The values of $\delta_{\text{rms}}^*$ and $\lambda_{\text{rms}}^*$ were normalised with respect to the disk spacing $d$ and the radial gap $b - a$, respectively. The results show that the polynomial order has only a minor effect on the results, which indicates that the resolution used in the $r - z$ plane is adequate. Changing $N_{\theta}$ while keeping the polynomial order constant, greater sensitivity is observed, although the differences are still small considering the large increase in the values of $N_{\theta}$. It should be noted that a full $360^\circ$ annulus is considered, since large-scale structures occur, thus requiring a large number of Fourier planes along $\theta$.

The numerical values shown in Table 6.1 reveal that the shroud thermal boundary layer thickness is the most sensitive of the three parameters analysed. It is interesting to note that the use of SVV with $N_{\theta} = 128$ improves the accuracy of $\lambda_{\text{rms}}^*$ relative to the reference value obtained with $N_{\theta} = 576$ in comparison with the case without SVV. A similar effect is observed for the shroud Nusselt number. The SVV technique was used considering an augmented viscosity amplitude of $\epsilon = 5\nu$ for both the spectral element and Fourier discretisations, whereas the cut-off wavenumbers were $M_P = 4$ and $M_\theta = 32$, respectively. While a study of the sensitivity of the results on the SVV parameters is out of the scope of this work, these values were chosen based on recommendations from previous investigations, e.g. Koal et al. [113].

101
For the purposes of this work, the grid with \( P = 7 \) and \( N_\theta = 128 \) using the SVV stabilisation is considered sufficiently accurate to obtain the results for \( Ra = 10^8 \). The same grid is employed for \( Ra = 10^7 \), whereas for \( Ra = 10^9 \) a grid with \( N_{el} = 216 \) and \( N_\theta = 288 \) was used, containing more elements in the near-wall region so as to capture the thinner boundary layers. All the results reported were obtained by time-averaging the solution for at least 50 disk revolutions after a statistically steady state was reached. Time-convergence of the profiles reported throughout the chapter was assessed by analysing the time-averaged results every approximately 10 disk revolutions. The profiles were considered to be converged if changes in the order of a few percent or less were observed between two averages collected 10 revolutions apart.

### 6.2 Results and discussion

Before proceeding to the analysis of time-averaged flow statistics, instantaneous contours of normalised temperature and radial and tangential velocities are shown in Fig. 6.2 for \( Ra = 10^8 \) with adiabatic disks, at the mid-axial position. Note that the velocities are normalised with respect to the local disk speed \( \Omega r \), and that the tangential velocity is measured in a rotating reference frame. The contours show that the flow structure is consistent with Rayleigh-Bénard convection, where pairs of counter-rotating cells dominate, as evidenced by the velocity contours. Figure 6.2a shows that the temperature field is fairly uniform throughout the cavity core, with steep gradients confined to the shaft and shroud boundary layer regions. When the results are averaged over time and \( \theta \), a core temperature, reported in Table 6.1, exists and spans most of the radial extent of the cavity. The core temperature is evaluated by averaging the temperature within the interval \( 0.25 \leq z^* \leq 0.75 \), and also along the radial direction in the region outside the boundary layers, where the temperature is nearly uniform.

![Instantaneous contours of normalised temperature, radial velocity, and tangential velocity](image.png)

Figure 6.2: Instantaneous contours of normalised (a) temperature, (b) radial velocity and (c) tangential velocity, at the mid-axial position for \( Ra = 10^8 \) with adiabatic disks. The cavity rotates in the counter-clockwise direction.
6.2.1 Disk boundary layers

Time-averaged behaviour

To elucidate the nature of the kinetic boundary layers forming near the disks, profiles of the mean radial velocity $\overline{u^*_r}$ and its fluctuation $u^*_{r,rms}$ are shown in Fig. 6.3 for the case $Ra = 10^8$ with adiabatic disks. All the mean values reported were obtained by averaging over time and along $\theta$. The values are normalised with the local disk speed $\Omega r$. The first thing to be noted is that, as the radius increases, the peak radial velocity decreases, eventually becoming negative, i.e. with fluid moving radially inwards, near $r^* = 0.75$.

Note that outside the boundary layer the radial velocity goes to zero, as in an Ekman layer for radial inflow or outflow without buoyancy. It should be kept in mind that, instantaneously, the flow is formed of a certain number of pairs of counter-rotating convection cells spanning the azimuthal direction, thus having alternating positive and negative radial velocity components. When the velocity is averaged over time and $\theta$, as in Fig. 6.3, the radial velocity goes to zero, except near the disk boundary layers. Still, even the peak values of $\overline{u^*_r}$ are an order of magnitude smaller than the fluctuations and are below 1% of the local disk speed. The unsteady nature of the boundary layer also makes it difficult to define its thickness. For a laminar Ekman layer, the thickness scales as $\sqrt{\nu/\Omega}$, the so-called Ekman depth, and it is possible to define a distance $\delta_0$ from the disk at which the radial velocity first crosses the origin, given by $\delta_0 = \pi \sqrt{\nu/\Omega}$ (see refs. [124, 3]). This value (normalised by $d$) is indicated by a dashed line in the plots of Fig. 6.3. Note that this analytical solution gives a good prediction for the boundary layer thickness in regions away from where the velocity reversal occurs.

The bottom row of Fig. 6.3 shows normalised radial velocity fluctuation profiles, which reach much higher values than the mean radial velocity, thus indicating that this is the preferred quantity to study the boundary layer behaviour. Note, also, that the fluctuations do not go to zero outside the boundary layer, due to the fluctuating inward and outward movement induced by convection. Since the $rms$ fluctuation only takes positive values, it provides a consistent way to define a boundary layer thickness, in terms of the distance $\delta^*_{rms}$ from the disk where $u^*_{r,rms}$ reaches its maximum.

Figures 6.4(a), (b) and (c) show how the boundary layer thickness varies along the radial direction for $Ra = 10^7$, $Ra = 10^8$ and $Ra = 10^9$, with adiabatic disks. The two horizontal lines included in the plots correspond to the Ekman depth $\sqrt{\nu/\Omega}$ and to the thickness of a laminar Ekman layer, $\pi \sqrt{\nu/\Omega}$, both normalised by the cavity width $d$. Note that we consider two definitions for the boundary layer thickness. The first, denoted by $\delta^*_0$, is defined as the distance from the disk where $\overline{u^*_r}$ crosses the origin for the first time, so that it corresponds to the definition employed for a laminar Ekman layer. The second definition, denoted by $\delta^*_{rms}$, corresponds to the point of maximum radial velocity fluctuation. As expected from inspection of Fig. 6.3, the value of $\delta^*_0$ increases radially outward until a discontinuity occurs due to the radial velocity reversal. The thickness $\delta^*_{rms}$, on the other hand, shows an uniform behaviour along most of the radial extent of the disk and is not affected by the mean velocity reversal. For $r^* < 0.2$ and $r^* > 0.8$, departures from the uniform value are observed due to the influence of the cylindrical surfaces. It is important to note the similitude between the profiles obtained for the different values of $Ra$. 
Figure 6.3: Near-disk profiles of normalised mean radial velocity $u^*_r$ and radial velocity fluctuation $u^*_{r,rms}$ at different radial locations $r^*$, for $Ra = 10^8$ and adiabatic disks. The dashed lines correspond to the thickness of a laminar Ekman layer, $(\pi/d)\sqrt{\nu/\Omega}$. (note the difference in the ranges of the vertical axes). In all cases the values of $\delta^*_{rms}$ are located between $(1/d)\sqrt{\nu/\Omega}$ and $(\pi/d)\sqrt{\nu/\Omega}$, which indicates that the disk boundary layers scale as an Ekman layer, even though they are unsteady.

To further confirm the conclusion above, Fig. 6.4(d) shows that $\delta^*_{rms}$ scales as $\Omega^{-0.517}$, or equivalently as $Re^{-0.517}$, where the exponent is close to that for an Ekman layer, $-0.5$. Note that the three numerical values represented by circles in the figure correspond to the cases $Ra = 10^7$, $Ra = 10^8$ and $Ra = 10^9$. The values of $\delta^*_{rms}$ were obtained by averaging over the interval $0.25 \leq r^* \leq 0.75$, where only minor variations occur. While it would be desirable to consider intermediate as well as lower and higher values of $Ra$, the results presented here do indicate that a substantial part of the disk boundary layer behaves as an Ekman layer. Still, it should be kept in mind that the boundary layers are unsteady and that the results are not valid very close to the shaft and shroud regions.

Figure 6.5 shows profiles of $u^*_{r,rms}$ at a fixed radial location $r^* = 0.5$ for the three values of $Ra$ considered. Obviously, as $Ra$ increases steeper gradients occur near the disk, as shown in Fig. 6.3(a). Both the peak and core (i.e., outside the boundary layer) normalised values decrease as $Ra$ is increased. This may be due to the fact that greater mixing occurs at high $Ra$, which implies a more uniform flow structure, or also because of the stabilising effect of the Coriolis force, which becomes stronger the larger the value of $Ra$. Close similarity between the profiles is observed when they are normalised by their maximum values and the axial coordinate is divided by the thickness $\delta_{rms}$, as shown in Fig. 6.5(b). The three lines are nearly indistinguishable, which indicates a strong scaling of the entire profile with respect to the boundary layer thickness. Additionally, these results show that the spatial resolution used for the highest $Ra$ case is adequate to resolve the disk boundary layer region.
When the disks are heated rather than adiabatic, a thermal boundary layer develops, but the behaviour of the kinetic boundary layer remains essentially the same. Figure 6.6(a) shows a comparison of $u_{r,rms}^*$ for adiabatic and heated disks, for $Ra = 10^8$ at $r^* = 0.5$. Although no changes are observed in the boundary layer thickness, $u_{r,rms}^*$ reaches higher values for the heated disks case. This trend has been verified to hold across different radial positions. Also, heating the disks promotes a significant increase in the near-wall temperature fluctuation $T_{rms}^*$, as shown in Fig. 6.6(b). This is expected if we consider that the temperature imposed on the disks breaks the quasi-homogeneous nature of the temperature distribution along the axial direction, thus causing temperature oscillations near the disk in order to satisfy the boundary condition. When the disks are adiabatic, the imposition of a zero temperature gradient merely reinforces the homogeneity along $z$, therefore the fluctuations observed are weaker than for the heated disks case.
Figures 6.5(a) and (b) show profiles of near-disk profiles of normalised radial velocity fluctuations $u_{r,rms}^*$ at $r^* = 0.5$, for the case of adiabatic disks at different $Ra$; (b) radial velocity fluctuation normalised by its maximum value versus $z/\delta_{rms}$.

As mentioned earlier, when the disks are heated the behaviour of the kinetic boundary layer is nearly unaffected, as it is clear comparing Figs. 6.4(d) and 6.7(c). The thermal boundary layer based on $T_{rms}$ is thicker than the kinetic one for all $Ra$, and a similar scaling with $Re$ is observed, $\delta_{T,rms}^* \propto Re^{-0.451}$.

**Instantaneous behaviour – adiabatic disks**

Due to the presence of counter-rotating convection cells along the azimuthal direction, instantaneously the radial velocity does not approach zero outside the disk boundary layers. Thus, a standard laminar Ekman solution for non-buoyant radial inflow or outflow is not valid. In that case the far-field tangential velocity is a constant, $u_{\theta,core}$, and the far-field radial velocity $u_{r,core}$ is zero. If, however, a non-zero far-field radial velocity is assumed, the Ekman solution can also be obtained in the following form,

$$u_{r,Ek} = u_{r,core} \left(1 - e^{-z/\delta} \cos \frac{z}{\delta}\right) - u_{\theta,core} e^{-z/\delta} \sin \frac{z}{\delta},$$

$$u_{\theta,Ek} = u_{\theta,core} \left(1 - e^{-z/\delta} \cos \frac{z}{\delta}\right) + u_{r,core} e^{-z/\delta} \sin \frac{z}{\delta},$$

where $\delta = \sqrt{\nu/\Omega}$ is the Ekman depth. Figure 6.8 shows normalised instantaneous
profiles of the radial and tangential velocities at different azimuthal locations for $r^* = 0.5$, $Ra = 10^8$ with adiabatic disks, as well as the corresponding analytical solutions given by Eqs. 6.3 and 6.4. Outside the boundary layers, $u^*_r$ and $u^*_\theta$ do not reach constant values, therefore we considered an average obtained over $0.1 \leq z^* \leq 0.2$ to evaluate $u_{r,core}$ and $u_{\theta,core}$. In general, the analytical solution is in remarkably good agreement with the numerical values. While the good agreement between the numerical and analytical profiles outside the boundary layers comes from the use of a core velocity, the most interesting feature of these results is the agreement between the numerical and theoretical boundary layer thickness and the peak velocities. This indicates that, despite the fact that the flow is unsteady and dominated by buoyancy effects, even instantaneously the disk boundary layers behave as laminar Ekman layers with a non-zero far-field radial velocity. It is important to note that the $\theta$ coordinates indicated in Fig. 6.8 are arbitrary, since the flow pattern drifts relative to the rotating frame. The positions shown were chosen so as to illustrate regions of high positive or negative radial velocity as well as regions of small velocity, where a boundary layer is less evident.

