F. Xavier Trias  
Assensi Oliva Llena

Directores de proyecto
Terrassa, Junio 2013
Abstract

The main aim of this work is understanding the phenomenon of Bénard cells in laminar and turbulent regimes as an application of studying the air flow in the air gap and honeycomb cells that installed in the flat plate solar collector. Furthermore analyzing the turbulent Rayleigh-Bénard convection components by performing direct numerical simulation of turbulent air flow in too long inclined cavity like the air gap. The first two chapters treat with known problems of laminar and turbulent flow as a necessary presentation of numerical methods used in solving the governing equation of flow motion and heat transfer, afterward the third chapter deals with the main object of investigating the turbulent and laminar Rayleigh-Bénard convection flow.


Contents

1 Driven-cavity and backward-facing step problems 7
  1.1 Driven-cavity problem ............................................. 7
    1.1.1 Introduction ................................................. 7
    1.1.2 Governing equations and problem parameters ..................... 7
    1.1.3 Numerical method ............................................ 8
    1.1.4 Results and comparisons ..................................... 9
    1.1.5 Conclusions ................................................. 13
  1.2 Backward-facing step problem ................................. 15
    1.2.1 Introduction ................................................. 15
    1.2.2 Governing equations and problem parameters ..................... 15
    1.2.3 Numerical method ............................................ 16
    1.2.4 Results and Comparisons .................................... 17
    1.2.5 Conclusions ................................................. 19

2 Direct numerical simulation of Turbulent and laminar air flow in differentially heated cavity 21
  2.1 Differentially heated cavity problem ........................... 21
    2.1.1 Introduction ................................................. 21
    2.1.2 Governing equations and problem parameters ..................... 21
    2.1.3 Numerical method ............................................ 22
    2.1.4 Results and comparisons ..................................... 24
    2.1.5 Conclusions ................................................. 29
  2.2 Direct numerical simulation of 2-D turbulent flow in a differentially heated cavity of aspect ratio 4 ......................................................... 30
    2.2.1 Introduction ................................................. 30
    2.2.2 Governing equations and problem parameters ..................... 31
    2.2.3 Numerical method ............................................ 32
    2.2.4 Results and discussion ....................................... 36
    2.2.5 Conclusions ................................................. 45

3 Direct numerical simulation of laminar and turbulent Rayleigh-Bénard convection 47
  3.1 Introduction .................................................. 48
  3.2 Problem Parameters ........................................... 49
  3.3 Governing equations and Numerical methods ..................... 49
    3.3.1 Governing equation ....................................... 49
    3.3.2 Numerical methods ....................................... 50
  3.4 Comparisons ................................................ 51
  3.5 Results and Discussion ....................................... 52
  3.6 Turbulent Rayleigh-Bénard convection flow in very large aspect ratio cavity ........ 61
    3.6.1 Resolution requirements ................................... 62
### Contents

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.6.2 Temporal analysis</td>
<td>65</td>
</tr>
<tr>
<td>3.6.3 Averaged Turbulent components</td>
<td>67</td>
</tr>
<tr>
<td>3.7 Temperature and thermal boundary layer</td>
<td>71</td>
</tr>
<tr>
<td>3.8 Thermal dissipation rate and thermal plumes</td>
<td>73</td>
</tr>
<tr>
<td>3.9 Conclusions</td>
<td>79</td>
</tr>
<tr>
<td>4 Conclusions</td>
<td>81</td>
</tr>
<tr>
<td>A Turbulent three dimensional Rayleigh-Bénard convection</td>
<td>87</td>
</tr>
<tr>
<td>B MPI parallel implementation for solving large linear systems</td>
<td>93</td>
</tr>
<tr>
<td>B.1 Introduction</td>
<td>93</td>
</tr>
<tr>
<td>B.2 problem description and Domain decomposition</td>
<td>94</td>
</tr>
<tr>
<td>B.3 Domain decomposition</td>
<td>95</td>
</tr>
</tbody>
</table>
Chapter 1

Driven-cavity and backward-facing step problems

1.1 Driven-cavity problem

1.1.1 Introduction

The well-known driven cavity problem can be described as a 2-D square cavity with no-slip conditions at the lateral and bottom walls and forced superficial velocity flow at the top wall, without heat transfer through the surfaces. This problem is resolved numerically by performing direct numerical simulation of the incompressible Navier-Stokes equations that govern the flow motion (momentum equations) and apply the fractional step (projection) method as an explicit algorithm that uses Helmholtz-Hodge decomposition theorem. Finite volume method is used to discretize the domain and fully explicit second-order Adams-Bashforth scheme is used as a time integration method. High order numerical scheme is utilized in evaluating the convective term of Navier-Stokes equations, and different grids are used at various \( Re \) in order to compare with the Benchmark results.

1.1.2 Governing equations and problem parameters

The non-dimensional form of Navier-Stokes and the continuity equations for incompressible Newtonian fluid considering the effects of the body forces are neglected, are given like:

\[
\nabla \cdot \mathbf{u} = 0 \tag{1.1}
\]

\[
\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla)\mathbf{u} = \frac{1}{Re} \nabla^2 \mathbf{u} - \nabla p \tag{1.2}
\]

Where \( Re = \rho V_o L/\mu \) is the Reynolds number, \( \rho \) and \( \mu \) are the density and the dynamic viscosity of the working fluid, \( L \) and \( V_o \) are the characteristic length and velocity respectively. \( \mathbf{u}, t \) and \( p \) are the dimensionless velocity vector, time and static pressure respectively with reference values are, \( V_o \) for the velocity, \( L/V_o \) for the time, \( \rho V_o^2 \) for the static pressure and \( L \) for lengths. The no-slip boundary conditions are imposed for the velocity at all the walls except the top one.

The problem is defined as a 2-D square cavity with length equals to 1 and a the velocity vector at the top side \( \mathbf{u} = \begin{bmatrix} 1 \\ 0 \end{bmatrix} \) in the non-dimensional form and Reynolds number changes \( Re = 100, 400, 1000, 3200, 5000, 7500, 10000 \).
1.1.3 Numerical method

Spatial discretization and time integration method We discretise the domain using the finite volume method and creating a uniform structured mesh with Cartesian coordinates \((x,y)\). The differential governing equations (eq. 1.1.2) are integrated spatially in a staggered mesh where the velocities are evaluated in the surfaces and the pressure in the nodes. The staggered grid can avoid the unrealistic solution of the pressure (checkerboard pressure distribution) that satisfies the momentum equations, as well the velocity field resolution that satisfies the continuity equation and likewise causes unrealistic resolution field, these unrealistic resolution can be found using the collocated grid. In the staggered grid, the mass flow rate across the control volume faces can be calculated without any interpolation for the relevant velocity component and the discretized continuity equation would contain the difference of adjacent velocity components preventing the wavy velocity field, on other hand the pressure oscillation will be canceled as a result of, the pressure difference between two adjacent grid points becomes the natural driving force for the velocity component located between these grid points consequently the felt of uniform pressure fields will be canceled [3].

In order to simplify the notation, momentum equation (1.2) can be rewritten as

\[
\frac{\partial u}{\partial t} = R(u) - \nabla p
\] (1.3)

Where \(R(u)\) represents the convective and diffusive terms. For the temporal discretization, a central difference scheme is used for the time derivative terms for \(u\), and a fully explicit second-order Adams-Bashforth scheme is used for \(R(u)\). A first-order backward Euler scheme is used for the pressure-gradient term. Incompressibility constraint is treated implicit. Thus, we obtain the semi-discretized Navier-Stokes equations

\[
\frac{u^{n+1} - u^n}{\Delta t} = \frac{3}{2}R(u^n) - \frac{1}{2}R(u^{n-1}) - \nabla p^{n+1}
\] (1.4)

\[
\nabla \cdot u^{n+1} = 0
\] (1.5)

To solve the velocity-pressure coupling we use a classical Fractional step projection method (see [4] and [5]), in these method solutions of the unsteady Navier-Stokes equations are obtained by first time-advancing the velocity field \(u\) without regard for its solenoidality constraint (eq. 1.5), then recovering the proper solenoidal velocity field, \(u^{n+1}\) after implying the continuity constraint in the Poisson equation, this projection method is derived from the well-known Helmholtz-Hodge vector decomposition theorem (see [5]), whereby the predictor velocity \(u^p\) can be uniquely decomposed into a divergence-free vector \(u^{n+1}\), and the gradient of a scalar field \(\nabla p^\sim\), this decomposition is written as

\[
u^p = u^{n+1} + \nabla p^\sim
\] (1.6)

Where the predictor velocity \(u^p\) is

\[
u^p = u^n + \Delta t \left(\frac{3}{2}R(u^n) - \frac{1}{2}R(u^{n-1})\right)
\] (1.7)

and the pseudo-pressure is \(p^\sim = \Delta t p^{n+1}\). Taking the divergence of (eq. 1.6) yields to Poisson equation like

\[
\nabla \cdot u^p = \nabla \cdot u^{n+1} + \nabla \cdot (\nabla p^\sim) \rightarrow \nabla^2 p^\sim = \nabla \cdot u^p \rightarrow \nabla^2 p^{n+1} = \frac{1}{\Delta t} (\nabla \cdot u^p)
\] (1.8)

Once the solution is obtained, \(u^{n+1}\) results from the correction

\[
u^{n+1} = u^p - \nabla p^\sim
\] (1.9)
1.1. Driven-cavity problem

The Poisson equation (eq. 1.8) has the form $A \cdot x = b$, we solve it linearly by applying LU decomposition for the matrix $A$ to upper and lower matrices or Band-LU that saves the matrix $A$ in compact form as $A$ is non-zero band diagonal coefficients matrix, we can use also the TDMA-Gauss iterative method (suppose values ⇒ calculate them line by line with TDMA from n→ s and then from e→ w ⇒ compare the calculated values with those have been supposed ⇒ resolving till the converging).

For evaluating the time step $\Delta t$, and returning to stability reasons of the explicit temporal schemes that have been used, $\Delta t$ must be bounded by the CFL condition which given like:

\[
\Delta t \left( \frac{|u_i|}{\Delta n_i} \right)_{max} \leq C_{conv} \rightarrow \Delta t \leq \frac{C_{conv}}{\left( \frac{|u_i|}{\Delta n_i} \right)_{max}}
\]

(1.10)

\[
\Delta t \left( \frac{|\nu_i|}{\Delta n_i^2} \right)_{max} \leq C_{visc} \rightarrow \Delta t \leq \frac{C_{visc}}{\left( \frac{|\nu_i|}{\Delta n_i^2} \right)_{max}}
\]

(1.11)

Where $n_i = \left[ x_i \ y_i \right]$ and the bounding values $C_{conv}, C_{visc}$ must be smaller than unity, we will follow the recommendations using values $C_{conv} = 0.35$, $C_{visc} = 0.2$, and $\nu_i$ is the viscosity that here the non-dimensional coefficient of the diffusive term $1/Re$. As we mentioned in evaluating $R(u)$ in equation 1.7, we need to evaluate the velocities at the faces. Different numerical schemes can be used, the most popular ones are Upwind scheme as a first-order scheme, its relation is defined as

\[
\phi_e = \phi_P \quad if \quad u_e > 0
\]

(1.12)

\[
\phi_e = \phi_E \quad if \quad u_e < 0
\]

(1.13)

Where $\phi_e$ is the unknown variable at the face, $\phi_P$ is the variable at the nearest grid point on the upwind side of the face, $\phi_E$ the variable at the nearest grid point on the downstream side of the face, $u_e$ the velocity at the face (see figure 1.1).