Since the Ekman solution neglects inertial effects, the agreement of the numerical results with this solution indicates that the response time for the disk boundary layers is much faster than the timescale for changes in the freestream velocity.

**Instantaneous behaviour – heated disks**

The analysis of the previous subsection revealed that the disk kinetic boundary layers behave similarly to laminar Ekman layers, and that the velocity gradients are mostly confined to these boundary layers. In this subsection instantaneous temperature profiles near the disks are analysed for different $Ra$ in a range of azimuthal locations, for the case of heated disks, as shown in Fig. 6.9. The vertical dashed lines indicated in the plots correspond to the thickness of the thermal boundary layer based on $T_{rms}$ given in Fig. 6.7(c). The behaviour of the temperature near the disk is less consistent across different $\theta$ in comparison with the velocity profiles, in the sense that at some locations the temperature gradient is confined to the bound-

Figure 6.6: Near-disk profiles of normalised (a) radial velocity fluctuation $u^*_{r, rms}$ and (b) temperature fluctuation $T^*_{rms}$ at $r^* = 0.5$, for $Ra = 10^8$. 

![Graph of radial velocity fluctuation](image1)

![Graph of temperature fluctuation](image2)
Figure 6.7: Averaged near-disk profiles of (a) temperature and (b) temperature fluctuation for different $Ra$ for the case of heated disks, at $r^* = 0.5$. (c) Thermal and kinetic boundary layer thickness evaluated over $0.25 \leq r^* \leq 0.75$.

ary layer, e.g. $\theta = 1.26$ for $Ra = 10^8$, whilst at others the temperature gradient is more gradual, e.g. $\theta = 1.51$ for $Ra = 10^8$. This indicates that the disk thermal boundary layer is characterised by a range of time and length scales, associated with convection induced by centrifugal buoyancy as well as by conduction effects. It is important to note that for $Ra = 10^9$ (Fig. 6.9(c)) the temperature changes near the disk are more abrupt, which suggests that at higher $Ra$ the effect of convection dominates the dynamics. At engine conditions, characterised by much higher Rayleigh numbers, this behaviour is expected to be even more pronounced.

### 6.2.2 Shroud boundary layer

Near the shroud, thermal and kinetic boundary layers are analysed considering fluctuating quantities. Figure 6.10 shows profiles of the radial and tangential velocity fluctuations normalised by the tangential speed at $r^* = 0$, denoted by $u_{r,rms}$ and
CHAPTER 6

$\theta = 0$ rad $\theta = 0.126$ rad $\theta = 0.628$ rad $\theta = 0.754$ rad $\theta = 0.880$ rad

Figure 6.8: Instantaneous profiles of $u^*_r$ and $u^*_\theta$ at different azimuthal locations for $r^* = 0.5$, $Ra = 10^8$ with adiabatic disks. The profiles are compared with Ekman-type solutions given by Eqs. 6.3 and 6.4

$u^*_{r,rms}$, respectively. As the plots are presented varying the radial coordinate, the velocity fluctuations were normalised with a constant velocity $\Omega a$, rather than with the local disk speed $\Omega r$, so as not to distort the profiles. The profiles of $u^*_{r,rms}$ reveal that the fluctuations reach a maximum in the cavity core, while two peaks occur for $u^*_{\theta,rms}$. This is consistent with the fact the flow consists of pairs of counter-rotating cells, such that fluid ejected from the shaft, for instance, flows outwards and decelerates when approaching the shroud, where it is deflected in the azimuthal direction. An analogous phenomenon occurs if we consider fluid moving inwards coming from the shroud. This behaviour is illustrated clearly in the contours shown in Fig. 6.2.

Although the author is not aware of any experimental or numerical study where statistics were reported for buoyant flow in a rotating cavity, the profiles are consistent with the numerical results reported by Kunnen et al. [133] for RB convection without rotation. For the radial velocity fluctuation it is rather clear that increasing the Rayleigh number tends to decrease the magnitude of the fluctuations relative to the rotational speed. For $u^*_{\theta,rms}$ this trend is also clear for $Ra = 10^9$, but $Ra = 10^7$ and $Ra = 10^8$ the maximum values are very similar.

Figure 6.11 shows profiles of $T^*$ and $T^*_{rms}$ for the three values of $Ra$ considered. Note that in the bottom row of the figure the boundary layer region is shown in detail. In all cases a uniform core is observed between the shaft and shroud boundary layers, where steep temperature gradients occur. As shown later, the core temperature can be used to estimate the shroud Nusselt number employing correlations for natural convection over a heated horizontal flat plate. Figure 6.11(a) reveals that the core temperature is only slightly affected when $Ra$ is varied, at least within the range of $Ra$ considered. If it is assumed that the heat transfer from both the shroud and shaft are proportional to $Ra^\alpha$, where $Ra'$ is a modified Rayleigh number based on the core temperature defined in Eq. 6.8 which will be explained in detail later, then a theoretical core temperature $T_{core,t}$ can be calculated assuming that $Nu \propto Ra^\alpha$ on both surfaces. With this assumption, $T_{core,t}$ is given by,
Figure 6.9: Instantaneous temperature profiles at $r^* = 0.5$ for the case of heated disks. The dashed lines represent the thermal boundary layer thickness based on the maximum value of $T_{rms}$. 
Figure 6.10: Profiles of (a) $u_{r,rms}^*$ and (b) $u_{\theta,rms}^*$ as a function of the radial coordinate $r^*$, for $Ra = 10^7$ (solid line), $Ra = 10^8$ (dashed line) and $Ra = 10^9$ (dash-dot line), with adiabatic disks. The profiles were obtained by averaging over $0.25 \leq z^* \leq 0.75$. The superscript $^*$ indicates that the fluctuations are normalised by $\Omega a$.

\[ T_{\text{core},t} = \frac{bT_b + aT_a}{b + a}, \]  

which corresponds to $T_{\text{core},t}^* = 0.6575$ using the parameters adopted in this study. Figure 6.11(a) reveals that $T_{\text{core},t}$ provides a good approximation to the observed core temperature, especially at the highest value of $Ra$ considered.

Note that the $T_{rms}^*$ profiles shown in Fig. 6.11(b) are not symmetric with respect to the radial position. For RB convection under gravity, this profile is expected to be symmetric, as the averaged core temperature approaches $T^* = 0.5$, and additionally because the gravitational acceleration is uniform across the vertical position. Here, on the other hand, the core temperature is considerably larger than $T^* = 0.5$, therefore stronger fluctuations are expected near the shaft region than near the shroud, as shown in Fig. 6.11(b). Again, increasing the Rayleigh number has the effect of suppressing the fluctuations in the core region. In the results of Fig. 6.11, the value of $T_{rms}^*$ at $r^* = 0.5$ scales as $Ra^{-0.123}$, which is close to the scaling $Ra^{-1/7}$ observed for the RB problem. This scaling was obtained, for instance, by Verzicco & Camussi [134] for a cylinder heated from below using DNS, and experimentally by Niemela et al. [57]. Thus, the results suggest that the suppression of temperature fluctuations with increasing $Ra$ is not caused by the stabilising effect of the Coriolis force, but instead by the fact that the temperature field becomes more uniform the higher the value of $Ra$; this is consistent with RB convection. Parts (c) and (d) of Fig. 6.11 show the near-shroud region, where a decrease in the boundary layer thickness for larger $Ra$ is evident. The boundary layer thickness used in this work, denoted by $\lambda_{rms}$, is based on the distance from the shroud where the peak values of $T_{rms}$ occur.

To verify whether the shroud thermal boundary layer has a universal character, $T_{rms}$ profiles normalised by the peak value were analysed – see Fig. 6.12(a) – using $(b-r)/\lambda_{rms}$ as the horizontal axis, which corresponds to the distance from the shroud
normalised by the corresponding boundary layer thickness. The figure shows a close overlap between the three curves, although not as evident as that observed for the disk kinetic boundary layer, cf. Fig. 6.5(b). The scaling of $\lambda_{rms}$ with $Ra$ is shown in Fig. 6.12(b). $\lambda_{rms}$ is related to the shroud heat transfer, and as mentioned in the literature review, previous studies suggest conflicting results for the Nusselt number scaling with $Ra$. In Fig. 6.12 it is shown that $\lambda_{rms}$ scales according to $\lambda_{rms} \propto Ra^{-0.321}$, which is consistent with $Nu \propto Ra^{1/3}$ correlations valid for natural convection under gravity at large Rayleigh numbers. While performing calculations at different values of $Ra$ would be important to extend the validity of this conclusion, the present results indicate that the shroud thermal boundary layer scales as a heated plate in high-Rayleigh number RB convection. It should be noted that our results show that when $Ra$ is increased the normalised fluctuations decrease (e.g. Figs. 6.10).
(a) Temperature fluctuation normalised by its maximum value.

Figure 6.12: (a) Temperature fluctuation normalised by its maximum value as a function of the distance from the shroud \((b-r)/\lambda_{rms}\) normalised by the thermal boundary layer thickness \(\lambda_{rms}\), for \(Ra = 10^7\) (solid line), \(Ra = 10^8\) (dashed line) and \(Ra = 10^9\) (dash-dot line); (b) \(\lambda_{rms}^*\) versus \(Ra\), showing a scaling of \(\lambda_{rms}^* \propto Ra^{-0.321}\).

and \(6.11\), which is due to greater uniformity of the flow structure at large \(Ra\). This, however, does not have an impact on the scaling of the shroud thermal boundary layer thickness with \(Ra\) in comparison with convection under gravity.

6.2.3 Heat transfer

Adiabatic disks

In this section the shroud heat transfer, quantified in terms of the Nusselt number \(Nu\), is compared with existing correlations from the literature. The Nusselt number can be defined as the ratio of convective over conductive heat transfer rate,

\[
Nu = \frac{\dot{q}_{conv}}{\dot{q}_{cond}},
\]

where \(\dot{q}_{conv}\) is calculated from the time- and circumferentially-averaged temperature gradient on the shroud, and \(\dot{q}_{cond}\) is obtained from a pure conductive temperature profile. Thus, for the basic, conductive state, \(Nu = 1\), whereas \(Nu > 1\) when convection occurs. Figure 6.13(a) shows the results obtained numerically for the three values of the Rayleigh number, as well as predictions from different correlations. The numerical values scale as \(Nu \propto Ra^{0.308}\), which is consistent with the scaling obtained for the shroud thermal boundary layer thickness.

The blue solid line in Fig. 6.13(a) shows values from the correlation obtained experimentally by Bohn et al. \[50\] for a sealed rotating annulus, and it is noted that they are considerably lower than the numerical values. In a later numerical study, Bohn & Gier \[53\] used a correction to the values of \(Nu\) obtained in one of their experiments reported in ref. \[50\], which resulted in higher values of \(Nu\) at high \(Ra\), as shown by the blue dotted line in Fig. 6.13(a). Still, the values of \(Nu\)
are much lower than the numerical ones obtained in the present study, although it should be noted that for the corrected correlation the slope of $Nu$ with $Ra$ is closer to the numerical trend. The authors of [50, 53] applied a correction since the side-walls (disks) used in the experiments were not perfectly adiabatic, thus causing an increase in the heat transfer from the cylindrical surfaces.

The two-dimensional calculations performed by King et al. [55] also revealed that the Nusselt number was much higher than the experimental correlation of Bohn et al. [50]. The values of $Nu$ reported in ref. [55] are in good agreement with the numerical results presented here. Sun et al. [54], who used a compressible flow solver to investigate the problem, obtained good agreement with the correlation of ref. [51] (without applying the correction for $Nu$), which could indicate that compressible effects may play a role in the heat transfer rate. This, however, seems unlikely, since the velocities and temperature differences involved are sufficiently small so that the flow can be assumed to be incompressible and the Boussinesq approximation is applicable.

The black dashed line of Fig. 6.13(a) represents a correlation proposed by Hollands et al. [56] for natural convection in a fluid layer heated from below and cooled from above. The present numerical results are in much better agreement with this correlation than with those of Bohn et al. [50, 53]. Considering the nature of the shroud boundary layer discussed in the previous section, the present results suggest that the heat transfer from the cylindrical surfaces is analogous to that from plates in a Rayleigh-Bénard system, such that the action of the Coriolis force does not cause a strong reduction in the heat transfer. One key issue that could be investigated in future studies is whether increasing the Rayleigh number towards engine-representative conditions, i.e. in the order of $Ra = 10^{13}$, would result in a change of the scaling observed here.

For engineering purposes, it is interesting to be able to estimate the Nusselt number using simple correlations. One possibility explored here is to assume that the heat transfer from the shroud to the cavity core is analogous to the heat transfer from a heated horizontal flat plate to a colder environment. To do this, it is necessary to use a core temperature for the cavity flow, which corresponds to the temperature of the cold environment in the heated flat plate case, as well as a local centrifugal acceleration, analogous to the acceleration of gravity. Thus, modified Nusselt $Nu'$ and Rayleigh $Ra'$ numbers are defined, for the shroud surface, as,

$$Nu' = \frac{q_{conv}(d/2)}{k(T_b - T_{core})}$$

and

$$Ra' = \frac{\Omega^2 b \alpha (T_b - T_{core})(d/2)^3}{\nu \kappa}$$

where $q_{conv}$ is the heat flux calculated on the shroud surface and $k$ is the thermal conductivity. The equations above can be adapted for the shaft surface by replacing $b$ by $a$, $T_b$ by $T_{core}$, $T_a$ by $T_{core} - T_a$, and then evaluating $q_{conv}$ on the shaft surface. For flat plate correlations, the reference length is defined as the area of the plate divided by its perimeter, which in the present case is approximated as $d/2$. Note that $\Omega^2 b$ represents the local centrifugal acceleration on the shroud. Figures 6.13(b) and (c) show the values obtained for $Nu'$ as well as predictions from the correlations obtained by Lloyd & Moran [49] for low and high $Ra'$, which scale, respectively, as $Nu' \propto Ra'^{1/4}$ and $Nu' \propto Ra'^{1/3}$. The results suggest that the correlations are able to
provide a good estimate for $Nu'$ for the rotating annulus, although, again, it would be interesting to test this assumption for higher values of the Rayleigh number.

**Heated disks**

In the case of adiabatic disks, the heat transfer rate from the shroud to the fluid is balanced by the heat transfer from the fluid to the shaft. When the disks are heated, in contrast, they also exchange heat with the fluid domain. In order to measure the relative importance of the disk heat transfer in comparison with the shroud heat transfer, the total heat transfer rate $\dot{q}$ divided by the thermal conductivity $k$ is plotted for these surfaces in Fig. 6.14(a). The values of $\dot{q}/k$ were obtained by
integrating the wall heat fluxes along the corresponding areas (disk and shroud). An overall energy balance was performed and the error was verified to be less than 1% of the shroud heat transfer rate in all cases. Although not shown in Fig. 6.14(a), the overall disk heat transfer is negative, i.e. from the fluid to the disk. This is so because of the temperature profile employed, which goes from $T_a$ to $T_b$ so as to avoid a discontinuity at $r^* = 0$. For all $Ra$ the shroud heat transfer is more than ten times greater than the disk heat transfer, but the scaling of both quantities with $Ra$ is similar.

Figure 6.14(b) shows the disk Nusselt number for the three values of $Ra$, where $Nu_d$ is defined as,

$$Nu_d = \frac{q_r}{k(T_{disk} - T_a)},$$

where $q_r$ is the local heat flux and $T_{disk}$ is the local disk temperature. It would also be possible to define the reference temperature difference as $T_{disk} - T_{core}$, however in that case a discontinuity where $Nu_d$ would tend to infinity would be observed where $T_{disk}$ was close to $T_{core}$. For this reason $T_{disk} - T_a$ is used in the present definition, so that $Nu_d$ tends to infinity near $r^* = 0$. The figure shows that $Nu_d$ is negative for a significant portion of the disk, since in this region the disk temperature is smaller than $T_{core}$.

### 6.3 Conclusion

The problem of buoyancy-induced flow in a sealed rotating annulus contains the basic ingredients encountered in the rotating cavities of turbomachinery internal air systems. Using a sealed cavity rather than a configuration with an axial throughflow of cooling air enables the effect of centrifugal buoyancy to be isolated. The global features of these flows are documented in the literature, but there are conflicting results regarding the shroud Nusselt number scaling with the Rayleigh number and
the nature of the disk boundary layers is not established. In this chapter LES was used to obtain time-averaged flow statistics for a sealed rotating annulus, and significant insight into the disk and shroud boundary layers behaviour was obtained.

Analysis of the disk kinetic boundary layer reveals a complex behaviour. In the time- and \( \theta \)-averaged sense, as the radius increases the maximum radial velocity decreases, eventually becoming negative close to the shroud. The results show that, away from the velocity reversal region, the boundary layer thickness \( \delta_0 \) can be approximated by that of a laminar Ekman layer, \( \pi \sqrt{\nu/\Omega} \). As the mean radial velocity is very small compared to the disk speed and becomes negative in some regions, it is interesting to analyse the boundary layer behaviour in terms of the radial velocity fluctuation \( u_{r,\text{rms}} \). The thickness \( \delta_{\text{rms}} \) based on the maximum value of \( u_{r,\text{rms}} \) shows a scaling of \( \delta_{\text{rms}} \propto \Omega^{-0.517} \), which is close to what is expected for an Ekman layer. Additionally, the shape of the \( u_{r,\text{rms}} \) profiles have a universal shape within the range of \( Ra \) studied here. Instantaneous velocity profiles obtained at different \( \theta \) positions were compared with an analytical Ekman solution where a non-zero far-field radial velocity is assumed. The good agreement between this solution and the numerical predictions indicates that, even instantaneously, the disk boundary layers behave as laminar Ekman layers.

When the disks are heated rather than adiabatic, a thermal boundary layer develops. The time-averaged temperature changes rather slowly near the disk, which indicates that its behaviour is influenced by conductive effects. The temperature \( \text{rms} \) fluctuation, in contrast, peaks at a short distance from the disk, comparable to where the peak of \( u_{r,\text{rms}} \) occurs. An analysis of the instantaneous behaviour of near-disk temperature profiles revealed that at some azimuthal locations the temperature gradients are confined to thin boundary layers, whilst at others the temperature varies more gradually. This suggests that both convective and conductive effects play a role in the dynamics, although the latter may become negligible at Rayleigh numbers higher than those considered here.

Along the radial direction, the averaged temperature profiles consist of a uniform core, which separates two thermal boundary layers near the shaft and shroud surfaces. The results show that the value of the core temperature increases slightly when \( Ra \) is increased, whereas temperature fluctuations decrease. Since the normalised core temperature \( T_{\text{core}}^* \) is larger than \( 0.5 \), the temperature fluctuation is larger near the shaft than near the shroud. The shroud boundary layer thickness \( \lambda_{\text{rms}} \) is observed to scale as \( \lambda_{\text{rms}}^* \propto Ra^{-0.321} \), in agreement with observations for RB convection.

Values of the shroud Nusselt number are significantly higher than those reported in the experimental work of Bohn et al. [53], but are in better agreement with a correlation for natural convection in a fluid layer heated from below. Previous numerical studies provided conflicting results regarding the scaling of \( Nu \) with \( Ra \). While more experimental work is needed to investigate this issue, other aspects that could be investigated numerically include the use of a fully compressible flow solver and also the use of higher Rayleigh numbers. These, however, would be significantly more computationally demanding than the approach used in the present research.

As far as the author is aware, this study is the first to report statistics for buoyancy-induced flow in a rotating cavity. The long-standing question about the nature of the disk boundary layers was addressed by analysing instantaneous and mean velocity profiles, as well as fluctuating quantities. The scaling relationships
obtained for the disk and shroud boundary layers can help in the development of simplified models for these flows.
Chapter 7

Buoyancy-induced flow in a rotating cavity with an axial throughflow

The studies presented in Chapters 5 and 6 focused on flow induced by centrifugal buoyancy in a sealed rotating annular cavity. In modern jet engines the rotating cavities are open, with an axial throughflow of cooling air entering and leaving the cavities, as shown schematically in Fig. 1.2. During this process, the cooling air extracts heat from the disks and from the shroud. As explained in previous chapters, the sealed cavity model can provide information of fundamental interest regarding the disk and shroud boundary layers, as well as about the general flow structure. In this chapter, the analysis presented in Chapter 6 is extended with the inclusion of an inlet and an outlet of cooling air. Specifically, some of the analyses performed for the sealed cavity case are repeated, and the effect of the axial Reynolds number on the flow structure is discussed. As for the sealed cavity case, previous studies of buoyant flow in a rotating cavity with an axial throughflow were focused on extracting global quantities, whereas here time-averaged profiles are analysed.

7.1 Problem description

7.1.1 Geometry and boundary conditions

Consider air with $\nu = 1.5 \times 10^{-5}$ m$^2$/s, $Pr = 0.7$ at temperature $T = T_i$, flowing through a rotating annular pipe of inner and outer radii $r_s$ and $a$, respectively, at a mean axial velocity $W$, entering a rotating cavity at $z = 0$ and exiting the cavity into an outlet pipe at $z = d$. The lengths of the inlet and outlet pipes are equal to $1.1d$. This configuration is illustrated schematically in Fig. 7.1(a), and the geometry of a sealed rotating annulus employed in previous chapters is shown in part (b) of the figure. In order to allow a comparison with the results obtained for the sealed cavity, the open cavity has $a = 0.125$ m, $b = 0.24$ m and $d = 0.12$ m, just as the sealed case, and $r_s = 0.115$ m.

As in Chapters 5 and 6, the centrifugal Rayleigh number $Ra$, the rotational Reynolds number $Re$ and the Prandtl number $Pr$ are considered here, with the same definitions previously presented. Additionally, an axial Reynolds number $Re_z$ based on the hydraulic diameter $d_h = 2(a - r_s)$ of the inlet pipe is defined as,
To quantify the importance of the axial throughflow jet in comparison with rotational effects, a Rossby number $Ro$ is also defined,

$$Ro = \frac{W}{\Omega a},$$

so that $Ro = 0$ for the case of zero throughflow rate and $Ro \rightarrow \infty$ for the case without rotation. In this chapter the Rayleigh number is kept constant, $Ra = 10^8$, and the rotational Reynolds number is calculated from $Ra$ using the same correlation employed in Chapter 6, $Re = 1.441Ra^{0.557}$. Two values of the axial Reynolds number are considered in detail, $Re_z = 1000$ and $Re_z = 2000$, giving Rossby numbers $Ro = 0.2$ and $Ro = 0.41$, respectively. While the values of $Ra$, $Re$ and $Re_z$ considered are lower than those found in engine conditions, the range of $Ro$ used here is consistent with engine applications, where $Ro$ is in the order of $10^{-1}$. Additionally, some results are presented for a case with $Re_z = 5600$. It should be noted that the fact that the flow at the inlet pipe is expected to be turbulent for this condition poses a challenge in the context of both DNS and LES, since a rather significant numerical effort is spent to resolve this flow. For the other two cases, where $Re_z$ is low enough so that the flow is laminar, the solution can be obtained in a straightforward manner, since a fully developed velocity profile for an annular pipe, which has an analytical expression, is imposed at the inlet.

In all cases the fluid enters the inlet pipe at a temperature $T_i$, the inlet and outlet pipes are adiabatic, the shroud is at a constant temperature $T_b$ and an outflow condition is imposed at the outlet. As in the study of a sealed cavity presented in Chapter 6, the disks are either adiabatic or heated. For the latter case, one set of experimental data reported by Atkins & Kanjirakkad [93] was interpolated to a polynomial of order 3, which was imposed to both the upstream and downstream disks. This profile is shown in Fig. 7.2. Note that for this profile the value of $T^*$ at $r^* = 0$ is greater than zero, which is consistent with the experimental conditions. For the case of a sealed cavity presented in Chapter 6 and repeated here, the profile
is such that \( T^* = 0 \) at \( r^* = 0 \) to avoid a discontinuity in the region where the disk and the shaft are connected.