The central difference first-order numerical scheme also will be a good option in quadratic discretization

\[
\phi_e = \frac{\phi_P + \phi_E}{2}
\]

(1.14)

And the Quick scheme as a higher differencing second-order numerical scheme, that uses three-point upstream quadratic interpolation.

\[
\phi_e = \frac{\phi_P + \phi_E - \phi_W}{8} \quad if \quad u_e > 0
\]

(1.15)

\[
\phi_e = \frac{\phi_P + \phi_E - 2\phi_W}{8} \quad if \quad u_e < 0
\]

(1.16)

Where $\phi_W$ is the variable at the nearest grid point on the upstream side of $\phi_P$, and $\phi_EE$ is the variable at the nearest grid point on the downstream side of $\phi_E$.

1.1.4 Results and comparisons

Using a high mesh of 100x100 on staggered grid, and implying the Quick numerical scheme we get the graphics of the stream line in the steady state in figures (1.2, 1.3, 1.3, 1.4, 1.5) where it can be noted that once the Reynolds number is being high, vortexes are appearing at the edges.
Chapter 1. Driven-cavity and backward-facing step problems

**Figure 1.1:** numerical Quick scheme.

**Figure 1.2:** maps of the streamlines in the cavity at $Re = 100, Re = 400$.

**Figure 1.3:** maps of the streamlines in the cavity at $Re = 1000, Re = 3200$. 
1.1. Driven-cavity problem

Figure 1.4: maps of the streamlines in the cavity at $Re = 5000, Re = 7500$.

Figure 1.5: maps of the streamlines in the cavity at $Re = 10000$.

Now for $Re = 10^2$ if we use a mesh 40x40 and compare with the Benchmark results \[52\] along vertical line through geometric center of the cavity for $u$, and along the horizontal line through the geometric center of the cavity for $v$, and do the same in mesh 100x100 we see that there is no big difference and the both results agree well the Benchmark results (see figures 1.6, 1.7).
Chapter 1. Driven-cavity and backward-facing step problems

Now when the Reynolds number increases to $Re = 10^3$ and doing the same before we can see that the mesh 40x40 gives results so far of Benchmark, and comparing with the mesh 100x100 we find that its conforming well the Benchmark results (see figures 1.8, 1.9).
1.1. Driven-cavity problem

Figure 1.8: Values of $u$ along the vertical middle line of the cavity at $Re=1000$.

Figure 1.9: Values of $v$ along the horizontal middle line of the cavity at $Re=1000$.

Now if we use the mesh $100\times100$ with Reynolds number $Re = 5000$ and compare with Benchmark we can note a little deviation that can be reduced by using a mesh more smooth like $160\times160$.(see figures 1.10, 1.11).

1.1.5 Conclusions

A good agreement is found between our presented results and the Benchmark ones [52], and that verifies the good efficiency of the followed methods in this simulation. From the sight of the examined cases of this problem, we can conclude that when the Reynolds number is getting increase, it needs mesh more smooth in order to close the Benchmark results and
Chapter 1. Driven-cavity and backward-facing step problems

Figure 1.10: Values of $u$ along the vertical middle line of the cavity at $Re=5000$.

Figure 1.11: Values of $v$ along the horizontal middle line of the cavity at $Re=5000$.

that is logic because when the Reynolds is being higher vortexes will appear that need mesh so smooth and sufficient time to be steady.
The relation between the mesh and the time step we can find it in CFL condition that shows, when the mesh is more fine the time step is more small and gives the vortexes the sufficient time to appear and be steady.
1.2 Backward-facing step problem

1.2.1 Introduction

The phenomenon of separation in the flow that returned to exists an adverse pressure gradient has been the subject of intensive study for many years. This adverse pressure gradient imposes on the boundary layer by the outer potential flow and causes a reversal flow characterized as having vortices, has a considerable impact on the flow structure.

Experimental investigations into the backward-facing step problem by Armaly et al. [1] have shed light on some major flow features at the sudden expansion in a channel, and followed by attempts to understanding this process in small scales using the numerical simulation and the computational power. In this work two dimensional numerical study of the backward-facing step flow problem in the case of laminar flow where $100 \leq Re \leq 1000$, has done by solving the non-dimensional Navier-Stokes equations of incompressible Newtonian flow in 2-D Cartesian coordinates, and find the approximated solutions using the Fractional step methods. We utilize a refined mesh for purposes of saving time and memory with proportional accuracy and that by allocating more nodes at the two channel walls and the step surfaces, than other regions. We use the step geometry and flow conditions reported by Armaly et al. [1], in purpose of direct comparison with the physical experiment streamwise velocity profiles of Armaly et al, at $z = 0$ symmetric plane for cases of $Re = 100$, $Re = 389$ and $Re = 1000$, and with Guerrero and Cotta [2] two dimensional streamwise velocity profiles in the case of $Re = 1000$. Schematically predictions of the streamlines flow in the channel and over the step have been presented for all studied cases $Re = 100, Re = 389, Re = 800, Re = 1000$, the lengths of the separation $x_4$ and reattachment flow $x_1, x_5$, see figure 1.12, have been compared with 2-D Gresho et al. [6] results in case of $Re = 800$, and with 2-D, T. P. Chiang and Tony W. H. Sheu [7] results in the case of $Re = 1000$. We find an acceptable agreement between our results and the references with a small ratio of difference, which seeks to the good advantage of using 2-D numerical simulation in predicting 3-D flow at the symmetric plane and mimicking the physical experiments using less computational memory.

1.2.2 Governing equations and problem parameters

The non-dimensional form of Navier-Stokes and the continuity equations for incompressible Newtonian fluid considering the effects of the body forces are neglected, are given like:

$$\nabla \cdot \mathbf{u} = 0$$

$$\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla) \mathbf{u} = \frac{1}{Re^*} \nabla^2 \mathbf{u} - \nabla p$$

(1.17) (1.18)

in the normalization, the mean velocity at the inlet plane $U_{mean} \equiv 1$ and $\rho U_{mean}^2$ are chosen as the reference quantities for the velocity and pressure respectively, and the upstream channel height $h \equiv 1$ and $h/U_{mean}$ as references for dimensions and time respectively. The Reynolds number is defined as: $Re^* = \rho U_{mean} h/\mu$ where: $\rho$ the density of the fluid, $\mu$ the dynamic viscosity of the fluid.

our Reynolds number $Re^*$ is different from Reynolds number used by Armaly et al. [1] which equals $Re = \rho U_{mean} (2h)/\mu$, this means $Re = 2 Re^*$ for comparison purposes.

the solution of the momentum Navier-Stokes equations is bounded by the inlet and outlet flow through the channel, and by the solid walls, at the upstream flow (before the sudden expansion) we suppose that fully developed flow prevails, therefore we define a fully developed
velocity profile yields a mean velocity $U_{\text{mean}} \equiv 1$ at the inlet plane as:

$$U_{\text{inlet}}(y) = \frac{48}{\alpha \pi^3} \beta(y)$$

$$\alpha = 1 - \frac{192B}{\pi^5} \sum_{i=1,3,5,\cdots}^\infty \frac{\tanh(\xi(i))}{i^3}$$

$$\beta(y) = \sum_{i=1,3,5,\cdots}^\infty (-1)^{(i-1)/2} \left(1 - \frac{\cosh((2y-1)\xi(i))}{\cosh(\xi(i))}\right) \frac{1}{i^3}$$

$$\xi(i) = \frac{\pi i}{2B}$$

where: $B = 35h$ and at the outflow boundary, the condition of giving zero gradients for field variables can be assumed, at the roof and at the floor of the channel, no-slip boundary conditions are prescribed to close the problem.

The geometry of the problem and analyzed Reynolds numbers have been presented here for mimicking Armaly [1] experiment conditions, therefore we attach a straight channel with length of $10h$ upstream of the step plane to consider the influence of the upstream channel on the flow development at the downstream, the step has an expansion ratio $\gamma = H/h = 1.9423$ where $H = h + S$ -see figure 1.12- and the outlet is set $55h$ downstream from the step where the flow will again be parabolic. The Reynolds numbers that have been tested here for verification purpose are $Re = 100$, $Re = 389$, $Re = 800$, $Re = 1000$ (or $Re^* = 50$, $Re^* = 194.5$, $Re^* = 400$, $Re^* = 500$), it’s important to mention that these values are corresponding to the laminar flow region where $Re < 1200$ (by Armaly [1]).

**Figure 1.12:** Backward-facing step channel geometry with the definition of separation length $x_4$ and the reattachment lengths $x_1$, $x_5$.

### 1.2.3 Numerical method

The same procedures that used in section 1.1.3 have been implemented here where the Quick numerical scheme has been used. A refined grid using hyperbolic tangent functions is used for cluster more control volumes at the walls of the group, we have to pay attention to the ratio of $\Delta x/\Delta y$ in the expansion region where there is an oblique flow. At the edge of the step, the flow changes his direction suddenly so a small ratio near to one should be considered to reduce the effect of the false diffusion.
1.2.4 Results and Comparisons

Using a number of nodes 100\(\times\)90 with a refinement at the step, the roof and the floor channel walls, see figure 1.13, we plots the streamwise velocity profiles along the channel, comparing with the measured results taken from 3-D experiments of Armaly et al. [1] at the symmetric plane where the span width is 35\(h\), for Reynolds numbers \(Re = 100\), \(Re = 389\), see figures 1.14.a, 1.14.b, and \(Re = 1000\) shown in figure 1.15. We can see a good agreement between our 2-D pattern results and the 3-D symmetric plane pattern experimental results except in the case of \(Re = 1000\), evidence that there are notable differences especially in distances of the range between 24 < \(x\) < 30.

Now we compare our 2-D streamwise velocity profile results with 2-D results obtained by Guerrero and Cotta [2] for \(Re = 1000\), shown in figure 1.16, we can note the good agreement between the both.

We present in the following figure (Fig.1.17) schematically expressions of our predicted streamlines for \(Re = 100\), \(Re = 389\), \(Re = 800\) and \(Re = 1000\).we compare our separation and reattachment lengths with 2-D results of Gresho et al. [6] for the case of \(Re = 800\) and \(S = 1.0\), and with the 2-D results of T. P. Chiang and Tony W. H. Sheu [7] for the case of \(Re = 1000\) and \(S = 0.9423\), see table 1.1, we can note an acceptable agreement between the results.

<table>
<thead>
<tr>
<th>2-D, (Re=800), (S = 1.0)</th>
<th>(x_1)</th>
<th>(x_4)</th>
<th>(x_5)</th>
<th>(x_5 - x_4)</th>
</tr>
</thead>
<tbody>
<tr>
<td>2-D, present</td>
<td>11.93</td>
<td>9.43</td>
<td>20.98</td>
<td>11.55</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>2-D, (Re=1000), (S = 0.9423)</th>
<th>(x_1)</th>
<th>(x_4)</th>
<th>(x_5)</th>
<th>(x_5 - x_4)</th>
</tr>
</thead>
<tbody>
<tr>
<td>2-D, present</td>
<td>12.6</td>
<td>9.92</td>
<td>23.48</td>
<td>13.56</td>
</tr>
</tbody>
</table>

Table 1.1: A summary of the lengths \(x_1\), \(x_4\), and \(x_5\).
Chapter 1. Driven-cavity and backward-facing step problems

Figure 1.14: Streamwise velocity profiles at the $z = 0$ symmetric plane for the cases of (a) $Re=100$; (b) $Re=389$.

Figure 1.15: Streamwise velocity profiles at the $z = 0$ symmetric plane for the case of $Re=1000$. 


1.2.5 Conclusions

From the two-dimensional results that we have been presented in our job, we can see the good ability that gives us this numerical simulation to predict the development of the flow over the backward-facing step in a channel at the symmetric plane of 3-D flow, where we have noted the good agreement with the experimental measurements for cases of low Reynolds numbers $Re = 100$ and $Re = 389$, therefore we can conclude that the end wall effect (in $z$ direction) on the flow at the mid plane is negligible for this range of $Re$.

In the case of $Re = 1000$ we have observed that our results correspond very well with the experimental ones outside the range $24 < x < 30$, depending on our streamline map in this case, obvious that this range is the range of the roof eddy so we can conclude that the roof eddy is changing with direction $z$ (a vortex motion in $z$ direction) and the end wall has a big effect on the flow structure in this range and a negligible effect outside it, this vortex motion stems from the nonlinear interaction between the primary circulation (step’s eddy) and the roof eddy which leads to appear the three dimensional nature of the flow at high Reynolds number and that agrees with Armaly et al. [1] that as $Re > 400$ the flow was found to be strongly three dimensional. Accordingly to that we can return and explain the deviation between our 2-D results and the mid plane experimental measurements in this case.

Not appearing the roof eddy in our streamline map at $Re = 389$ corresponds well with what is reported in the experimental physical measurements by Armaly which declare that the roof eddy begins to appear at $Re > 400$.

The good agreement between our results and the same 2-D Guerrero and Cotta [2] results in the case of $Re = 1000$ supports the validity of our calculate method (Fractional step) and the scheme of approximation (Quick), that have been used.

By observing the streamline maps in figure 1.17, we can see the development of the eddy which formed behind the step and growing accompanying the increase of Reynolds number in this laminar flow region.

In our study we have presented just one mesh 100x90 because it gives an acceptable agreement with the references results, and we found that using another meshes finer would not give more closer results, taking into account the value of the stretching factor of the refinement and the ratio of $\Delta x/\Delta y$ around the step region where the false diffusion could be arise.
Figure 1.17: Streamline maps for cases of Reynolds number (a) Re=100, (b) Re=389, (c) Re=800, (d) Re=1000.
Chapter 2

Direct numerical simulation of Turbulent and laminar air flow in differentially heated cavity

2.1 Differentially heated cavity problem

2.1.1 Introduction

Natural convection flows in enclosures are usually subdivided into three main classes, enclosures heated from below (Rayleigh Bénard convection), others with an induced flow due to internal heat generation and enclosures heated from the side. The prototype configuration of the latter class is the differentially heated cavity. This configuration models many engineering applications such as cooling of electronic components, nuclear reactor insulation, ventilation of rooms,...etc.

The first numerical studies had concentrated on the configurations characterized by small Rayleigh number in the steady laminar regime. After the pioneering work of Vahl Davis & Jones [12], where the original Benchmark formulation was established for a set of square two-dimensional cavities with $10^3 \leq Ra \leq 10^6$. Solutions for the full range of two-dimensional steady-state $Ra \leq 10^6$ have been obtained using different methods by Le Quéré [13], Ravi et al. [14] and Wan et al. [15]. In this work we present 2-D numerical simulation of the non-dimensional natural convection group equations of a Boussinesq fluid using the Fractional step methods, in the case of square differentially heated cavity. The finite volume discretization method is employed and high-order numerical scheme is used in evaluating the convective terms of Navier-Stokes and energy equations. Staggered grid solving type is utilized on uniform and refined mesh at the cavity walls. We got the Nusselt number and the flow pattern results and compared them with the results presented by Vahl Davis [12] as a Benchmark reference.

2.1.2 Governing equations and problem parameters

Considering a square cavity with characteristic length $L$ filled of a Newtonian fluid of kinematic viscosity $\nu$ and thermal diffusivity $\alpha$, a fluid transparent to the Radiation, its physical properties are considered constant except in the flotation term of the momentum equations where Boussinesq hypothesis is used, the effect of viscous dissipation is neglected in the energy equation compared with heat transfer energy term, under these assumptions the dimensionless form of the mass, momentum and energy equations can be written as:
\[ \nabla \cdot \mathbf{u} = 0 \quad (2.1) \]
\[ \frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla) \mathbf{u} = Pr \nabla^2 \mathbf{u} - \nabla p + \mathbf{f} \quad (2.2) \]
\[ \frac{\partial T}{\partial t} + (\mathbf{u} \cdot \nabla) T = \nabla^2 T \quad (2.3) \]

Where \( Ra \) is the Rayleigh number \( Ra = g\beta L^3 \Delta T / (\nu \alpha) \) their values are varying among \( 10^3, 10^4, 10^5, 10^6 \). \( \beta \) is the volumetric thermal expansion coefficient and \( g \) is the gravitational acceleration, \( Pr \) is the prandtl number which equals \( Pr = \nu / \alpha \) and here for the air (\( Pr = 0.71 \)), the body forces vector \( \mathbf{f} = (0, RaPrT) \).

\( \mathbf{u}, t, p \) and \( T \) are the dimensionless velocity vector, time, dynamic pressure and temperature respectively with reference values are, \( \alpha / L \) for the velocity, \( L^2 / \alpha \) for the time, \( \rho (\alpha^2 / L^2) \) for the dynamic pressure, \( \Delta T \) for the temperature and \( L \) for the lengths, where \( \Delta T = T_h - T_c \) is the temperature difference between the hot and the cold vertical walls, whereas the horizontal walls are adiabatic. The no-slip boundary conditions are imposed for the velocity at all the walls.

(a) Uniform mesh.  
(b) Refined mesh.

**Figure 2.1:** 30x30 nodes.

### 2.1.3 Numerical method

**Spatial discretization and time integration method** We discrete the domain of air using the finite volume method and creating a structured mesh with Cartesian coordinates \((x, y)\). two types uniform and refined grids have been used (see figure 2.1). The refinement function that selected is the hyperbolic-tangent function, that takes the form:

\[ n_j = \frac{l}{2} \left( 1 + \frac{\tanh \{ \gamma_n (2(j - 1)/N_n - 1) \} }{\tanh \gamma_n} \right) \quad (2.4) \]

where: \( n_j \) the Cartesian coordinate \((x\ or\ y)\) with total length \( l \), \( j \) is the number of the grid point in the corresponding direction with total number \( N_n \), and \( \gamma_n \) is the concentration parameter at the same direction.
The differential governing equations (eq. 2.1, 2.2, 2.3) are integrated spatially in a staggered mesh where the velocities are evaluated in the surfaces and the pressure in the nodes, where the Quick scheme is used to evaluate the convective terms in the equations. For simplifying the notation, the momentum (eq. 2.2) and the energy (eq. 2.3) can be rewritten as

\[ \frac{\partial \mathbf{u}}{\partial t} = R(\mathbf{u}, \mathbf{f}) - \nabla p \] (2.5)

\[ \frac{\partial T}{\partial t} = R(\mathbf{u}, T) \] (2.6)

Where \( R(\mathbf{u}, \mathbf{f}) = -(\mathbf{u} \cdot \nabla)\mathbf{u} + Pr\nabla^2 \mathbf{u} + \mathbf{f} \), and \( R(\mathbf{u}, T) = -(\mathbf{u} \cdot \nabla)T + \nabla^2 T \). For the temporal discretization, a central difference scheme is used for the time derivative terms for \( \mathbf{u} \) and \( T \), a fully explicit second-order Adams-Bashforth scheme is used for \( R(\mathbf{u}, \mathbf{f}) \), \( R(\mathbf{u}, T) \). A first-order backward Euler scheme is used for the pressure-gradient term. Incompressibility constraint is treated implicit. Thus, we obtain the semi-discretized equations like

\[ \frac{\mathbf{u}^{n+1} - \mathbf{u}^n}{\Delta t} = \frac{3}{2} R(\mathbf{u}, \mathbf{f})^n - \frac{1}{2} R(\mathbf{u}, \mathbf{f})^{n-1} - \nabla p^{n+1} \] (2.7)

\[ \frac{T^{n+1} - T^n}{\Delta t} = \frac{3}{2} R(\mathbf{u}, T)^n - \frac{1}{2} R(\mathbf{u}, T)^{n-1} \] (2.8)

\[ \nabla \cdot \mathbf{u}^{n+1} = 0 \] (2.9)

To solve the velocity-pressure coupling we use a classical Fractional step projection method, in these methods solutions of the unsteady Navier-Stokes equations are obtained by first time-advancing the velocity field \( \mathbf{u} \) without regard for its solenoidality constraint (eq. 2.9), then recovering the proper solenoidal velocity field, \( \mathbf{u}^{n+1} \) after implying the continuity constraint in the Poisson equation, this projection method is derived from the well-known Helmholtz-Hodge vector decomposition theorem, whereby the predictor velocity \( \mathbf{u}^p \) can be uniquely decomposed into a divergence-free vector \( \mathbf{u}^{n+1} \), and the gradient of a scalar field \( \nabla p^- \), this decomposition is written as

\[ \mathbf{u}^p = \mathbf{u}^{n+1} + \nabla p^- \] (2.10)

Where the predictor velocity \( \mathbf{u}^p \) is

\[ \mathbf{u}^p = \mathbf{u}^n + \Delta t \left( \frac{3}{2} R(\mathbf{u}, \mathbf{f})^n - \frac{1}{2} R(\mathbf{u}, \mathbf{f})^{n-1} \right) \] (2.11)

and the pseudo-pressure is \( p^- = \Delta t p^{n+1} \).

The time-advancing temperature is evaluated from eq. 2.8

\[ T^{n+1} = T^n + \Delta t \left( \frac{3}{2} R(\mathbf{u}, T)^n - \frac{1}{2} R(\mathbf{u}, T)^{n-1} \right) \] (2.12)

Taking the divergence of (eq. 2.10) yields to Poisson equation like

\[ \nabla \cdot \mathbf{u}^p = \nabla \cdot \mathbf{u}^{n+1} + \nabla \cdot (\nabla p^-) \longrightarrow \nabla^2 p^- = \nabla \cdot \mathbf{u}^p \longrightarrow \nabla^2 p^{n+1} = \frac{1}{\Delta t} (\nabla \cdot \mathbf{u}^p) \] (2.13)

Once the solution is obtained, \( \mathbf{u}^{n+1} \) results from the correction

\[ \mathbf{u}^{n+1} = \mathbf{u}^p - \nabla p^- \] (2.14)

The Poisson equation (eq. 2.13) has the form \( A \cdot x = b \), we solve it linearly by using LU decomposition for the matrix \( A \) to upper and lower matrices or Band-LU that saves the
matrix A in compact form as A is non-zero band diagonal coefficients matrix, we can use also the TDMA-Gauss iterative method (suppose values ⇒ calculate them line by line with TDMA from n → s and then from e → w ⇒ compare the calculated values with those have been supposed ⇒ resolving till the converging).

For evaluating the time step $\Delta t$, and returning to stability reasons of the explicit temporal schemes that have been used, $\Delta t$ must be bounded by the CFL condition which given like:

$$
\Delta t \left( \frac{|u_i|}{\Delta n_i} \right)_{\text{max}} \leq C_{\text{conv}} \rightarrow \Delta t \leq \frac{C_{\text{conv}}}{\left( \frac{|u_i|}{\Delta n_i} \right)_{\text{max}}} \tag{2.15}
$$

$$
\Delta t \left( \frac{|\nu_i|}{\Delta n_i^2} \right)_{\text{max}} \leq C_{\text{visc}} \rightarrow \Delta t \leq \frac{C_{\text{visc}}}{\left( \frac{|\nu_i|}{\Delta n_i^2} \right)_{\text{max}}} \tag{2.16}
$$

Where $n_i = \begin{bmatrix} x_i \\ y_i \end{bmatrix}$ and the bounding values $C_{\text{conv}}, C_{\text{visc}}$ must be smaller than unity, we will follow the recommendations using values $C_{\text{conv}} = 0.35$, $C_{\text{visc}} = 0.2$, and $\nu_i$ is the viscosity that here the non-dimensional coefficient of the diffusive term $Pr$ in momentum equation and 1 in energy equation.