While the flow at the inlet pipe is laminar, the flow exiting the cavity into the outlet pipe is expected to be very chaotic. If large vortices are present at the outflow surface, a negative velocity might exist, which may cause numerical instabilities, especially if a standard outflow boundary condition is imposed using a Dirichlet condition for the pressure and a zero-stress condition for the velocities. One alternative is to ensure that the flow is fully developed at the outlet. This means that the outlet pipe has to be sufficiently long so that the flow achieves a fully developed state, which increases the computational effort. The alternative adopted in this work consists on using the ‘robust’ outflow boundary condition proposed by Dong et al. \[135\], which allows for truncated domains to be used without compromising the solution accuracy. In terms of the numerical implementation, this method also uses a Dirichlet condition for the pressure and Neumann for the velocities, but the prescribed values are calculated such that any energy entering the domain (reverse flow) is balanced by an imposed stress. Mathematically, this can be written as \[135\],

\[
- \frac{p}{\rho} \mathbf{n} + \nu \mathbf{n} \cdot \nabla \mathbf{u} = \left[ \frac{1}{2} |\mathbf{u}|^2 S_0(\mathbf{n} \cdot \mathbf{u}) \right] \mathbf{n},
\]

which is used at the outflow boundary. In the equation above, \( \mathbf{n} \) is the unit vector pointing outwards from the outflow boundary, and \( S_0(\mathbf{n} \cdot \mathbf{u}) \) acts as a smoothed step function, such that \( S_0 = 1 \) if \( \mathbf{n} \cdot \mathbf{u} < 0 \) (i.e., if there is a negative velocity on the boundary), and \( S_0 = 0 \) otherwise. Thus, the stress \(- (p/\rho) \mathbf{n} + \nu \mathbf{n} \cdot \nabla \mathbf{u}\) will also be zero if \( \mathbf{n} \cdot \mathbf{u} > 0 \), otherwise it is equal to the influx of kinetic energy into the domain, given by the right-hand side of Eq. 7.3.

### 7.1.2 Details of the computations

For the cases of a laminar throughflow, the mesh used has 452 spectral elements in the \( r - z \) plane. The resolution near the shroud and disk surfaces is the same as for the sealed cavity case described in Chapter 6 and more elements are included at low radial positions, since this is the region where the axial jet interacts with the flow inside the cavity. This mesh is shown in Fig. 7.3. Note that, since the flow at the inlet pipe is laminar, the resolution in that region can be significantly relaxed.
Since the calculations are expensive to run, a grid-independence study was performed considering two configurations with significantly different degrees of freedom. A coarse mesh was considered setting the polynomial order to $P = 7$ and using $N_\theta = 96$, whereas in a finer mesh the parameters $P = 9$ and $N_\theta = 256$ were used. From the grid-independence study presented in Chapter 6, the results are expected to be more sensitive to the resolution along the azimuthal direction than in the $r-z$ plane. The two grids were assessed by running the case $Re_z = 1000$ with adiabatic disks, and using SVV stabilisation with an amplitude $\epsilon = 4\nu$. For the coarse mesh, $M_P = 5$ and $M_\theta = 24$, whereas for the finer mesh $M_P = 6$ and $M_\theta = 64$, where $M_P$ and $M_\theta$ are the SVV wavenumber cut-off parameters.

Table 7.1 shows the effect of $P$ and $N_\theta$ on key quantities analysed in this chapter. The results were obtained from time- and circumferentially-averaging the solution for at least 10 disk revolutions, after a statistically steady state based on the core temperature and shroud Nusselt number was reached. To extract the core temperature, the solution was averaged along the axial direction, $0 \leq z^* \leq 1.0$, and only a limited radial section, $0.4 \leq r^* \leq 0.6$, was averaged to obtain the value of $T_{core}^*$. Note that the non-dimensional radial coordinate is defined in the same way as in Chapter 6 i.e. $r^* = (r-a)/(b-a)$, and $z^* = z/d$. In contrast with the sealed system studied previously, in the axial throughflow case the solution is not nearly homogeneous in the axial direction, as discussed later. The disk kinetic boundary layer thickness $\delta_{rms}^* = \delta_{rms}/d$ was calculated based on the distance from the upstream disk where the radial velocity fluctuation $u_{r,rms}$ peaks, and averaged over the interval $0.2 \leq r^* \leq 0.8$ so as to exclude the region influenced by the cylindrical surfaces. $\lambda_{rms}^* = \lambda_{rms}/(b-a)$ represents the shroud thermal boundary layer thickness based on the peak of $T_{rms}$, and was calculated from an average along $0.2 \leq z^* \leq 0.8$. Finally, the Nusselt number shown in Table 7.1 was evaluated by computing the heat flux on the shroud surface and dividing it by the heat flux due to conduction in the sealed cavity case.

Although only two meshes are considered, it is clear that the quantities analysed are only weakly affected by the change in resolution. As observed in Chapter 6, the shroud boundary layer thickness is the most sensitive of the quantities, while the disk kinetic boundary layer thickness is effectively independent of the resolution. To have better reliability on the values of $\lambda_{rms}^*$, the mesh with $P = 9$ and $N_\theta = 256$ was employed in the remaining calculations.

At the end of this chapter, the case of a turbulent inlet with $Re_z = 5600$ is...
Table 7.1: Effect of the polynomial order \( P \) and number of azimuthal planes \( N_\theta \) on \( T_{\text{core}}, \delta_{\text{rms}}, \lambda_{\text{rms}} \) and \( Nu \), for \( Ra = 10^8 \) and \( Re_z = 1000 \) on the mesh with \( N_{el} = 452 \) spectral elements shown in Fig. 7.3.

<table>
<thead>
<tr>
<th>( P )</th>
<th>( N_\theta )</th>
<th>( T_{\text{core}}^* )</th>
<th>( \delta_{\text{rms}}^* )</th>
<th>( \lambda_{\text{rms}}^* )</th>
<th>( Nu )</th>
</tr>
</thead>
<tbody>
<tr>
<td>7</td>
<td>96</td>
<td>0.2312</td>
<td>0.01768</td>
<td>0.01394</td>
<td>64.51</td>
</tr>
<tr>
<td>9</td>
<td>256</td>
<td>0.2260</td>
<td>0.01768</td>
<td>0.01516</td>
<td>62.09</td>
</tr>
</tbody>
</table>

considered. The mesh used in this case has 608 spectral elements, \( P = 7 \) and \( N_\theta = 256 \), with significantly more elements in the inlet pipe region. Although the number of degrees of freedom of this mesh is not significantly larger than for the cases of a laminar axial throughflow, in practice the time step required to maintain numerical stability is an order of magnitude smaller than for the laminar case. For this reason, statistical convergence has not been obtained, and thus only preliminary results are presented. For reference, the time steps used, normalised by a characteristic time \( \Omega^{-1} \), are \( \Delta t^* = 2.9 \times 10^{-4} \) for \( Re_z = 1000 \), \( \Delta t^* = 5.9 \times 10^{-5} \) for \( Re_z = 2000 \) and \( \Delta t^* = 1.5 \times 10^{-5} \) for \( Re_z = 5600 \).

### 7.2 Results and discussion

To have a first impression of how the presence of an axial throughflow affects the flow structure inside the cavity, Fig. 7.4 shows contours of the normalised temperature \( T^* = (T - T_i)/(T_b - T_i) \) for the cases of a sealed cavity, \( Re_z = 1000 \) and \( Re_z = 2000 \). In all cases the snapshots were taken at the mid-axial position \( z^* = 0.5 \), and the disks are adiabatic. For the sealed cavity the temperature is nearly constant except near the boundary layers, although there is a rather weak signature of a structure comprising pairs of counter-rotating cells, or radial arms of cold (hot) fluid moving outwards (inwards). When the axial throughflow is included the temperature field is significantly less uniform, as shown in Figs. 7.4(b) and (c). The presence of radial arms of cold fluid penetrating the cavity is more evident, and the overall temperature in the cavity core is significantly reduced, which has an impact on the shroud heat transfer, as discussed later.

The instantaneous contours for \( Re_z = 1000 \) and \( Re_z = 2000 \) show a similar behaviour, although for the latter case the cold fluid penetrates further radially into the cavity. An analysis of time- and circumferentially-averaged contours in the \( r - z \) plane shows that the temperature along the axial direction has a rather different behaviour between the two cases – see Fig. 7.5(a). First, note that for the sealed cavity case the temperature is quasi-uniform along the axial direction, and the contours are symmetric relative to \( z^* = 0.5 \). Also, the axial velocity, here denoted by \( u_z^* = u_z/(\Omega a) \), is very small. When the axial throughflow is included the flow loses its near-homogeneity along the axial direction, and a decrease of \( T^* \) with \( z^* \) occurs for \( Re_z = 1000 \), whereas for \( Re_z = 2000 \) the opposite occurs. These contours indicate that the core is not fully mixed, which may be due to the low axial Reynolds numbers considered here. The contours of axial velocity shown in Fig. 7.5(b) help with the interpretation of these results. For \( Re_z = 1000 \) the axial jet is almost immediately broken, in the sense that its axial velocity reduces
significantly, when it enters the cavity. The cold fluid entering the cavity is then pulled radially outwards by buoyancy effects, which causes a greater reduction in the temperature in the left part of the cavity. For $Re_z = 2000$ the cold fluid entering the cavity has more momentum and penetrates further axially into the cavity before it is significantly affected by the buoyant force. Thus, the temperature reduction inside the cavity is greater on the right-hand side.

### 7.2.1 Kinetic boundary layers

An analysis of averaged fields in terms of fluctuating quantities reveals that the flow throughout the cavity has some similarities with the case of a sealed cavity. Near each of the disks, for instance, a kinetic boundary layer can be defined in terms of the peak of the radial velocity fluctuation $u_{r,rms}^*$. Figure 7.6(a) shows near-disk profiles of $u_{r,rms}^*$ at $r^* = 0.5$ for the sealed cavity and for the two axial throughflow cases with adiabatic disks. Although all the curves are similar, it is clear that the presence of an axial throughflow increases the magnitude of the velocity fluctuations, which is not surprising considering the fact that, in these cases, cold fluid is fed continually into the cavity, whereas in the sealed cavity case the fluid circulation inside the cavity occurs due to natural convection only. Despite the qualitative differences in the flow structure between the cases $Re_z = 1000$ and $Re_z = 2000$, the profiles are nearly identical at the radial station considered. Similarly – see Fig. 7.6(b) – a radial profile of $u_{r,rms}^*$ averaged along the entire axial extent of the cavity ($0 \leq z^* \leq 1$) also reveals that the presence of the axial throughflow increases the fluctuations in comparison with the sealed cavity case. The profiles are again rather similar for the two values of $Re_z$. Please note that the axial profile $u_{r,rms}^*$ is normalised with $\Omega r$, where $r$ is the local radius, whereas the radial profile $u_{r,rms}^*$ is normalised with $\Omega a$ so that its shape is not distorted. Also, note that the non-dimensional radial coordinate is defined as $r^* = (r - a)/(b - a)$, just as in Chapter 6, and that is the reason why the region between the shaft and the inner disk radius has a negative $r^*$ in Fig. 7.6(b).
In contrast with the sealed cavity case, the cases with axial throughflow are not symmetric with respect to the $z^* = 0.5$ plane, which means that the upstream and downstream disks may behave differently. Figures 7.7(a) and (b) show the radial variation of the disk kinetic boundary layer thickness based on the peak of $u^*_{r,rms}$ for the upstream and downstream disks, respectively, for the two values of $Re_z$ considered. The dashed and dot-dashed lines indicate the thickness of a laminar Ekman layer and the Ekman depth, respectively. Note that these figures can be compared to Fig. 6.4(b), which shows an analogous plot for the sealed cavity case, where both disks behave identically. The values of $\delta^*_{rms}$ are overall slightly higher in the cases with an axial throughflow, but they are still consistent with the thickness of an Ekman layer. For $Re_z = 1000$ $\delta^*_{rms}$ is nearly constant along $r^*$ for both disks, but for $Re_z = 2000$ much more significant variations occur. One possible explanation for the change in behaviour near the downstream disk for $Re_z = 2000$ is that the axial jet is impinging on the disk. This is supported by the contours of temperature and axial velocity shown in Fig. 7.5, where it is clear that the cavity is colder near the downstream disk. In order to fully characterise the nature of the disk boundary layers, it would be necessary to consider higher values of $Re_z$, to assess whether this would promote more mixing inside the cavity, and also different values of the rotational Reynolds number $Re$ and Rayleigh number $Ra$ (recall that here only $Ra = 10^8$ is studied), to analyse the scaling of $\delta^*_{rms}$ with $Re$. The results...
presented here suggest, however, that the values of $\delta_{\text{rms}}^*$ still scale consistently with a laminar Ekman layer.