### 2.1.4 Results and comparisons

From engineering sight the non-dimensional heat transferred, is a very important request in all the natural convection cases, and this heat is expressed by the overall Nusselt number near the hot wall.

Using the reference heat flux that is given by $(\lambda \Delta T)/L$, where $\lambda$ is the thermal conductivity, the non-dimensional local averaged Nusselt number at the vertical hot wall is given by:

$$
Nu_y = -\frac{\partial T}{\partial x} \bigg|_{x=0} \tag{2.17}
$$

and the overall averaged Nusselt at the hot wall can be written:

$$
Nu_{2D} = \frac{1}{L} \int_0^L -\frac{\partial T}{\partial x} \, dy \tag{2.18}
$$

Where: $L = L/L$ is the non-dimensional length of the hot wall.

In the following tables, the results of simulation are presented with the percentage of deviation from the Benchmark results. $(error_1)$ is the deviation error from the Benchmark results of the smoothest uniform mesh, $(error_2)$ is the deviation error from the Benchmark results of the mesh with refinement. The Nusselt numbers and the maximum velocities are presented with the following definitions:

$Nu_{2D}$ the overall Nusselt number at the hot wall.

$Nu_{\text{max}}$ the maximum value of the local Nusselt number at the hot wall, with its position.

$Nu_{\text{min}}$ the minimum value of the local Nusselt number at the hot wall, with its position.

$u_{\text{max}}$ the maximum horizontal velocity on the vertical mid-plane of the cavity, with its position.

$v_{\text{max}}$ the maximum vertical velocity on the horizontal mid-plane of the cavity, with its position.
2.1. Differentially heated cavity problem

\[ Ra = 10^3 \]

<table>
<thead>
<tr>
<th>Uniform mesh</th>
<th>Refinement mesh</th>
<th>Benchmarkerror1 % error2%</th>
</tr>
</thead>
<tbody>
<tr>
<td>mesh 40x40</td>
<td>mesh 60x60</td>
<td></td>
</tr>
<tr>
<td>( Nu_{max} )</td>
<td>1.513</td>
<td>1.506</td>
</tr>
<tr>
<td>( y )</td>
<td>0.088</td>
<td>0.091</td>
</tr>
<tr>
<td>( Nu_{min} )</td>
<td>0.690</td>
<td>0.691</td>
</tr>
<tr>
<td>( y )</td>
<td>0.987</td>
<td>0.992</td>
</tr>
<tr>
<td>( Nu_{2D} )</td>
<td>1.120</td>
<td>1.119</td>
</tr>
<tr>
<td>( v_{max} )</td>
<td>3.690</td>
<td>3.696</td>
</tr>
<tr>
<td>( x )</td>
<td>0.187</td>
<td>0.175</td>
</tr>
<tr>
<td>( u_{max} )</td>
<td>3.649</td>
<td>3.647</td>
</tr>
<tr>
<td>( y )</td>
<td>0.812</td>
<td>0.808</td>
</tr>
</tbody>
</table>

Table 2.1: The results at \( Ra = 10^3 \)

(a) Contour map of the stream line function.  
(b) Contour map of the temperature.

Figure 2.2: Rayleigh number \( Ra = 10^3 \) and mesh 80x80.
Chapter 2. Direct numerical simulation of Turbulent and laminar air flow in differentially heated cavity

\[ Ra = 10^4 \]

<table>
<thead>
<tr>
<th></th>
<th>Uniform mesh mesh 40x40</th>
<th>mesh 60x60</th>
<th>mesh 80x80</th>
<th>Refinement mesh mesh 60x60</th>
<th>Benchmark error1 %</th>
<th>error2%</th>
</tr>
</thead>
<tbody>
<tr>
<td>( Nu_{max} )</td>
<td>3.603</td>
<td>3.560</td>
<td>3.546</td>
<td>3.534</td>
<td>3.528</td>
<td>0.529 %</td>
</tr>
<tr>
<td>( y )</td>
<td>0.137</td>
<td>0.141</td>
<td>0.144</td>
<td>0.142</td>
<td>0.143</td>
<td>0.524 %</td>
</tr>
<tr>
<td>( Nu_{min} )</td>
<td>0.585</td>
<td>0.585</td>
<td>0.585</td>
<td>0.586</td>
<td>0.586</td>
<td>0.2 %</td>
</tr>
<tr>
<td>( y )</td>
<td>0.987</td>
<td>0.991</td>
<td>0.993</td>
<td>0.997</td>
<td>1</td>
<td>0.625 %</td>
</tr>
<tr>
<td>( Nu_{2D} )</td>
<td>2.267</td>
<td>2.253</td>
<td>2.249</td>
<td>2.245</td>
<td>2.238</td>
<td>0.516 %</td>
</tr>
<tr>
<td>( v_{max} )</td>
<td>19.593</td>
<td>19.584</td>
<td>19.625</td>
<td>19.558</td>
<td>19.617</td>
<td>0.04 %</td>
</tr>
<tr>
<td>( x )</td>
<td>0.112</td>
<td>0.125</td>
<td>0.119</td>
<td>0.125</td>
<td>0.119</td>
<td>0.21 %</td>
</tr>
<tr>
<td>( u_{max} )</td>
<td>16.141</td>
<td>16.182</td>
<td>16.176</td>
<td>16.167</td>
<td>16.178</td>
<td>0.013 %</td>
</tr>
<tr>
<td>( y )</td>
<td>0.812</td>
<td>0.825</td>
<td>0.819</td>
<td>0.825</td>
<td>0.823</td>
<td>0.516 %</td>
</tr>
</tbody>
</table>

Table 2.2: The results at \( Ra = 10^4 \).

Figure 2.3: Rayleigh number \( Ra = 10^4 \) and mesh 80x80.

(a) Contour map of the stream line function.  
(b) Contour map of the temperature.
2.1. Differentially heated cavity problem

\[ Ra = 10^5 \]

<table>
<thead>
<tr>
<th>Uniform mesh</th>
<th>Refinement mesh</th>
<th>Benchmark error1</th>
<th>error2</th>
</tr>
</thead>
<tbody>
<tr>
<td>mesh 40x40</td>
<td>mesh 60x60</td>
<td></td>
<td></td>
</tr>
<tr>
<td>8.516</td>
<td>7.444</td>
<td>2.4%</td>
<td>0.35%</td>
</tr>
<tr>
<td>8.051</td>
<td>7.717</td>
<td></td>
<td></td>
</tr>
<tr>
<td>7.889</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>7.744</td>
<td></td>
<td>2.2%</td>
<td>0.3%</td>
</tr>
<tr>
<td>7.452</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>8.075</td>
<td></td>
<td>0.35%</td>
<td>0.2%</td>
</tr>
<tr>
<td>0.081</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.0853</td>
<td></td>
<td>0.3%</td>
<td>0.2%</td>
</tr>
<tr>
<td>0.081</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.083</td>
<td>0.3%</td>
<td>0.3%</td>
<td>0.2%</td>
</tr>
<tr>
<td>0.084</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.083</td>
<td></td>
<td>0.3%</td>
<td>0.2%</td>
</tr>
<tr>
<td>0.084</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.083</td>
<td></td>
<td>0.3%</td>
<td>0.2%</td>
</tr>
<tr>
<td>0.084</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 2.3: The results at \( Ra = 10^5 \).

(a) Contour map of the stream line function.  
(b) Contour map of the temperature.

Figure 2.4: Rayleigh number \( Ra = 10^5 \) and mesh 60x60.
Chapter 2. Direct numerical simulation of Turbulent and laminar air flow in differentially heated cavity

\[ Ra = 10^6 \]

<table>
<thead>
<tr>
<th>Uniform mesh</th>
<th>Refinement mesh</th>
<th>Benchmark</th>
<th>error1%</th>
<th>error2%</th>
</tr>
</thead>
<tbody>
<tr>
<td>mesh 40x40</td>
<td>mesh 60x60</td>
<td>mesh 100x100</td>
<td>100x60</td>
<td></td>
</tr>
<tr>
<td>( Nu_{\text{max}} )</td>
<td>21.904</td>
<td>20.768</td>
<td>19.613</td>
<td>18.859</td>
</tr>
<tr>
<td>( y )</td>
<td>0.037</td>
<td>0.025</td>
<td>0.031</td>
<td>0.035</td>
</tr>
<tr>
<td>( Nu_{\text{min}} )</td>
<td>1.112</td>
<td>1.006</td>
<td>0.986</td>
<td>0.981</td>
</tr>
<tr>
<td>( y )</td>
<td>0.987</td>
<td>0.991</td>
<td>0.993</td>
<td>0.995</td>
</tr>
<tr>
<td>( Nu_{2D} )</td>
<td>9.991</td>
<td>9.361</td>
<td>9.11</td>
<td>8.999</td>
</tr>
<tr>
<td>( v_{\text{max}} )</td>
<td>220.915</td>
<td>219.204</td>
<td>218.342</td>
<td>220.876</td>
</tr>
<tr>
<td>( x )</td>
<td>0.037</td>
<td>0.041</td>
<td>0.031</td>
<td>0.035</td>
</tr>
<tr>
<td>( u_{\text{max}} )</td>
<td>66.291</td>
<td>65.825</td>
<td>65.465</td>
<td>65.223</td>
</tr>
<tr>
<td>( y )</td>
<td>0.862</td>
<td>0.858</td>
<td>0.856</td>
<td>0.855</td>
</tr>
</tbody>
</table>

Table 2.4: The results at \( Ra = 10^6 \).

(a) Contour map of the stream line function.  
(b) Contour map of the temperature.

Figure 2.5: Rayleigh number \( Ra = 10^6 \) and mesh 80x80.

The general description from the tables shows that the results are getting improved with the mesh density. A good agreement has been observed between our results and the Benchmark ones and the values of \( Nu_{\text{max}} \) are improved with the refinement mesh style because of the well resolved thermal boundary layer in this form of grid.

If we plot the error percentage of \( Nu_{2D} \) with the grids used, in \( log – log \) scales, we can note that the error is being higher with the increment of \( Ra \) and being lower with the increment of mesh density. we can note also the \(-2\) slope error that corresponds with the second-order scheme used.
2.1. Differentially heated cavity problem

Figure 2.6: The deviation error of $Nu_{2D}$ from Benchmark results at different meshes and different $Ra$.

2.1.5 Conclusions

We conclude from the results and maps of the stream lines and temperature that the air starts to circulate experienced from the buoyancy forces driven from the hot boundary layer to the cold one. Recirculation has been appeared in the center zone and divided to more vortices with increasing the buoyancy forces i.e. the $Ra$ number.

Figure 2.7: Rayleigh number $Ra = 10^3$, $Ra = 10^4$ in mesh 80x80.
2.2 Direct numerical simulation of 2-D turbulent flow in a differentially heated cavity of aspect ratio 4

2.2.1 Introduction

At high Reynolds number there are unavoidable perturbations in the boundary conditions, the initial conditions and material properties. The structure of the flow becomes more sensitive to these perturbations which give the random phenomenon of the turbulent flows. Turbulent flow can be considered as a flow composed of eddies in different sizes, the large scale motions which are strongly influenced by the geometry of the flow are unstable, and breaking up transferring their kinetic energy to smaller scale motions that in turn undergo a similar breaking up transferring their energy to more smaller scales, at so small scales where the Reynolds number is sufficiently small the eddy motions become stable, and highly affected by the viscosity that works as a factor of dissipation of the kinetic energy and transforming it to heat energy.

In turbulent motion, after an initial transient period it can reach a statistically stationary state in which, even thought the flow variables vary with time the statistics are independent of time. This physical phenomena of turbulent flow can be observed by direct approach of solving Navier-Stokes equations since they accurately describe the turbulent flow, this direct approach is called direct numerical simulation (DNS), applicating DNS could provide new insights into the physics of turbulence which help basically in understanding the turbulent flows and be a big support in developing the turbulence models.

Natural convection is a recurrent phenomenon in the world around us and most of these natural convection flows, especially those encountered in engineering applications are turbulent. The unsteady and turbulent natural convection has thus attracted increasing interest over the last decade for two main reasons: on the one hand there is a desire to improve our phenomenological understanding of turbulent natural convection and on the other hand there is a pressing need for numerical models capable of predicting the corresponding flow structures and related heat transfer in industrial applications.

The first numerical studies had concentrated on the configurations characterized by small Rayleigh number in the steady laminar regime. After the pioneering work of Vahl Davis & Jones [12], where the original benchmark formulation was established for a set of square two-
2.2. Direct numerical simulation of 2-D turbulent flow in a differentially heated cavity of aspect ratio 4

Dimensional cavities with $10^4 \leq Ra \leq 10^6$. Solutions for the full range of two-dimensional steady-state $Ra \leq 10^4$ have been obtained using different methods by Le Quéré [13], Ravi et al. [14] and Wan et al. [15]. Beyond a critical Rayleigh number, the two-dimensional differentially heated cavity flows become time-dependent (periodic, chaotic and eventually fully turbulent), for the square cavity with adiabatic horizontal walls, Le Quéré & Behnia [16] determined the critical number as $Ra = 1.82 \pm 0.01 \times 10^8$ and studied the time-dependent chaotic flows up to $Ra = 10^{10}$. For the case of cavities also with adiabatic horizontal walls and height aspect ratio 4, Le Quéré [17] determined that there is a Hopf bifurcation at $Ra = 2.3 \times 10^8$. Two-dimensional chaotic flows have been studied by Farhangnia et al. [18], who carried out a direct numerical simulation for $Ra = 6.4 \times 10^8$, and by Xin & Le Quéré [11], who studied the cases of $Ra = 6.4 \times 10^8$, $2 \times 10^9$ and $10^{10}$, recording statistics of the flow for $2 \times 10^9$ and $10^{10}$. The cavity with aspect ratio 8 and $Ra = 3.4 \times 10^5 (Ra$ based on the width), has been chosen as a test problem by Christon et al. [19]. Time-dependent two-dimensional flows have also been studied without using the Boussinesq approximation by Paolucci & Chenoweth [20], and Paolucci [21], for Rayleigh numbers up to $10^{10}$ and different aspect ratios.

Recently, a set of two-dimensional and three-dimensional DNS simulations of an air flow in a differentially heated cavity of aspect ratio 4 with $Ra = 6.4 \times 10^8$, $2 \times 10^9$ and $10^{10}$ have been presented and analyzed by Trias et al [3] and here in this work we confirm the general feature flow aspects and the Nusselt number values that have been founded.

2.2.2 Governing equations and problem parameters

Considering a cavity of aspect ratio $\Gamma = H/L$ equals to 4, with high $H$ and length $L$ filled of a Newtonian fluid of kinematic viscosity $\nu$ and thermal diffusivity $\alpha$, a fluid transparent to the Radiation, its physical properties are considered constant except in the flotation term of the momentum equations where Boussinesq hypothesis is used, the effect of viscous dissipation is neglected in the energy equation compared with heat transferred energy term, under these assumptions the dimensionless form of the mass, momentum and energy equations can be written as:

\[
\nabla \cdot \mathbf{u} = 0 \quad (2.19)
\]

\[
\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla)\mathbf{u} = \frac{Pr}{Ra^{0.5}} \nabla^2 \mathbf{u} - \nabla p + \mathbf{f} \quad (2.20)
\]

\[
\frac{\partial T}{\partial t} + (\mathbf{u} \cdot \nabla)T = \frac{1}{Ra^{0.5}} \nabla^2 T \quad (2.21)
\]

Where $Ra$ is the Rayleigh number which based on the cavity height $Ra = \frac{g \beta H^3 \Delta T}{(\nu \alpha)}$ their values are varying among $6.4 \times 10^8$, $2 \times 10^9$, $10^{10}$, $\beta$ is the volumetric thermal expansion coefficient and $g$ is the gravitational acceleration, $Pr$ is the Prandtl number which equals $Pr = \nu/\alpha$ and here for the air ($Pr = 0.71$), the body forces vector $\mathbf{f} = (0, Pr T)$. $\mathbf{u}$, $t$, $p$ and $T$ are the dimensionless velocity vector, time, dynamic pressure and temperature respectively with reference values are, $(\alpha/H)Ra^{0.5}$ for the velocity, $(H^3/\alpha)Ra^{-0.5}$ for the time, $(\rho(\alpha^2/H^2)Ra)$ for the dynamic pressure, $\Delta T$ for the temperature and $H$ for the lengths, where $\Delta T = T_h - T_c$ is the temperature difference between the hot and the cold vertical walls, whereas the horizontal walls are adiabatic. The no-slip boundary conditions are imposed for the velocity at all the walls (see figure 2.9).
Chapter 2. Direct numerical simulation of Turbulent and laminar air flow in differentially heated cavity

Figure 2.9: Schema of two-dimensional differential heated cavity of aspect ratio 4 with adiabatic horizontal walls.

2.2.3 Numerical method

Spatial discretization and time integration method  We discrete the domain identically as in 2.1.3. The differential governing equations (eq. 2.19, 2.20, 2.21) are integrated spatially in a staggered mesh using the second-order symmetry-preserving scheme for the lowest $Ra$ number and the fourth-order one for the two higher $Ra$ numbers that works better in coarser mesh and gives more accurate results.

The main idea of these schemes is, making the discretization in a manner that the difference operators mimic the crucial symmetry properties of the underlying differential operators, i.e., the convective operator is represented by a skew-symmetric matrix and the diffusive operator by a symmetric, positive-definite matrix. In this way not only mass and momentum are conserved but also the kinetic energy.

In the discrete kinetic energy equation, the convective and pressure terms are vanished. If we write the continuity and Navier-Stokes equations in operators form like:

\begin{align}
M \mathbf{u}_h &= 0_h \\
\Omega \frac{d\mathbf{u}_h}{dt} &= -C(\bar{u})\mathbf{u}_h + D\mathbf{u}_h + f_h - \Omega G P_h
\end{align}  (2.22, 2.23)

Where $\mathbf{u}_h$ is the discrete velocity vector, $\Omega$ is a positive-definite diagonal matrix representing the sizes of the control volumes associated with the discrete velocity field $\mathbf{u}_h$, $C(\bar{u})$ the convective coefficient matrix which is skew-symmetric and $\bar{u}$ is the mass fluxes $\bar{u}$ and $\bar{v}$ which form these coefficients, $D$ the discrete diffusive operator which is negative-definite matrix, $f_h$ the discrete source vector and $M,G$ are the discrete divergence and gradient operators respectively (the superscript $t$ is the transpose), the global kinetic energy equation is written like:

\[
\frac{d}{dt}||\mathbf{u}_h||^2 = -u^t_h (C(\bar{u}) + C^t(\bar{u}))\mathbf{u}_h + u^t_h (D + D^t)\mathbf{u}_h + u^t_h (f_h + f^t_h) - u^t_h \Omega G P_h - P^t_h \Omega^t G^t \mathbf{u}_h
\]  (2.24)
The convective operator is skew-symmetric that means:

\[ C(\bar{u}) = -C^t(\bar{u}) \]  \hspace{1cm} (2.25)
\[ \text{diag}(C(\bar{u})) = 0 \] \hspace{1cm} (2.26)

diagonal (C(\bar{u})) as diagonal zero.

This is done by interpolating the flow at the faces as the half sum of their neighbors flow in the corresponding main direction and the same for evaluate the scalar variable at the face, in the two directions.

The negative conjugate transpose of the discrete gradient operator is exactly equal to the divergence operator:

\[ -(\Omega G)^t = M \] \hspace{1cm} (2.27)

That leads to kinetic energy equation like:

\[ \frac{d}{dt} \|u_h\|^2 = u^t_h(D + D^t)u_h + u^t_h(f_h + f^t_h) \] \hspace{1cm} (2.28)

the diffusive term is negative and works as a dissipative factor to the kinetic energy whereas the source term is positive and works as a generator of energy, by this way the kinetic energy is not systematically damped by discrete convective operator or does not need to be damped explicitly to ensure the stability of the method.

In the fourth-order symmetry-preserving scheme, three times larger volume \(\Omega_3\) that is the smallest one possible for which the same discretization rule can be applied as for the original volume \(\Omega_1\), on a staggered grid is used, and then a Richardson extrapolation is applied to remove the local truncation error which is of order \(2 + d\) (where \(d = 2\) in two spatial dimensions and \(d = 3\) in \(3D\) ) and the spatial discretized convective term represented like:

\[ \Omega_3 \frac{d u_h}{dt} + C_3(\bar{u})u_h \] \hspace{1cm} (2.29)

Where \(C_3(\bar{u})\) consists of the flux contributions (\(\tilde{u}\) and \(\tilde{v}\)) through the faces of the large control volumes and \(C_1(\bar{u})\) represents the flux \(\bar{u}\) and \(\bar{v}\) through the faces of the original control volumes. The resulting fourth-order approximation is:

\[ (3^{2+d}\Omega_1 - \Omega_3) \frac{d u_h}{dt} + (3^{2+d}C_1(\bar{u}) - C_3(\bar{u}))u_h \] \hspace{1cm} (2.30)

Where the convective term is represented by a skew-symmetric matrix with zero-diagonal elements.

A short explanation of evaluating the fluxes and velocities at the faces of \(u\) control volume according the second-order scheme is described below with figure 2.10 (for more details about the second and fourth-order symmetry-preserving schemes the reader is refereed to [8])

\[ \bar{u}_e = \frac{u_{(i,j)} - u_{(i+1,j)}}{2} \quad u_c = \frac{u_{(i,j)} + u_{(i+1,j)}}{2} \]
\[ \bar{u}_w = \frac{u_{(i-1,j)} - u_{(i,j)}}{2} \quad u_w = \frac{u_{(i-1,j)} + u_{(i,j)}}{2} \]
\[ \bar{v}_n = \frac{v_{(i,j)} - v_{(i+1,j)}}{2} \quad u_n = \frac{v_{(i,j+1)} + v_{(i,j)}}{2} \]
\[ \bar{v}_s = \frac{v_{(i,j-1)} - v_{(i+1,j-1)}}{2} \quad u_s = \frac{v_{(i,j)} + v_{(i,j-1)}}{2} \]

In order to simplify the notation, momentum and energy equations can be rewritten as

\[ \frac{\partial u}{\partial t} = R(u, f) - \nabla p \] \hspace{1cm} (2.31)
Chapter 2. Direct numerical simulation of Turbulent and laminar air flow in differentially heated cavity

(a) The control volume of the horizontal velocity \( u_{(i,j)} \)

(b) Construction of the fourth-order symmetry-preserving discretization

Figure 2.10: Symmetry-preserving discretization method.

\[
\frac{\partial T}{\partial t} = R(u, T) \tag{2.32}
\]

Where \( R(u, f) = -(u \cdot \nabla)u + (Pr/Ra^{0.5})\nabla^2u + f \), and \( R(u, T) = -(u \cdot \nabla)T + (1/Ra^{0.5})\nabla^2T \).