In Chapter 6, it was shown that the disk boundary layers behave as laminar Ekman layers even in the instantaneous sense. Defining a non-zero core radial velocity, as well as a core tangential velocity, the analytical solution given by Eqs. 6.3 and 6.4 was compared with instantaneous profiles obtained numerically. Figures 7.8 and 7.9 show analogous profiles obtained for the axial throughflow cases near the upstream and downstream disks, respectively. Note that the values of $\theta$ were chosen to illustrate regions where the numerical solution is and is not consistent with the laminar Ekman solution. One difficulty imposed by the axial throughflow is that the flow is not nearly homogeneous along $z$, which makes it difficult to define a core velocity. Still, the profiles are overall consistent with the Ekman solution in terms of the thickness of the boundary layer. Note that the range of $z^*$ shown in the plots is very narrow, up to $z^* = 0.05$ and $1 - z^* = 0.05$ for the upstream and downstream disks.

### 7.2.2 Temperature profiles and heat transfer

Although the instantaneous contours of Fig. 7.4 revealed that when an axial throughflow is present the temperature across the cavity is much less uniform than in a sealed system, when the results are time- and circumferentially-averaged a core of constant temperature is still observed, as shown in Fig. 7.10(a). As expected, the cooling air entering the cavity causes a dramatic reduction in the core temperature, however it is interesting to note that the value of $Re_z$ has little to no effect on the results. The existence of a core temperature motivates a comparison of the shroud heat transfer with correlations for natural convection over a horizontal heated flat plate, just as in Chapter 6. A modified Nusselt number $Nu'$, which depends on the core tempera-
Table 7.2: Modified shroud Nusselt number $Nu'$ for the case of adiabatic disks.

<table>
<thead>
<tr>
<th>Case</th>
<th>$q_{conv}/k [K/m]$</th>
<th>$Nu'$</th>
<th>$Nu'$ (low $Ra'$)</th>
<th>$Nu'$ (high $Ra'$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Sealed</td>
<td>160.2</td>
<td>26.84</td>
<td>27.46</td>
<td>28.26</td>
</tr>
<tr>
<td>$Re_z = 1000$</td>
<td>396.6</td>
<td>30.74</td>
<td>33.30</td>
<td>36.54</td>
</tr>
<tr>
<td>$Re_z = 2000$</td>
<td>407.2</td>
<td>31.62</td>
<td>33.28</td>
<td>36.52</td>
</tr>
</tbody>
</table>

Figure 7.7: Radial variation of the disk kinetic boundary layer thickness based on the peak of $u_{r,rms}^*$, for the (a) upstream and (b) downstream disks.

7.2.3 Heated disks

When the temperature profile shown in Fig. 7.2 is imposed on both the upstream and downstream disks, no major difference in the flow behaviour is observed. Please
Figure 7.8: Instantaneous profiles of (a,c) radial and (b,d) tangential velocity at $r^* = 0.5$ near the upstream disk, for (a,b) $Re_z = 1000$ and (c,d) $Re_z = 2000$, with adiabatic disks. The dashed lines indicate the laminar Ekman solutions introduced in Chapter 6.
Figure 7.9: Instantaneous profiles of (a,c) radial and (b,d) tangential velocity at $r^* = 0.5$ near the downstream disk, for (a,b) $Re_z = 1000$ and (c,d) $Re_z = 2000$, with adiabatic disks. The dashed lines indicate the laminar Ekman solutions introduced in Chapter 6.
recall that the profile of Fig. 7.2 is applied for the cases with axial throughflow, whereas for the sealed cavity a similar profile, with $T^*$ ranging from 0 to 1, is considered, just as presented in Chapter 6. Figure 7.11 shows temperature and axial velocity contours for the sealed cavity and for $Re_z = 1000$ and $Re_z = 2000$. These contours can be compared with Fig. 7.4 for the case with adiabatic disks. Due to the imposition of a temperature profile, a thermal boundary layer develops near the disks, but away from them the temperature contours are similar in the two cases. The contours of axial velocity are virtually unaffected by the disk thermal boundary conditions. Figure 7.12 shows profiles of the disk kinetic boundary layer thickness $\delta_{rms}$ near the upstream and downstream disks. In Fig. 7.7 the corresponding plots for the adiabatic disks case were presented. Again, these figures indicate that heating the disks has little effect on the behaviour of the kinetic boundary layer, whose thickness is consistent with that of a laminar Ekman layer.

The disk heat transfer is analysed in terms of a disk Nusselt number, $Nu_d$, presented in Chapter 6, given by,

$$Nu_d = \frac{q_r}{k(T_{disk} - T_i)},$$

where $T_i$ is the temperature of the cooling air. For a sealed cavity, the profiles of $Nu_d$ of the upstream and downstream disks are identical, but in the cases with axial throughflow the heat transfer is expected to be different between the disks, especially if jet impingement occurs. Figure 7.13 shows profiles of $Nu_d$ on both disks for $Re_z = 1000$ and 2000. For the lower value of $Re_z$, the two profiles are similar, which indicate that jet impingement effects are rather weak. For $Re_z = 2000$, in contrast, $Nu_d$ reaches much higher values near the downstream disk and is positive for all $r^*$, i.e., the disk is losing heat to the cooling flow. This, together with the fact that the thickness of the kinetic boundary layer changes significantly with $r^*$ for $Re_z = 2000$ (see Figs. 7.7 and 7.12), indicates that jet impingement effects are strong for this value of $Re_z$, even though this is not evident from the contours of
Figure 7.11: Time- and circumferentially-averaged contours of (a) normalised temperature and (b) normalised axial velocity, for the case of heated disks.

Figure 7.12: Radial variation of the disk kinetic boundary layer thickness based on the peak of $u_{r,rms}$ for the case of heated disks, for the (a) upstream and (b) downstream disks.
Figure 7.13: Profiles of the disk Nusselt number $N_u_d$ as a function of the radial position, for both the upstream and downstream disks.

Figure 7.13 also shows plots of the modified disk Nusselt number $N_u'_d$, which is given by,

$$N_u'_d = \frac{(T_{disk} - T_{r,core})r}{k\delta_0(T_{disk} - T_i)}, \quad (7.5)$$

where $T_{r,core}$ is a radially local core temperature, defined as the averaged temperature along the axial direction, and $\delta_0$ is the thickness of a laminar Ekman layer, $\delta_0 = \pi \sqrt{\nu/\Omega}$. Note that $N_u'_d$ is the same for both disks, as it is based on a core temperature. Although this simple expression is not able to reproduce the values of the disk Nusselt number with good accuracy, it provides a reasonable qualitative description of the disk heat transfer.

### 7.2.4 Spectral analysis

To analyse the differences between the flow features observed across the cavity, time series collected at specific locations are considered. First, the frequency contents of points located inside and outside the disk boundary layers are compared. Figure 7.14 shows, on the top row, time series of the normalised temperature $T^*$ as a function of the number of revolutions $t^*$ at $r^* = 0.5$, $\theta = 0$, for three axial positions: $z^* = 2 \times 10^{-4}$, $z^* = 0.02$ and $z^* = 0.5$. It should be noted that for the point very close to the disk the temperature oscillates with a rather small amplitude in all cases, but the frequency content shown by the power spectral density (PSD) curves is similar to those obtained further from the disk. The point located at $z^* = 0.02$ is immediately outside the boundary layer – cf. Figs. 7.7 and 7.12 – and shows more significant oscillation amplitudes, comparable to those observed at $z^* = 0.5$. For the sealed cavity, the oscillations observed are remarkably slow compared to the axial throughflow cases, which is reflected by the lack of high-frequency content shown in the PSD curves. Despite the relatively slow frequency of the oscillations, they
do present a chaotic behaviour, although the PSD does not seem to have a $f^{-5/3}$ region, which is characteristic of the inertial subrange in turbulent flows.

When an axial throughflow is present, the flow becomes much more chaotic and dominated by high-frequency oscillations, even very close to the upstream disk (Fig. 7.14(a)), and despite the fact that the axial throughflows considered are laminar. For both $Re_z = 1000$ and $Re_z = 2000$ the PSD plots reveal a $f^{-5/3}$ region, thus suggesting that the flow is fully turbulent – note that this occurs even at $z^* = 2 \times 10^{-4}$.

Figure 7.15 shows temperature time histories at $r^* = 0.98$, i.e. inside the shroud thermal boundary layer, and at $r^* = 0.9$, or just outside the boundary layer, as well as the corresponding PSD curves. In this region, again, the presence of an axial throughflow increases the range of frequencies observed, as well as the amplitude of the temperature oscillations. The chaotic nature of the time series suggests that the heat transfer from the shroud to the cavity occurs via turbulent convection, even for the sealed cavity case. To investigate the interaction between the axial jet and the cavity, Fig. 7.16 shows temperature time series at $r^* = 0.04$ and at the mid-axial position. At this location the spectra of frequencies become even broader than at the other points analysed. It is interesting to note that the increase observed in the frequency range comes solely from the interaction between the jet and the cavity flow, and not from the jet itself, which is laminar at the cavity inlet. Previous studies suggested that buoyancy-induced flows in cavities with an axial throughflow can be split into a region of forced convection at low radii and a region dominated by natural convection, near the shroud. The results presented here show qualitatively that the jet has a strong interaction with the cavity flow at low radii, but also that it has a significant influence on the frequency content even very close to the shroud. As shown earlier, however, correlations commonly employed for natural convection are able to provide reasonable predictions for the shroud heat transfer. Therefore, while the present results indicate that the shroud region is dominated by natural convection, they also show that the axial throughflow has a strong effect on the temperature oscillations near the shroud.

7.3 Turbulent inlet

In this section preliminary results are presented for the case of a turbulent axial throughflow with $Re_z = 5600$. The calculations are considerably more demanding when the flow entering the cavity is turbulent, since the pipe section of the mesh must be fine enough to resolve the flow, and also because the time step required is significantly smaller. The calculations performed for this case took several weeks to run, therefore a more detailed analysis, which would require longer averaging periods, was not carried out.

Figure 7.17 shows normalised time- and $\theta$-averaged temperature and velocity contours for the case of insulated disks. Although the averages were collected for only about two disk revolutions, they reveal features similar to those observed for the cases of lower $Re_z$. For $Re_z = 5600$ the axial jet penetrates further into the cavity, becoming weaker as it approaches the outlet pipe. In agreement with the cases at lower $Re_z$, the jet is not affected by vortex breakdown, and the fluid is instead transported into the cavity due to buoyancy effects. The temperature contours show that the cavity core is fairly isothermal, although the temperature is lower close to
Figure 7.14: Normalised temperature $T^*$ as a function of $t^*$ (top row) for the sealed cavity (black), $Re_z = 1000$ (blue) and $Re_z = 2000$ (magenta) and corresponding power spectral densities, for (a) $z^* = 2 \times 10^{-4}$, (b) $z^* = 0.02$, (c) $z^* = 0.5$. In all cases $r^* = 0.5$ and $\theta = 0$. The dashed lines indicate a $f^{-5/3}$ slope.
Figure 7.15: Normalised temperature $T^*$ as a function of $t^*$ (top row) for the sealed cavity (black), $Re_z = 1000$ (blue) and $Re_z = 2000$ (magenta) and corresponding power spectral densities, for (a) $r^* = 0.98$ and (b) $r^* = 0.9$. In all cases $z^* = 0.5$ and $\theta = 0$. The dashed lines indicate a $f^{-5/3}$ slope.
Figure 7.16: (a) Normalised temperature $T^*$ as a function of $t^*$ for $Re_z = 1000$ (blue) and $Re_z = 2000$ (magenta) at $r^* = 0.04$, $z^* = 0.5$ and $\theta = 0.0$. (b,c) Corresponding power spectral densities. The dashed lines indicate a $f^{-5/3}$ slope.