For the temporal discretization, a central difference scheme is used for the time derivative terms for \( u \) and \( T \), a fully explicit second-order one-leg scheme is used for \( R(u, f) \), \( R(u, T) \). A first-order backward Euler scheme is used for the pressure-gradient term. Incompressibility constraint is treated implicit. Thus, we obtain the semi-discretized Navier-Stokes equations

\[
\frac{(k + 1/2)u^{n+1} - 2ku^n + (k - 1/2)u^{n-1}}{\Delta t} = (1 + k)R(u, f)^n - kR(u, f)^{n-1} - \nabla p^{n+1} \tag{2.33}
\]

\[
\frac{(k + 1/2)T^{n+1} - 2kT^n + (k - 1/2)T^{n-1}}{\Delta t} = (1 + k)R(u, T)^n - kR(u, T)^{n-1} \tag{2.34}
\]

\[
\nabla \cdot u^{n+1} = 0 \tag{2.35}
\]

The parameter \( k \) is computed each time step to adapt the linear stability domain of the time-integration scheme in order to use the maximum \( \Delta t \) possible. Therefore, we look for stability domains which include the eigenvalues of the dynamical system \( \lambda = x + iy \). As \( D \) is a symmetric and negative-definite matrix, the real part \( x \) is negative and its values can be bounded by means of the Gershgorin circle theorem. The skew-symmetry of the \( C(\bar{u}) \) discrete operator allows us to bound the imaginary part \( y \) in the same way.

The approximation of the eigenvalue of the diffusive term in Navier-Stokes equation is given like:

\[
\lambda_{EigenCD}^{D} = -2 \max_{p(i,j)} \left\{- \frac{Pr}{Ra^{0.5}} \left( \frac{\Delta y}{(\partial x)_w} + \frac{\Delta y}{(\partial y)_n} + \frac{\Delta x}{(\partial y)_s} \right) \right\} \tag{2.36}
\]

whereas in the energy equation is given:

\[
\lambda_{EigenCD}^{D} = -2 \max_{p(i,j)} \left\{- \frac{1}{Ra^{0.5}} \left( \frac{\Delta y}{(\partial x)_w} + \frac{\Delta y}{(\partial y)_n} + \frac{\Delta x}{(\partial y)_s} \right) \right\} \tag{2.37}
\]
Where: \((\delta x)_c, (\delta x)_u, (\delta y)_n, (\delta y)_s\) the distances between the node and its neighbor in the four directions, \(\Delta x, \Delta y\) are the two coordinate distances of the volume \(\Omega_{(i,j)}\) of node. This term is evaluated one time and then the approximation of the eigenvalue of the convective term is calculated every time step:

\[
\lambda^C_{EigenCD} = \frac{1}{2} \max_p \left\{ \left( |u_{i,j}| + |u_{i-1,j}| \right) \Delta y + \left( |v_{i,j}| + |v_{i,j-1}| \right) \Delta x \right\} \Delta x \Delta y
\]  

(2.38)

Where: \(u_{i,j}, v_{i,j}, u_{i-1,j}, v_{i,j-1}\) are the velocities at the faces of the control volume \(\Omega_{p(i,j)}\). Once the eigenvalues of our dynamical system are bounded its easy to compute the \(k\) value that better fits the linear stability domain. The optimal value of \(k\) is calculated depending on the ratio of the eigenvalues

\[
\frac{\lambda^C_{EigenCD}}{\lambda^D_{EigenCD}} = \frac{y}{x} \quad \lambda = \lambda^D_{EigenCD} + i\lambda^C_{EigenCD}
\]

which is the tangent of angle \(\varphi\) in the complex coordinates (for details the reader is refereed to [9]), then the optimal \(\Delta t\) is founded depending on \(\|\lambda\|\). Comparing with the standard CFL-based method together with a standard second-order Adams-Bashforth \((k = 1/2)\) scheme, this method leads to time step more than two times greater, that means twice cheaper, without affecting the quality of the numerical results.

To solve the velocity-pressure coupling we use a classical Fractional step projection method, in these method solutions of the unsteady Navier-Stokes equations are obtained by first time-advancing the velocity field \(\mathbf{u}\) without regard for its solenoidality constraint (eq. 2.35), then recovering the proper solenoidal velocity field, \(\mathbf{u}^{n+1}\) after applicate the continuity constraint in the Poisson equation, this projection method is derived from the well-known Helmholtz-Hodge vector decomposition theorem, whereby the predictor velocity \(\mathbf{u}^p\) can be uniquely decomposed into a divergence-free vector \(\mathbf{u}^p\) and the gradient of a scalar field \(\nabla p^\sim\), this decomposition is written as

\[
\mathbf{u}^p = \mathbf{u}^{n+1} + \nabla p^\sim
\]

(2.39)

Where the predictor velocity \(\mathbf{u}^p\) is

\[
\mathbf{u}^p = \frac{2k\mathbf{u}^n - (k - 1/2)\mathbf{u}^{n-1}}{k + 1/2} + \frac{\Delta t}{k + 1/2} ((1 + k)R(\mathbf{u}, f)^n - kR(\mathbf{u}, f)^{n-1})
\]

(2.40)

and the pseudo-pressure is \(p^\sim = \Delta t/(k + 1/2)p^{n+1}\) The time-advancing temperature is evaluated from eq. 2.34

\[
T^{n+1} = \frac{2kT^n - (k - 1/2)T^{n-1}}{k + 1/2} + \frac{\Delta t}{k + 1/2} ((1 + k)R(\mathbf{u}, T)^n - kR(\mathbf{u}, T)^{n-1})
\]

(2.41)

Taking the divergence of (eq. 2.39) yields to Poisson equation for \(p^\sim\)

\[
\nabla \cdot \mathbf{u}^p = \nabla \cdot \mathbf{u}^{n+1} + \nabla \cdot (\nabla p^\sim) \Rightarrow \nabla^2 p^\sim = \nabla \cdot \mathbf{u}^p
\]

(2.42)

Once the solution is obtained, \(\mathbf{u}^{n+1}\) results from the correction

\[
\mathbf{u}^{n+1} = \mathbf{u}^p - \nabla p^\sim
\]

(2.43)

The Poisson equation (eq. 2.42) is of form \(A \cdot X = b\), we solve it linearly by using LU decomposition for the matrix \(A\) to upper and lower matrices like

\[
(L \cdot U) \cdot X = L \cdot (U \cdot X) = b
\]

(2.44)

First its solved for the vector \(Y\) as

\[
L \cdot Y = b
\]

(2.45)
Chapter 2. Direct numerical simulation of Turbulent and laminar air flow in differentially heated cavity

and then solving

\[ \mathbf{U} \cdot \mathbf{X} = \mathbf{Y} \]  \hspace{1cm} (2.46)

The matrix \( \mathbf{A} \) is a sparse matrix contains coefficients and a lot of zeros, it saved using (CSC) compressed sparse column method which stores \( \mathbf{A} \) in three arrays, one array saves the non-zero values of the matrix \( \mathbf{A} \) from top to bottom then left to right, an array saves the row indices corresponding to the position of the non-zero values in his column and the last array saves the indices of the non-zero values where each column starts. This saving of matrix \( \mathbf{A} \) reduces a lot the memory requirements and solves the Poisson equation more rapidly.

**Averaging operators**  The main interest of the direct simulation is not the instantaneous fields but their statistics, in our case after a long enough time from the initial conditions the flow becomes statistically stationary that means all its statistics are invariant under a shift of time and statistically skew-symmetric to the center of the cavity \( i.e., < \Theta(x, y) >= -< \Theta(0.25 - x, 1 - y) > \) where the variable \( \Theta \) can be \( u, v \) or \( T - 0.5 \) and \( x, y \) are the dimensionless coordinates of the cavity.

The time-averaging operator is written as

\[ < \phi >_t = \frac{1}{\Delta t_a} \int_{t_0}^{t_0+\Delta t_a} \phi(t) dt \]  \hspace{1cm} (2.47)

Where \( < \phi >_t \) is the averaged-time of variable like \( u, v, T \) or product of variables like \( (u \cdot u), (v \cdot v), (T \cdot T), (v \cdot T) \).

\( t_0 \) is the starting instant for the evaluation of averaging and its not known at the beginning of simulation, \( \Delta t_a \) is the averaging period that is neither known how long enough should be, to do so partial integrals of the variables and the products of them for evaluation the fluctuation terms such as \( < a' b' > \) which equals \( < ab > - < a > < b > \), are evaluated and stored every 1000 iterations (for example) from the beginning, after a long enough time beyond the stationary state we start the evaluation of the average (here just by combine the partial integrals and divide the sum by \( \Delta t_a \) ) from the end, \( \Delta t_a \) should be enough to obtain the skew-symmetry property of our statistics respect to the center of cavity.

In order to reduce the averaging period \( \Delta t_a \) and consequently reducing the computational cost, a symmetry average operator can be used like

\[ < \Theta(x, y) >= \frac{< \Theta(x, y) >_t - < \Theta(0.25 - x, 1 - y) >_t}{2} \]  \hspace{1cm} (2.48)

Where the subscript \( t \) refers to the averaged-time values.

2.2.4 Results and discussion

We choose grids with sufficient refinement at the vertical boundary layers in order to solve all the relevant turbulence scales, especially in the downstream areas where the boundary layer is disrupted by high fluctuated eddies ejected to the core. The purpose beyond of increasing the concentration parameter at the walls with increasing \( Ra \), is the fact that the thickness of the boundary layers there decreases with the Rayleigh number.

Proposed grid points numbers \( N, M \) with their concentration parameters \( \gamma_x, \gamma_y \) for the three cases of Rayleigh number, the total calculating time, the average period and the values of time step \( \Delta t \) in every case, are presented in table 2.5 accompanying with the data that proposed by Xin & Le Quéré [11](Where Xin & Le Quéré had applicated a spectral algorithm using spatial Chebyshev approximations).

Good agreement results with those founded by Xin & Le Quéré for the values of \( Nu \) and
2.2. Direct numerical simulation of 2-D turbulent flow in a differentially heated cavity of aspect ratio 4

a perfect corresponding with F. X. Trias [10] results, in the three cases of Ra number are presented in table 2.6, with the following quantities:

\( \overline{Nu} \): The overall averaged Nusselt number at the hot wall.

\( Nu_{max} \): The maximum averaged local Nusselt with its position \( y \) at the hot wall.

\( Nu_{min} \): The minimum averaged local Nusselt with its position \( y \) at the hot wall.

\( < u >_{max} \): The maximum averaged horizontal velocity at the vertical mid-length plane with its position \( y \).

\( < v >_{max} \): The maximum averaged vertical velocity at the horizontal mid-height plane with its position \( x \).

<table>
<thead>
<tr>
<th>Ra</th>
<th>( h \cdot M )</th>
<th>( \gamma_x )</th>
<th>( \gamma_y )</th>
<th>( \Delta t )</th>
<th>Total time</th>
<th>Average time</th>
<th>Symmetry scheme</th>
</tr>
</thead>
<tbody>
<tr>
<td>( 6.4 \times 10^8 )</td>
<td>156312</td>
<td>1.5</td>
<td>1.5</td>
<td>( 1.4 \times 10^{-3} )</td>
<td>1000</td>
<td>800</td>
<td>( 2^{nd} )</td>
</tr>
<tr>
<td>( 2 \times 10^9 )</td>
<td>262580</td>
<td>1.75</td>
<td>0.0</td>
<td>( 4.1 \times 10^{-4} )</td>
<td>900</td>
<td>300</td>
<td>( 4^{th} )</td>
</tr>
<tr>
<td>( 10^{10} )</td>
<td>252594</td>
<td>2.0</td>
<td>0.0</td>
<td>( 4.8 \times 10^{-4} )</td>
<td>900</td>
<td>300</td>
<td>( 4^{th} )</td>
</tr>
</tbody>
</table>

(Xin & Le Quéré) \( 6.4 \times 10^8 \) | 64 | 128 | – | – | \( 2.0 \times 10^{-3} \) | 400 | – |

(Xin & Le Quéré) \( 2 \times 10^9 \) | 64 | 256 | – | – | \( 1.5 \times 10^{-3} \) | 350 | – |

(Xin & Le Quéré) \( 10^{10} \) | 96 | 768 | – | – | \( 8.0 \times 10^{-4} \) | 250 | – |

**Table 2.5:** Numerical simulation parameters.

<table>
<thead>
<tr>
<th>Ra</th>
<th>( 6.4 \times 10^8 )</th>
<th>( 2 \times 10^9 )</th>
<th>( 10^{10} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \overline{Nu} )</td>
<td>49.23</td>
<td>66.22</td>
<td>100.60</td>
</tr>
<tr>
<td>( \overline{Nu}_{XQ} )</td>
<td>49.2</td>
<td>66.5</td>
<td>101.0</td>
</tr>
<tr>
<td>( Nu_{max} )</td>
<td>169.88</td>
<td>248.80</td>
<td>446.30</td>
</tr>
<tr>
<td>( y )</td>
<td>4.50x10^{-3}</td>
<td>2.60x10^{-3}</td>
<td>8.45x10^{-4}</td>
</tr>
<tr>
<td>( Nu_{min} )</td>
<td>2.58</td>
<td>3.41</td>
<td>5.16</td>
</tr>
<tr>
<td>( y )</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
</tbody>
</table>

\( < u >_{max} \) | 2.33x10^{-2} | 2.01x10^{-2} | 8.45x10^{-3} |
| \( y \) | 9.49x10^{-1} | 9.48x10^{-1} | 9.58x10^{-1} |

\( < v >_{max} \) | 2.23x10^{-1} | 2.23x10^{-1} | 2.29x10^{-1} |
| \( x \) | 7.50x10^{-3} | 5.79x10^{-3} | 3.90x10^{-3} |

**Table 2.6:** Summery of the averaged flow results. By rows from top to bottom, the magnitudes are: the overall averaged Nusselt number, the maximum averaged local Nusselt number and its position at the hot wall, the minimum averaged local Nusselt number and its position at the hot wall, the maximum averaged horizontal velocity \( < u >_{max} \) and its position at the vertical mid-length plane, the maximum averaged vertical velocity \( < v >_{max} \) and its position at the horizontal mid-height plane.
Averaged fields  Time and symmetry, averaged fields for the temperature and the stream lines in the three cases of Ra have been presented in figure 2.11. Although the large range of Ra number used, the flow characteristics can be described the same of: thin vertical boundary layers near the walls and a large core area with very low velocity and stratified temperature distribution.

The flow motion in the mean upstream part of the boundary layer is laminar until becomes turbulent with strong recirculation near the downstream corners.

Waves are traveling in the vertical boundary layers and grow with moving downstream until they become enough large to deform the boundary layer throwing unsteady large eddies towards the core, the point where these waves disrupt the boundary layer is being closer to the core with increasing the Ra number and this can be noted clearly in figures (2.11 and 2.13.d), where figure 2.13.d shows the averaged temperature at the vertical mid-length plane in the three cases of Ra number.

The mixing effect of these eddies tends to almost gradual distribution of hot upper isothermal regions and cold lower ones, resulting a considered long motionless stratified core area, the phenomenon of ejecting eddies to the core area and compressing its thermal stratification increasingly with the Ra number can be seen clearly from the averaged temperature and stream line maps in figure 2.11.

The strengthening of the horizontal motion by means of large unsteady eddies also results into a remarkable thickening of the boundary layer in the downstream part and consequently a sudden decrease of the vertical velocity (figure 2.12).

The averaged temperature and vertical velocity profiles displayed in the figure 2.12, show that identical profiles are obtained for more than half vertical boundary layer when the lengths are scaled by laminar Ra$^{1/4}$ factor, discrepancies only occur from the point where waves traveling downstream grow large enough to totally disrupt the boundary layer, this confirming that in the range of Rayleigh numbers investigated, the main part of the vertical boundary layer is still laminar or quasi-laminar.

All figures presented confirming well the results of F. X, Trias [10] with an acceptable accuracy.

Heat transfer  From engineering sight the non-dimensional heat transferred, is a very important request in all the natural convection cases, and this heat is expressed by the overall Nusselt number near the hot wall.

Using the reference heat flux that is given by $(\frac{\lambda \Delta T}{H})/H$, where $\lambda$ is the thermal conductivity, the non-dimensional local averaged Nusselt number at the vertical hot wall is given by:

$$Nu_y = -\frac{\partial <T>}{\partial x} \bigg|_{x=0}$$

and the overall averaged Nusselt at the hot wall can be written :

$$\overline{Nu} = \frac{1}{H} \int_0^H -\frac{\partial <T>}{\partial x} dy$$

Where $H = H/H$ the non-dimensional height of the cavity. Table 2.7 shows that the Nusselt correlation is much closer to Ra$^{1/4}$ correlation for laminar flow than the Ra$^{1/3}$ correlation for turbulent flow, the reason for this behavior is that most of heat transfer occurs in the upstream part of the boundary layer where it is almost laminar. To approve this point we have computed the mean Nusselt number at the most downstream part where the boundary layer becomes turbulent. In the last column of table 2.7, we see that the $\overline{Nu}_{down}$, that has been integrated from $y = 0.8$ to $y = 1$ over the hot sidewall, is very close to the classical
2.2. Direct numerical simulation of 2-D turbulent flow in a differentially heated cavity of aspect ratio 4

Figure 2.11: Averaged temperature (left) and stream line (right) at: (a) $Ra = 6.4 \times 10^8$, (b) $Ra = 2 \times 10^9$, (c) $Ra = 10^{10}$.

Figure 2.12: Averaged temperature (left) and vertical velocity (right) profiles at $y = 0.1, 0.2, 0.3, 0.4, 0.5, 0.6, 0.7, 0.8, 0.9$. Each vertical subdivision represents 0.5 units of temperature and 0.2 units for vertical velocity. For plots on the left, the abscissa scale factor is $4x$ and for plots on the right, is $4xRa^{1/4}$, $Ra$ values as figure 2.13.d.
Chapter 2. Direct numerical simulation of Turbulent and laminar air flow in differentially heated cavity

$Ra^{1/3}$ turbulent scaling. This confirms that at the most downstream part boundary layers become turbulent.

The figure 2.13 shows the local averaged Nusselt number distribution at the hot wall in the three cases of $Ra$ number, we can note the position where the boundary layer be fluctuated, that allocated in the downstream part of the boundary layer, we can see also moving this point to the upstream with increasing the $Ra$ number. These phenomena has also been observed by Trias et al. [10] and Le Quéré [11].

<table>
<thead>
<tr>
<th>$Ra$</th>
<th>$Nu$</th>
<th>$Nu_{XQ}$</th>
<th>$Nu/Ra^{1/4}$</th>
<th>$Nu/Ra^{1/3}$</th>
<th>$Nu_{down}$</th>
<th>$Nu_{down}/Ra^{1/3}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$6.4 \times 10^8$</td>
<td>49.23</td>
<td>49.2</td>
<td>0.3095</td>
<td>0.05712</td>
<td>3.03</td>
<td>3.51$\times 10^{-4}$</td>
</tr>
<tr>
<td>$2 \times 10^9$</td>
<td>66.22</td>
<td>66.5</td>
<td>0.3131</td>
<td>0.05255</td>
<td>4.54</td>
<td>3.60$\times 10^{-5}$</td>
</tr>
<tr>
<td>$10^{10}$</td>
<td>100.60</td>
<td>101.0</td>
<td>0.3183</td>
<td>0.04672</td>
<td>7.03</td>
<td>3.26$\times 10^{-3}$</td>
</tr>
</tbody>
</table>

Table 2.7: Nusselt number and correlations.

Figure 2.13: Local Nusselt number distribution at the hot wall: (a) $Ra = 6.4 \times 10^8$, (b) $Ra = 2 \times 10^9$, (c) $Ra = 10^{10}$ and the averaged temperature profile at the mid-length plane (d).
2.2. Direct numerical simulation of 2-D turbulent flow in a differentially heated cavity of aspect ratio 4

Instantaneous fields  Figure 2.14 shows instantaneous snapshots of isothermal maps in the three cases of $Ra$ number $A$, $B$ and $C$ after a long enough time, in the stationary state. Its clearly in general sight, the motionless cavity core and the high fluctuated motion in the two downstream corners of the cavity, in all the cases. However we can see that the high oscillated eddies at the downstream parts of boundary layers that are ejected increasingly with $Ra$ number to the core area deforming its thermal stratification. Snapshots series of the instantaneous isothermal maps in the stationary state, and zoomed in the upper part of the cavity, are showed for the three cases in figures (2.15, 2.16, 2.17), they represent the states at every block of time, we can note that are almost stable in the upstream parts and periodic oscillations can be observed in the downstream parts.

![Snapshots of isothermal maps](image)

**Figure 2.14:** Isothermal instantaneous in the stationary state for: (a) $Ra = 6.4 \times 10^8$, (b) $Ra = 2 \times 10^9$, (c) $Ra = 10^{10}$.

Second-order statistics  The turbulent kinetic energy $K = 0.5(<u'u' > + <v'v'>)$, the temperature variance $<T'T'>$, the turbulent heat flux $<v'T'>$ and two of four non-zero components of Reynolds stress $<u'u'>$, $<v'v'>$, have been represented in figures (2.18, 2.19, 2.20). We can note that these second-order statistics are only significant in the downstream part of the vertical boundary layers and have almost zero values in the upstream part and the cavity core while these areas are laminar.

With increasing the $Ra$ number the fluctuations are being distributed more widely, having more complex form, they are concentrated in the top and bottom region at the downstream parts and that confirms well with the results of Xin & Le Quéré [11] and F. X. Trias [10].
Figure 2.15: Isothermal instantaneous snapshots for $Ra = 6.4 \times 10^8$.

Figure 2.16: Isothermal instantaneous snapshots for $Ra = 2 \times 10^9$.

Figure 2.17: Isothermal instantaneous snapshots for $Ra = 10^{10}$. 
2.2. Direct numerical simulation of 2-D turbulent flow in a differentially heated cavity of aspect ratio 4

The Turbulent kinetic energy $K$ is essentially contributed by $< u' u' >$ whose maxima is located outside the boundary layer making the maxima of $K$ to be located outside too, as shown in figure 2.21 which presents the horizontal profiles of $K = < u' u' > + < v' v' >$ at $y = 0.8$.

Figure 2.18: Second order statistics at $Ra = 6.4 \times 10^8$ from left to right: $K$, $< T' T' >$, $< v' T' >$, $< u' u' >$, $< v' v' >$.

Figure 2.19: Second order statistics at $Ra = 2 \times 10^9$ from left to right: $K$, $< T' T' >$, $< v' T' >$, $< u' u' >$, $< v' v' >$. 

43
Chapter 2. Direct numerical simulation of Turbulent and laminar air flow in differentially heated cavity

Figure 2.20: Second order statistics at $Ra = 10 \times 10^{10}$ from left to right: $K$, $<T'T'>$, $<v'T'>$, $<u'u'>$, $<v'v'>$. 