Figure 7.17: Contours of (a) normalised temperature (b) and axial velocity for the case of insulated disks with $Re_z = 5600$. The contours are plotted on a $r-z$ plane and correspond to a time- and $\theta$-average.

the downstream disk. This, however, might be due to the small time period used to collect the averages.

A spectral analysis based on time history data, which is done for the cases of heated disks only, could not be performed at this stage for $Re_z = 5600$ due to the small sampling period obtained. However, visual observation of the signals leads to the same conclusions obtained for the laminar axial throughflow cases. Figure 7.18 shows the temperature history at two points, one near the disk and another where the jet is expected to have a strong interaction with the cavity flow. At $r^* = 0.5$ and $z^* = 0.02$ (part (a) of the figure) the oscillations have less high-frequency content when compared to $r^* = 0.04$ and $z^* = 0.5$ (part (b)), in agreement with the observations at lower $Re_z$. 

136
Figure 7.18: Normalised temperature $T^*$ as a function of $t^*$ for $Re_z = 5600$ at (a) $r^* = 0.5$, $z^* = 0.02$, $\theta = 0.0$ and (b) $r^* = 0.04$, $z^* = 0.5$, $\theta = 0.0$. 
7.4 Conclusion

The results presented in this chapter extend the study of buoyant flow in a sealed rotating cavity presented in Chapter 6 by the inclusion of an axial throughflow of cooling air. Generally, the axial throughflow has the effect of decreasing the temperature inside the cavity, thus promoting an increase of the heat transfer from the shroud. Despite this, some of the conclusions drawn from the results for a sealed cavity are also valid for the cases $Re_z = 1000$ and 2000. Specifically, the thickness of the disk kinetic boundary layers near both the upstream and downstream disks is consistent with that of a laminar Ekman layer, although for $Re_z = 2000$ the thickness changes with the radial coordinate. Instantaneously, the definition of a core radial and tangential velocity is less evident than in the sealed cavity case, due to the axial asymmetry of the solution. Still, instantaneous velocity profiles are in fair agreement with a laminar Ekman solution, especially in terms of the boundary layer thickness.

An analysis of the shroud heat transfer reveals that, although the presence of the axial throughflow has a significant effect on the heat transfer, the results are not very sensitive to the value of $Re_z$, at least for the range considered here. Defining modified Rayleigh and Nusselt numbers $Ra'$ and $Nu'$, respectively, based on the core temperature $T_{core}$ and on the local centrifugal acceleration at the shroud, the same correlations for natural convection over a horizontal flat plate presented in Chapter 6 were considered for $Re_z = 1000$ and 2000. The values of $Nu'$ obtained from the correlations overestimate the numerical results by up to 20%, therefore further studies with different $Ra$ and $Re_z$ would be necessary to assess whether this analogy can be applied successfully for rotating cavities with an axial throughflow.

A spectral analysis based on time history data collected at different locations revealed that, even with a laminar axial throughflow, the interaction between the axial jet and the cavity alone promotes an increase of the high-frequency content of the signals. For the sealed cavity the temperature oscillates rather slowly over time, but in a chaotic manner. When the axial throughflow is present, the oscillations occur faster, especially in the region of low radius, where the jet interacts with the cavity flow. Thus, although the shroud region is dominated by natural convection effects, its frequency content is significantly altered by the presence of an axial throughflow.

As a next step to improve the understanding of these flows, it would be of interest to use different Rayleigh numbers and higher axial Reynolds numbers. The results presented here, however, indicate that $Re_z$ has a small effect on key features of the flow, such as the shroud heat transfer and the power spectral density plots. More significant differences are observed for the disk heat transfer, although this is mostly related to the axial distribution of the temperature inside the core.
Chapter 8

Conclusions

In this thesis a high-fidelity CFD technique was employed to study buoyancy-induced flows in rotating cavities. Canonical configurations relevant within the context of gas turbines compressor cavities were considered, and significant insight into the physics of the flows studied was obtained. In this chapter a summary of the main findings is presented, as well as the limitations of the approach used. At the end of the chapter, suggestions for future research that could extend and complement the analyses presented here are given.

8.1 Summary of findings and limitations

Fundamental investigations of buoyancy-induced flows in rotating cavities were presented in Chapters 5, 6 and 7. These flows occur, for instance, inside the cavities formed between the compressor disks of gas turbines, and have long been recognised as challenging to study, either theoretically, numerically or experimentally. The fact that unsteady and three-dimensional effects should be considered in numerical analyses implies that turbulence models commonly used in industry are inadequate to predict buoyancy-induced flows. Thus, in this study model-free approaches were employed, namely direct numerical simulation and implicit large-eddy simulation. These approaches are associated with considerably higher computational costs than simulations that use RANS-based turbulence models.

In high-fidelity numerical simulations, numerical methods able to provide high-order accuracy are desirable. While the finite volume method is commonly used to simulate industrial flows, high-order and spectral methods have better convergence properties and provide better accuracy per degree of freedom, and are therefore more economical, particularly in the context of unsteady simulations. In the past, the use of spectral methods was limited to canonical configurations only, however with advances in the development of spectral element methods complex geometries can now be modelled. In this work, an existing incompressible Navier-Stokes spectral element-Fourier code was adapted and applied to study flows induced by centrifugal buoyancy.

8.1.1 Contributions

The first numerical study considered in the thesis consists of flow in a rotor/stator cavity, presented in Chapter 4. This can be seen as a preliminary study, as its main
objective was to assess the suitability of the code and to investigate the sensitivity of the results to mesh refinement. The numerical results compare favourably with experimental data available in the literature for the two values of the rotational Reynolds number considered. Applying the spectral vanishing viscosity stabilisation technique, good results in terms of velocity and velocity fluctuation profiles could be obtained using much coarser meshes than for a DNS, however the moment coefficient was shown to be more sensitive to the resolution employed. The results show that even when the near-wall resolution is relaxed the results are still in agreement with the experimental data. In rotor/stator flows there is usually an inviscid core, in the time-average sense, which separates the rotor and stator boundary layers. Obviously, significantly more resolution is necessary near the boundary layers than in the core region, which makes the spectral element method a suitable candidate for this kind of flow, since it allows local mesh refinement, in contrast with global spectral methods, whose mesh-refinement capabilities are more limited. It is worth mentioning that turbulence models have been used with success in numerical studies of rotor/stator flows relevant in industrial applications. Still, the use of more accurate methods, such as that used here, allows for studies of fundamental aspects of these flows, such as the onset of instabilities in the rotor and stator boundary layers and the resulting large-scale structures. Additionally, model-free approaches are suited for cases where the flow is transitional, allowing better predictions than those obtained with turbulence models.

In Chapters 5 and 6, flow induced by centrifugal buoyancy in a sealed rotating annulus bounded by plane, parallel disks was investigated. In modern jet engines, the compressor cavities are not sealed, as an axial throughflow of cooling air is used to extract heat from the disks more effectively. Still, the sealed annulus configuration is of interest since it reproduces the physical phenomena encountered in an engine, such as rotation, a positive temperature gradient in the radial direction, and the presence of side-wall disks. Additionally, this model has also been employed in many experimental and numerical studies of geophysical phenomena.

Linear stability analysis and direct numerical simulation were used in the study of Chapter 5 where the onset of convection due to centrifugal buoyancy was investigated for different radius ratios. DNS was then used to corroborate some of the results obtained from linear stability theory and to study the dynamics of the non-linear regime for different Rayleigh numbers. The linear stability results reveal that convection is initiated in the form of pairs of counter-rotating cells, of alternating positive and negative temperature, analogously to Rayleigh-Bénard convection. As the radius ratio is increased, the critical Rayleigh number for the onset of convection decreases monotonically, approaching the value 1708 as the radius ratio tends to one. The critical azimuthal wavenumber, in contrast, increases with the radius ratio. Assuming that the convection cells are circular at the onset, a simple expression was obtained to estimate the critical azimuthal wavenumber, as a function of the radius ratio only, with excellent accuracy. DNS calculations were performed starting from a purely conductive state, to which small-amplitude sinusoidal perturbations of different azimuthal wavenumbers were added. By doing this, the exponential growth rates obtained from the non-linear calculations could be compared with those of the linear stability analysis. Up to a certain time, in the DNS, the growth rates matched the linear predictions, however triadic interactions arising from the non-linear term of the governing equations occurred, which increased the growth rate of certain az-
imathal modes, even before the energy level saturated. Two Rayleigh numbers were considered in the non-linear calculations: $Ra = 10^5$ and $Ra = 10^8$. For the lower $Ra$, the non-linear solution obtained was dominated by a single wavenumber, corresponding to the highest wavenumber initially perturbed, and its harmonics, which indicates that non-linear effects were not strong enough to eliminate the influence of the initial perturbation. For $Ra = 10^8$, in contrast, effects of the initial perturbation quickly disappeared after the transition, and the azimuthal energy spectrum was dominated by broadband effects, as it is typical for turbulent flows.

Another aspect considered in Chapter 5 was the influence of the side-walls, or disks, on the solution. From previous studies it was known that, for an infinitely long annulus, i.e., without disks, the solution consisted of a standing wave, whereas with solid walls the flow structure drifted along the azimuthal direction. This behaviour was observed in the present study. In addition, for $Ra = 10^5$, the inclusion of solid disks not only caused the flow structure to drift, but it also triggered a limit-cycle, characterised by a back-and-forth motion of the convection cells in the azimuthal direction, superposed to a global drift. This limit cycle is entirely absent in the case of an infinitely long annulus.

In Chapter 6 the non-linear regime was analysed in greater detail, using Rayleigh numbers equal to $Ra = 10^7$, $10^8$ and $10^9$. While previous numerical studies focused on the instantaneous flow structure and on the shroud heat transfer, the work presented here provides a more detailed description of buoyancy-induced flows in sealed rotating cavities. Specifically, the disk kinetic boundary layer was analysed by looking at time-averaged profiles of velocity and velocity fluctuation, as well as instantaneous velocity profiles. The results revealed that these boundary layers are laminar Ekman-type layers, both in terms of the boundary layer thickness and of the shape of the radial and tangential velocities. For the three values of $Ra$ considered, the boundary layer thickness, defined as the distance from the disk where the radial velocity fluctuation reaches a peak, scales as $\Omega^{-0.5}$, as expected for an Ekman layer. Instantaneously, the velocity profiles agree well with a laminar Ekman solution where both the core radial and tangential velocities are non-zero. The fact that the flow structure is formed of pairs of counter-rotating convection cells implies that the time-averaged velocities are very small, therefore it is more meaningful to analyse time-averaged rms quantities rather than the velocity components directly.

Temperature and velocity profiles along the radial direction revealed that, as $Ra$ increased, the magnitudes of the corresponding non-dimensional fluctuations in the cavity core decreased. Thus, increasing $Ra$ results in a more uniform flow structure, in the sense that the magnitudes of the non-dimensional fluctuations decrease. It should be noted that, since the system is rotating, Coriolis effects may also further damp the fluctuations. However, the scaling observed for the temperature fluctuations is consistent with previous results for Rayleigh-Bénard convection without rotation, which suggests that the Coriolis force plays a secondary role in promoting a more uniform flow structure.

The shroud Nusselt number and thermal boundary layer thickness scalings are consistent with that of Rayleigh-Bénard convection for high Rayleigh numbers, for which $Nu \propto Ra^{1/3}$ and $\lambda_{rms} \propto Ra^{-1/3}$. The values of $Nu$ obtained here are significantly higher than the experimental data available, but are in good agreement with correlations for natural convection under gravity. More experimental work would be important to investigate this issue. Additionally, numerical studies using
a compressible flow solver would also be useful, although for the range of parameters employed here compressibility effects are expected to be negligible.

To the author’s knowledge, the work presented in Chapter 6 is the first to report flow statistics for buoyancy-induced flows in sealed rotating cavities, and to provide a detailed description of the disk and shroud boundary layers. Thus, this can be seen as a key contribution of the present thesis.

Finally, in Chapter 7 buoyancy-induced flow in a rotating cavity with an axial throughflow of cooling air was considered. The analysis of the results was similar to that used for a sealed cavity in Chapter 6 and comparisons between the two cases were made. One major effect of the axial throughflow is to promote a reduction in the temperature inside the cavity, with a corresponding increase in the shroud heat transfer. For the limited range of parameters considered, it was observed that the core temperature and the shroud heat transfer were only weakly affected by the value of the axial Reynolds number. The disk boundary layers are again consistent with laminar Ekman layers, however the downstream disk presented a different behaviour in comparison with the upstream one, with rather significant variations in the boundary layer thickness occurring along the radial direction, possibly due to jet impingement.

An analysis of time-history data revealed that, when an axial throughflow is present, PSD curves present significantly more high-frequency content than for a sealed cavity, even close to the shroud. This increase of high-frequency content arises from the interaction between the axial jet and the buoyant flow inside the cavity. Interestingly, this is observed even for the cases where the axial throughflow is laminar. For a sealed cavity, on the other hand, the flow has a rather slow response, although it is clearly chaotic, as observed from the time signals.

8.1.2 Limitations

Since the calculations performed during this research are computationally demanding, which is usually the case when high-fidelity simulations are run, there are some limitations in terms of the range of parameters and geometries considered.

Perhaps the main limitation of the present work is that the Rayleigh and rotational Reynolds numbers employed are significantly lower than those found in gas turbine applications. This is, however, inherent to the fact that only DNS and LES were considered, which limits considerably the maximum value of parameters that can be achieved while still being able to obtain the results within a feasible time scale.

In terms of the modelling approach, it is important to note that incompressibility was assumed, with buoyancy effects accounted for using the Boussinesq approximation. This hypothesis is justifiable for the range of parameters used in this research, however for higher Rayleigh and Reynolds numbers compressibility effects may become important, depending on the Mach number. Still, knowledge of the flow physics neglecting compressibility provides good guidance for further studies.

The spectral element-Fourier code used throughout the research allows for arbitrary geometry complexity in the radial-axial plane, while the azimuthal direction is assumed to be homogeneous. This means that the code cannot account for changes in geometry along $\theta$, which are common in many industrial applications. In gas turbine compressor cavities, however, the geometry is generally axisymmetric, so
the assumption of homogeneity along $\theta$ is adequate. Also, the fact that the code is parallelised taking advantage of the linearity of the Fourier decomposition along $\theta$ means that the number of processors that can be used is limited to the number of Fourier modes. In practice, this means that performance becomes limited if the geometry is too complex in the radial-axial plane. However, for cases where canonical geometries are used and one is interested in focusing on fundamental aspects of the flow physics, as it was the case in this research, the numerical method used has proven to be very effective.

### 8.2 Suggestions for future research

In the following, topics that could be considered in future studies of buoyant flows in rotating cavities are suggested. The realisation of some of them is, however, heavily limited by the computational effort required.

- The study of Chapter 5 could be extended by performing DNS increasing the Rayleigh number progressively, from $Ra = 10^5$, where the flow is well-organised, to $Ra = 10^8$, where it is fully chaotic. This would provide more information about the key mechanisms that cause transition to turbulence.

- In the context of the studies of Chapters 6 and 7, it would be interesting to repeat the analyses for a wider range of the non-dimensional parameters, most importantly the Rayleigh number. This would clarify whether the conclusions obtained in terms of the disk and shroud boundary layers scalings are valid for lower and higher values of $Ra$. Also, it is of great interest to perform the calculations of Chapter 7 for higher values of the axial Reynolds number.

- To investigate to what extent the Coriolis force suppresses the fluctuations inside the sealed cavity, one could perform calculations without rotation and including a radial gravity component, to mimic the centrifugal buoyancy that would arise due to rotation. This would isolate the effect of centrifugal buoyancy and the importance of the Coriolis term on the results could be assessed.

- At higher Reynolds and Rayleigh numbers, compressibility effects may become important if the Mach number is high enough. Thus, future studies could use a fully compressible formulation, which is, however, significantly more computationally expensive and requires different solution algorithms than those used in this thesis.

- Finally, as a general suggestion, fundamental studies related to turbomachinery internal air systems could consider the use of high-order, or spectral, methods. Numerical studies within the community usually rely on low-order finite volume formulations. As shown in this thesis, in the context of model-free numerical simulations more accurate methods should be preferred. As an example, rim sealing flows are still very challenging to model, and high-fidelity simulations become relevant to study fundamental aspects of these flows which are still unclear. The work presented here shows that spectral element formulations have potential to perform such kind of investigations.
Appendix A

Boussinesq approximation for centrifugal buoyancy

In this appendix the application of the Boussinesq approximation to flows induced by centrifugal buoyancy is discussed. Traditionally, when the governing equations are written in a rotating frame, a variable density is considered on the centrifugal force only, and this approach was employed throughout this thesis. However, in a 2013 paper, Lopez et al. [11] proposed an extended Boussinesq approximation, in which the equations are written in an inertial, non-rotating reference frame. According to the authors, the approximation is able to account for different sources of buoyancy, such as internal vorticity of the flow, since a unique angular velocity is not assumed. In the present discussion, it is shown that, when the approximation of ref. [11] is transferred to a rotating frame, an extra term arises, which makes the approximation inconsistent. It is important to note, though, that the authors of ref. [11] discuss general aspects of the Boussinesq approximation in a very useful and clear way, and some of their arguments are repeated below.

In a non-inertial frame rotating at a constant angular speed $\Omega$, the momentum equations are,

$$\rho \frac{\partial \mathbf{u}}{\partial t} + \rho (\mathbf{u} \cdot \nabla \mathbf{u} + 2\Omega \times \mathbf{u} + \Omega \times (\Omega \times \mathbf{r})) = -\nabla p + \mu \nabla^2 \mathbf{u}, \tag{A.1}$$

where $\rho$ is a non-constant density. As discussed in Chapter 2 in the Boussinesq approximation it is assumed that: density variations $\rho'$ are small in comparison with a reference density $\rho_0$, i.e. $\rho' \ll \rho_0$; density is a function of temperature only, $\rho = \rho(T)$; density variations are only important in potential terms, i.e., those that can be written as the gradient of a scalar. Thus, $\rho \approx \rho_0 + \rho' = \rho_0 [1 - \alpha (T - T_0)]$, where $\alpha$ is the thermal expansion coefficient and $T_0$ is a reference temperature. In Eq. (A.1) the only term that can be written as the gradient of a scalar is the centrifugal force $\rho \Omega \times (\Omega \times \mathbf{r})$, whose constant-density part can be absorbed into the pressure gradient to form a reduced pressure $p^*$, already discussed in Chapter 2. The same treatment would apply for a gravitational force. Thus, for all the terms on the left-hand side of Eq. (A.1) $\rho \approx \rho_0$, except for the centrifugal force, for which $\rho \approx \rho_0 + \rho'$. This equation then becomes:

$$\rho_0 \frac{\partial \mathbf{u}}{\partial t} + \rho_0 \mathbf{u} \cdot \nabla \mathbf{u} = -\nabla p^* + \mu \nabla^2 \mathbf{u} - 2\rho_0 \Omega \times \mathbf{u} + \rho_0 \alpha (T - T_0) \Omega \times (\Omega \times \mathbf{r}). \tag{A.2}$$
Now, the argument used by Lopez et al. [11] is that, if the equations are written in a stationary frame, the rotation of the system enters the equations via the boundary conditions, and the non-linear term contains all the dynamics that arise from buoyancy effects. Thus, they propose a formulation where $\rho = \rho_0 + \rho'$ in the advective term, and $\rho = \rho_0$ for the remaining terms. Denoting the velocity vector in a stationary frame as $\mathbf{v}$, the approximation is given by,

$$\rho_0 \frac{\partial \mathbf{v}}{\partial t} + (\rho_0 + \rho') (\mathbf{v} \cdot \nabla \mathbf{v}) = -\nabla p + \mu \nabla^2 \mathbf{v}. \quad (A.3)$$

Since the transformation to a rotating frame $\mathbf{v} = \mathbf{u} + \Omega \times \mathbf{r}$ does not imply a loss of generality, it is possible to write Eq. [A.3] in a rotating frame. To treat the time-derivative term, first note that a constant angular velocity is assumed, thus $\partial_t (\Omega \times \mathbf{r}) = 0$. It is also necessary to consider a difference in the time derivative term between the two frames. To do this, consider the relationship between the flow variables in the two frames, in a cylindrical coordinate system, where the prime denotes a quantity in the rotating frame:

\begin{align*}
  r &= r', \\
  z &= z', \\
  \theta &= \theta' + \Omega t, \\
  t &= t'.
\end{align*}

The time derivative term is, then:

$$\frac{\partial \mathbf{u}}{\partial t} = \frac{\partial \mathbf{u}}{\partial \theta'} \frac{\partial \theta'}{\partial t} + \frac{\partial \mathbf{u}}{\partial t'} \frac{\partial t'}{\partial t}, \quad (A.4)$$

which reduces to:

$$\left( \frac{\partial \mathbf{u}}{\partial t} \right)_{\text{inert}} = \left( \frac{\partial \mathbf{u}}{\partial t} \right)_{\text{rot}} - \Omega \frac{\partial \mathbf{u}}{\partial \theta'}. \quad (A.5)$$

When the advection term is expanded, the following expression is obtained:

$$\mathbf{v} \cdot \nabla \mathbf{v} = \mathbf{u} \cdot \nabla \mathbf{u} + 2\Omega \times \mathbf{u} + \Omega \times (\Omega \times \mathbf{r}) + \nabla \times (\mathbf{u} \times (\Omega \times \mathbf{r})). \quad (A.6)$$

In their paper, Lopez et al. [11] show that, for a constant angular velocity, the last term of the equation above becomes:

$$\nabla \times (\mathbf{u} \times (\Omega \times \mathbf{r})) = \Omega \frac{\partial \mathbf{u}}{\partial \theta'}, \quad (A.7)$$

where the prime has been dropped from $\theta$, which now represents the azimuthal coordinate in the rotating frame. Finally, considering a general, non-constant density $\rho$, the momentum equations in a rotating frame become:

$$\rho \left( \frac{\partial \mathbf{u}}{\partial t} - \Omega \frac{\partial \mathbf{u}}{\partial \theta} \right) + \rho \left( \mathbf{u} \cdot \nabla \mathbf{u} + 2\Omega \times \mathbf{u} + \Omega \times (\Omega \times \mathbf{r}) + \Omega \frac{\partial \mathbf{u}}{\partial \theta} \right) = -\nabla p + \mu \nabla^2 \mathbf{u}. \quad (A.8)$$
Equation A.8 should now be compared to Eq. A.1. If the densities on the left-hand side of Eq. A.8 are the same, then the terms $\Omega \frac{\partial u}{\partial \theta}$ cancel out, and one obtains the ‘classical’ form of the momentum equations in a rotating frame. In the formulation proposed in ref. [11] (Eq. A.3), the densities are not the same, since $\rho = \rho_0 + \rho'$ for the terms within the second bracket of Eq. A.8. Therefore, an extra term, $\rho' \Omega \frac{\partial u}{\partial \theta}$, is retained, but it would be absent if a variable density was considered in the time-derivative term. For this reason, the extended approximation presents an inconsistency, and in this thesis the standard, or traditional, Boussinesq approximation was employed, Eq. A.2.

Note that the extra term in the approximation is proportional to the derivative of the velocity field with respect to $\theta$. Thus, if the flow is axisymmetric, or nearly-axisymmetric, then the standard and extended approximations will give similar, if not equal, results. It is when non-axisymmetry becomes important that the two approximations differ significantly. Although the extra term is multiplied by $\rho'$, which is a small quantity, it is also proportional to $\Omega$, which makes it non-negligible in comparison with other terms of the governing equations proportional to $\frac{\partial u}{\partial \theta}$.

To illustrate the difference between the two formulations with a practical example, consider buoyancy-induced flow in a sealed rotating annulus at $Ra = 10^8$, as reported in Chapters 5 and 6. Figure A.1 shows that the two approximations yield dramatically different results. During the preparation of this thesis, a colleague at the University of Surrey has performed the same calculation using the compressible flow solver Hydra, which uses the finite volume method. Figure A.1(c) shows a snapshot obtained from that calculation, received via private communication with Dr. Feng Gao. Clearly, the contours obtained with the traditional approximation (Fig. A.1(a)) and with the compressible solver (Fig. A.1(c)) are in excellent agreement, which shows the suitability of the approximation used. The only clear difference between the contours is that in Fig. A.1(a) the solution is dominated by an azimuthal wavenumber $k = 5$, whereas in Fig. A.1(c) $k = 4$. This, however, is not surprising, since the dominant wavenumber may change in time, as observed in many of the calculations performed during this research.
Figure A.1: Snapshots of instantaneous normalised temperature $T^*$, ranging from 0 to 1, at the mid-axial plane of a sealed rotating annulus with $Ra = 10^8$. The contour obtained with the traditional Boussinesq approximation, which was used throughout this research, is compared with those of the extended Boussinesq approximation and with a fully compressible formulation.
Appendix B

Natural convection in a differentially heated tall enclosure

In the early stages of this research, a study of natural convection in a differentially-heated tall enclosure was performed. The results were compared with benchmark data from the literature and with results obtained with the commercial code ANSYS Fluent. The results from this investigation were published in a conference paper [119], and are reproduced in this appendix with minor adaptations.

Introduction

Natural convection flows occur in a large range of natural phenomena and in engineering applications. This phenomenon has been investigated extensively both numerically and experimentally over the past decades, usually adopting canonical flow configurations with simplified geometries and boundary conditions, such as rectangular cavities heated from the side or heated from below. The latter configuration becomes unstable at a relatively low Rayleigh number, since the direction of the heat flux is opposed to that of gravity. Buoyant flows in differentially-heated cavities, on the other hand, become unstable at much higher values of the Rayleigh number. For low aspect ratios, the instabilities depend on the boundary conditions imposed on the horizontal walls, but when the aspect ratio is greater than or equal 4, the transition to an unstable state is initiated on the vertical boundary layers, independently of the boundary conditions imposed on the horizontal walls [136]. In differentially-heated tall cavities, the flow is governed by the aspect ratio, by the Prandtl number and by the Rayleigh number based on the cavity height. To obtain accurate solutions for turbulent natural convection flows it is necessary to conduct unsteady, three-dimensional numerical simulations of the Navier-Stokes equations. The use of direct numerical simulation (DNS) to study turbulent natural convection flows is important to investigate instability mechanisms and to improve turbulence modelling.

Xin & Le Quéré [136] carried out unsteady two-dimensional numerical simulations for natural convection in a differentially heated cavity of aspect ratio 4, using a
spectral method employing Chebyshev polynomials as expansion bases and a second-order semi-implicit time integration scheme. This flow configuration is known to become unstable for $Ra > 10^8$, so the authors studied the cases $Ra = 6.4 \cdot 10^8$, $Ra = 2.0 \cdot 10^9$ and $Ra = 10^{10}$. The calculations revealed that the vertical boundary layers remain laminar over most of the cavity height, even for the largest value of $Ra$ studied. For the lowest value of $Ra$ studied, the flow is weakly turbulent, with the instabilities concentrated in the downstream part of the boundary layers; the cavity core remains stratified, as would occur in a laminar solution. As the Rayleigh number is increased, the flow in the core of the cavity becomes unsteady and it is no longer stratified for $Ra = 10^{10}$. In a more recent work, Trias et al. [120] performed two- and three-dimensional numerical simulations for this problem, considering the flow to be periodic in the third direction. These authors used second- and fourth-order spectro-consistent spatial discretisation and a second-order time integration scheme. The results are in general good agreement with those reported by [136], but it was observed that in the two-dimensional cases the fluctuation levels are over-predicted in comparison with the three-dimensional simulations. This occurs because the third coordinate allows the energy to be dissipated to the smallest scales, whilst in 2D the energy is transferred to the large scales. The three-dimensional calculations revealed that even for $Ra = 10^{10}$ the instabilities are concentrated at the lower and upper parts of the cavity, with the core remaining stratified. Trias et al. [137, 138] observed that significant changes in the flow structure occur for $Ra > 10^{10}$ as the transition point moves upstream the boundary layers.

Ghaisas et al. [139] evaluated the performance of various large-eddy simulation (LES) techniques to solve the problem of turbulent natural convection in a differentially-heated tall cavity. Excellent agreement was obtained for the velocity and temperature fields in comparison with the DNS results from [120]. The second-order statistics, on the other hand, were not very accurately predicted by any of the LES models. LES models are desirable since they have the potential to provide accurate solutions with a much lower computational cost when compared to DNS. DNS solutions are necessary, though, to provide results for a thorough validation of LES models, especially in terms of high-order statistics.

In this work the code Semtex [104], which employs a spectral element method, is used to conduct direct numerical simulations of turbulent natural convection in a tall differentially-heated enclosure. The flow is solved in a three-dimensional, periodic domain for $Ra = 2.0 \cdot 10^9$. The solutions are compared against existing DNS data and with results obtained with the commercial, low-order finite volume code ANSYS Fluent. The fact that the flow is periodic in one of the directions allows the use of Fourier expansions, which provides a favourable framework for parallelisation.

**Problem description**

The geometry and thermal boundary conditions used to solve the problem are shown schematically in Fig. B.1 (a no-slip condition is used for velocity on all walls). The flow is governed by the aspect ratio $H/L$, which is 4 in this study, by the Prandtl number and by the Rayleigh number based on the cavity height,

$$Pr = \frac{\nu}{\kappa} = 0.7,$$

(B.1)
\[ Ra = \frac{g \alpha \Delta T H^3}{\nu \kappa} = 2 \cdot 10^9, \] (B.2)

where \( \nu \) is the kinematic viscosity, \( \kappa \) is the thermal diffusivity, \( \alpha \) is the thermal expansion coefficient, \( g \) is the acceleration of gravity and \( \Delta T \) is the temperature difference between the hot and cold walls. The flow domain is three-dimensional with periodicity imposed in the third, normal direction, with a periodicity length \( L_z = L \). The time and velocity scales for the problem are \( t_s = \left( \frac{H^2}{\alpha} \right) Ra^{-0.5} \) and \( U_s = \left( \frac{\alpha}{H} \right) Ra^{0.5} \).

Figure B.1: Geometry and thermal boundary conditions for the differentially-heated cavity. The aspect ratio is \( H/L = 4 \) and the domain is periodic in the normal direction.

The governing equations are solved in incompressible form using the Boussinesq approximation to account for changes in density only in the term that is multiplied by the acceleration of gravity.

The mesh used in this study has 17 spectral elements in \( x \) and 29 elements in \( y \), and the polynomial order was fixed at \( P = 9 \). 64 planes, corresponding to 32 Fourier modes, are used in the periodic direction, which gives a total of approximately 3.1 million degrees of freedom. Grid refinement studies were carried out and it was concluded that this mesh is sufficient to obtain accurate results for \( Ra = 2.0 \cdot 10^9 \). For comparison purposes, a mesh with similar number of degrees of freedom was used to solve the problem using the commercial software ANSYS Fluent. The non-dimensional time step used in \textit{Semtex}, normalised with the flow time scale, is \( \Delta t = 1.79 \cdot 10^{-3} \), and in Fluent the calculations were carried with \( \Delta t = 4.47 \). This great difference in the values of the time step between the codes is due to the fact that \textit{Semtex} uses a semi-implicit time scheme, whilst a fully implicit scheme is used.
in Fluent. A time step 10 times smaller was tested in Fluent but no significant difference was observed in the statistics.

**Results and discussion**

The solution was initiated from a pure-conduction temperature profile and zero velocity. In *Semtex* a random perturbation was imposed to obtain transition to the unstable state. Transition to the final state was monitored by analysing the variation of the Nusselt number on the hot wall with calculation time. After the value of $Nu$ started to oscillate around a constant value, statistics were collected until a statistically steady state was obtained. Figure B.2 shows the distribution of the modal energy $E(k)$ as a function of the Fourier mode $k$, normalised with the energy of the first mode, not shown in the graph as it is much larger than the energy for the rest of the modes. It is noted that the energy decays more than four orders of magnitude between the second and the last modes, indicating that a large range of scales is resolved. The lowest modes contain most of the energy of the flow, whilst the highest modes contain little energy and are responsible for energy dissipation.

![Figure B.2: Distribution of the modal energies as a function of the Fourier mode $k$.](image)

Instantaneous temperature contours obtained with the spectral element code are illustrated in Fig. B.1(b), where the general behaviour of the flow can be understood. Hot fluid is transported towards the upper part of the cavity along the hot vertical wall. The boundary layer is laminar in most of the vertical length, and becomes unstable in the downstream part, near the top of the cavity, where strong mixing occurs. On the right-hand side, on the cold wall, a similar behaviour is observed, but with cold fluid moving downwards in a laminar boundary layer which only becomes unstable in the downstream part, i.e., in the bottom part of the cavity. Note that the core of the cavity remains stratified, in agreement with the results reported by Trias et al. [120]. Figure B.3 shows iso-surfaces of the $Q$ criterion, where it is noted that small-scale structures are formed downstream in the boundary layers and that the regions of turbulent flow are located in the top and bottom of the cavity.
Figure B.3: Iso-surfaces of the $Q$ criterion for $Ra = 2 \cdot 10^9$.

Figure B.4 shows profiles for the horizontal velocity and for the $v'v'$ component of the Reynolds stress tensor at the mid vertical position, $y^* = y/H = 0.5$ obtained with both codes, where $v$ denotes the vertical velocity component. These results were obtained averaging the flow field both in time and along the periodic direction, and are normalised with $U_s$ and $U_s^2$, respectively. The horizontal velocity profile reveals that there is no motion in the horizontal direction in the cavity core, with the horizontal velocity having a small component in the boundary layers only. Note that $v'v'$ has the same order of magnitude as the horizontal velocity, indicating that at this location the vertical motion dominates the flow structure. The vertical velocity at this location, not shown here, is approximately three orders of magnitude larger than the horizontal velocity. For this location both Semtex and Fluent give very good predictions for the horizontal velocity component in comparison with the DNS results from [120]. The turbulent fluctuation $v'v'$, on the other hand, is underpredicted by Fluent. As expected, the fluctuation is much higher near the walls than in the cavity core.

Figure B.5 depicts the same profiles at $y^* = 0.9$, in the upper part of the cavity. At this location, which exhibits stronger fluctuations compared to the cavity mid-height, Fluent gives poor predictions for both the horizontal velocity and for the turbulent fluctuation of the vertical velocity, whilst the results obtained with Semtex agree well with existing DNS results. It should be noted that at this location the horizontal component is much greater than at $y^* = 0.5$, since the hot fluid is transported horizontally in the upper part of the cavity. In agreement with the qualitative observations made based on the temperature contours of Fig. B.1, it is noted that $v'v'$ is much larger near the hot wall than near the cold wall, indicating that more mixing occurs near the downstream part of the boundary layers. An analogous effect occurs in the bottom part of the cavity, where the turbulent intensities are greater near the cold wall. This behaviour is contrasted with that observed at $y^* = 0.5$, where the level of turbulence near both walls is similar. The predictions for the vertical velocity and for the temperature profiles obtained with Semtex and Fluent are in general good agreement with previous results. The other second-order statistics, in contrast, are only predicted correctly using Semtex, whilst
Fluent generally under-estimates these quantities significantly on the present mesh.

The mean Nusselt number on the vertical walls is very well predicted by both codes. [120] reported a value of $Nu = 66.63$, whilst $Nu = 66.59$ and $Nu = 65.64$ were obtained with Semtex and Fluent, respectively, giving relative errors of 0.06% and 1.49%, respectively. In general it is observed that Fluent can give accurate predictions for global features of the flow, such as velocity profiles and heat transfer rates, but it fails to give accurate predictions for second-order statistics. The computing time per time step is almost 50% smaller with the spectral element code than with Fluent. It should be noted that the calculations performed with Semtex took much longer since the time step is much smaller when compared with Fluent, requiring a higher number of time steps.
Conclusion

In this work the performance of two codes was compared by performing numerical simulations of turbulent natural convection in a differentially-heated cavity on meshes with similar numbers of degrees of freedom. Both codes provided very good predictions for the Nusselt number and for velocity and temperature profiles at the cavity mid-height. In a region in the upper part of the cavity, however, the results obtained with Fluent for the horizontal velocity profile did not agree well with previous results. The turbulent fluctuation of the vertical velocity was well predicted by the spectral element code in both regions analysed, whilst Fluent gave very poor predictions for the second-order statistics. It is noted that a similar number of degrees of freedom was used in both codes, which shows that the spectral element code requires less resolution for a good level of accuracy when compared with Fluent. Finally, it is concluded that Semtex is an attractive tool to carry out fundamental investigations of turbulent flow. The fact that the code employs the spectral element method allows the use of complex geometries and unstructured meshes, as opposed with global spectral methods, which are useful only when simple geometries are used.
Bibliography