Figure 2.21: Horizontal profiles at $y = 0.8$ of turbulent kinetic energy $K = (<u'u'> + <v'v'>)$ with scale multiplied by $10^{-3}$. (a) $Ra = 6.4 \times 10^8$, (b) $Ra = 2 \times 10^9$, (c) $Ra = 10^{10}$. 
2.2. Direct numerical simulation of 2-D turbulent flow in a differentially heated cavity of aspect ratio 4

2.2.5 Conclusions

A set of two-dimensional direct numerical simulations of a buoyancy driven flow in a differentially heated air-filled \((Pr = 0.71)\) cavity of aspect ratio 4 and Rayleigh numbers \(Ra = 6.4 \times 10^8, 2 \times 10^9, 10^{10}\) have been presented. An explicit scheme has been used for temporal integration and second and fourth-order schemes for spatial discretization of Navier-Stokes equations whereas just a second order scheme has been used for the energy equation. These schemes preserve the global kinetic energy balances even for very coarse meshes. The main features of the flow, including the time and symmetry averaged flow structure and the turbulent statistics have been presented and compared.

All simulations share some basic flow features: a stratified cavity core, recirculating structures near the downstream corners and thin vertical boundary layers that remain laminar at their upstream part up to a point above the mid-height where transition occurs. Periodic oscillations are amplified in the boundary layer and trigger non-linear effects provoking the transition. Unsteady large eddies disrupt the boundary layer at the downstream part and be ejected to the cavity core increasingly with Rayleigh number forming a motionless thermal stratified core area and upper hot and lower cold fluid at the corners.

For the range of Rayleigh number investigated here, second-order statistics are significant only on the downstream corners of the cavity and have more complex shape increasingly with the Rayleigh number.

Comparing these two-dimensional simulation of turbulent air flow in a differentially heated cavity of aspect ratio 4 with their counterparts of three-dimensional simulation we can note the closed \(Nu\) number values and the similar general features of the flow. However stronger unsteady large eddies are noted in the two-dimensional simulation at the downstream parts with higher scale of the mixing effect making the cavity core region less stratified and the transition point closer to the center than in three-dimensional case as showed by Trias et al. [10].

We can conclude that the two-dimensional simulations can give first and cheaper acceptable description of turbulent flow in the range of Rayleigh number investigated here.
Chapter 2. Direct numerical simulation of Turbulent and laminar air flow in differentially heated cavity
Chapter 3

Direct numerical simulation of laminar and turbulent Rayleigh-Bénard convection
3.1 Introduction

Numerous meteorological, geophysical, astrophysical problems and industrial applications like solar collector design, passive energy storage, cooling of electronic components and others can be formulated as Rayleigh-Bénard convection problems i.e natural convection of fluids heated from below and cooled from above. Many authors through decades have paid big attention and interesting to study this phenomenon in various aspects. Gelfgat [22], Yang [23] and Koschmeider [24] had cared in investigate the instability of the Bénard cells in 2-D and 3-D formulation. The evolution of Rayleigh-Bénard convection from stable bi-modal convection to unstable convection is examined by Mukutmoni and Yang [25] in a cavity with isothermal hot and cold walls. Several researchers have examined the influence of the cavity inclination on the natural of the flow and on the convection transfers. Cianfrini et al. [26] performed a study on natural convection in air filled, tilted square enclosures with two adjacent walls heated and the two opposite walls cooled, the considered Rayleigh numbers varies between $10^4$ and $10^6$ and the tilt angle of the cavity between 0 and 360°. Some studies draw attentions to the complexity of the numerical resolution of flow in tilted cavities, among other parameters, the initial conditions play a major role on the numerical solution that could not be unique. Soong et al. [27] showed that two different solutions exist for some configurations. In certain situations, it is convenient to consider non-isothermal active hot walls in order to comply with a more realistic design of the treated problem, it is the case of electronics where the flux dissipated by the components is neither homogeneous nor steady. Fusegi et al. [28] and Janssen et al. [29] have shown that the transition from the conductive mode to the convective one takes place at higher Rayleigh numbers than those for cavities with isothermal hot walls, and the flow in the central part of the cavity is more complex and non-permanent. Another authors like C. Xia and J. Murthy [30] had investigated the flow transitions in deep 3-D cavities heated from below with various aspect ratio in range $1 - 5$, they proposed two critical Rayleigh numbers, the first below which the flow is at rest and the second for the transition from steady state to oscillatory flow.

Bénard cells can be originated when large enough vertical temperature gradient of long thin horizontal fluid layer heated from below. An arbitrary fluctuation takes a place and a small parcel of the hotter fluid than the neighboring experiencing a buoyancy force begins to rise, the fluid ascends from the underneath to fill the void left by rising part inducing an amplification of temperature fluctuation, as the fluid coming up from under the rising parcel is from the bottom and is hotter than the fluid above. By drawing up more fluid from the hot region, the original temperature fluctuation is amplified and the plume formulated of rising fluid becomes stronger with time. As the temperature difference between the effective surfaces exceeds at high Rayleigh number, the cellular flow becomes more complicated and the two dimensional rolls break up into three dimensional cells that develop to mushroom-like forms, these mushroom-like are the thermal plumes. Assenheimer and Steinberg [31] observed hexagon patterns that occur for Rayleigh number around $6.8 \times 10^3$. When the Rayleigh number exceeds a value of order $10^4$ spoke patterns evolve as showed by Busse and Clever [32], these spokes tends to be stable for lower Ra and appear chaotically when Ra passes $10^5$. Further increase of Ra tears off these spokes to form more independent large-scale flow structures created in the thermal boundary layer that are the plumes, they can be identified as fragments of the thermal boundary layer that detach permanently and move into the bulk, their stems have relatively high amplitude contributions of the local heat flux and the vertical velocity field, also they can be assigned with high-amplitude events of thermal dissipation rate, these facts have been noted by many authors in the last decades like Olga Shishkina and Claus Wagner [33], who have many research in detecting the thermal plumes using specific functions and analyzing the thermal dissipation rate and that in cylindrical cells with different aspect ratios and in rectangular cells by Claus Wagner and Matthias Kaczorowski [34], also Mohammad Emran and Jörg Schumacher [44] have investigated
the fine-scale statistics of temperature in high turbulent cases and presented new strategies in extract the plumes and the turbulent background zones. Grossmann and Lohse \[35\] before, studied the dependence of the flow on the Rayleigh and Prandtl numbers and the aspect ratio, Verzicco and Camussi \[36\] also carried out numerical simulation of turbulent convection within cylindrical container of low aspect ratio $\Gamma = D/H$, both demonstrated the evidence of a decreasing role of plumes for high Rayleigh numbers in thermally driven convection, and the effect of the aspect ratio.

In this work we investigate the flow of air in tilted cavity heated from below and cooled from above as an inclined Rayleigh Bénard convection, in 2-D and 3-D laminar flow previews. We study the effect of some parameters like the aspect ratio and the initial temperature field, that can play a main role in changing the formation of air flow. Then we pass to analyze a turbulent inclined Rayleigh-Bénard convection produced by so large aspect ratio at the same Rayleigh number, we recaptured the events founded by the authors in analyzing the turbulent flow of Rayleigh-Bénard convection. The particular data of Rayleigh and Prandtl number that given here, represent a practical application of studying the air gap in flat plate collector and they employ in analyzing the design of the honey comb cells made of materials that support high temperature of air.

### 3.2 Problem Parameters

The Approximated Rayleigh-Bénard convection in rectangular tilted cavity filled of air, heated from below and cooled from above with adiabatic side walls can represent the events that occur in the air gap of the flat plate collector between the absorber plate and the glass. The sample is inclined by $\theta = 40^\circ$ of horizontal, and the thermal difference between the top and bottom plates about $T_h - T_c = 150^\circ C$ where $T_h = 200^\circ C$ and $T_c = 50^\circ C$. The height of the cavity $H$ at $z$ direction is given to be $4[cm]$, this leads to Rayleigh number $Ra = 2.47 \times 10^5$ and Prandtl number to $Pr = 0.7$, that their correlations are defined in the following. Two aspect ratio have been defined, $\Gamma_1 = L/H$ and $\Gamma_2 = W/H$, where $L$ and $W$ are the length and width of the cavity at $x$ and $y$ directions respectively.

### 3.3 Governing equations and Numerical methods

#### 3.3.1 Governing equation

Considering the air as Newtonian fluid and transparent to the Radiation, its physical properties are evaluated at the mean temperature $(T_h + T_c)/2$, they considered constant except in the flotation term of the momentum equations where Boussinesq hypothesis is used, the effect of viscous dissipation is neglected in the energy equation compared with heat transfer energy term. Under these assumptions the dimensionless form of the mass, momentum and energy equations can be written as:

\[
\nabla \cdot \mathbf{u} = 0 \quad (3.1)
\]

\[
\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla) \mathbf{u} = \sqrt{\frac{Pr}{Ra}} \nabla \mathbf{u} - \nabla p + \mathbf{f} \quad (3.2)
\]

\[
\frac{\partial T}{\partial t} + (\mathbf{u} \cdot \nabla) T = \frac{1}{\sqrt{PrRa}} \nabla^2 T \quad (3.3)
\]

All quantities are expressed in characteristic units. Length scales are normalized with respect to the height of the cavity $H$, velocities with respect to the free-fall velocity, $U = \sqrt{\beta g \Delta T H}$, time scales with respect to $H/U$, the temperature is scaled by the difference between the
hot and cold plates $\Delta T = T_h - T_c$ and the pressure with respect to $\rho U^2$. Rayleigh number is evaluated like $Ra = \beta g \Delta T H^3/\nu \alpha$ and the Prandtl number as $Pr = \nu/\alpha$, where $g$ is the gravity, $\nu$ the kinematic viscosity, $\beta$ the volumetric thermal expansion coefficient and $\alpha$ the thermal diffusivity. $u$, $p$, $T$ and $t$ are dimensionless velocity vector, pressure, temperature and time respectively, the body forces vector $f = (\sin \theta T, 0, \cos \theta T)$. The horizontal plates are assumed to be isothermal with non-dimensional temperature $T_h = 1$ and $T_c = 0$, the adiabatic lateral walls are implemented by means of zero temperature gradient perpendicular to the wall, i.e. $\partial T/\partial x = 0$. No-slip conditions are used for the solid walls whereas the spanwise walls are changed among periodic conditions or adiabatic walls.

### 3.3.2 Numerical methods

**Spatial discretization and Time integration method** The domain of the cavity is discretised using finite volume method creating Cartesian mesh with refinement near the walls (see figure 3.1). The hyperbolic-tangent function refinement has used with the form:

$$n_j = \frac{1}{2} \left( 1 + \frac{\tanh(\gamma_n (2(j - 1)/N_n - 1))}{\tanh \gamma_n} \right)$$

where: $n_j$ the Cartesian coordinate ($x$, $y$ or $z$) with total length $l$, $j$ is the number of the grid point in the corresponding direction with total number $N_n$, and $\gamma_n$ is the concentration parameter in the same direction.

![Discretization mesh](image)

**Figure 3.1:** Discretization mesh, (a) 2-D, (b) 3-D.

The governing equations are integrated spatially in a staggered mesh using the second-order symmetry-preserving scheme. The continuity and Navier-Stokes equations can be written in operators form like:

$$Mu_h = 0_h \quad (3.5)$$

$$\Omega \frac{d u_h}{dt} = -\nabla (\bar{u}) u_h + \nabla u_h + f_h - \Omega G P_h \quad (3.6)$$
Where $\mathbf{u}_h$ is the discrete velocity vector $\mathbf{u} = \begin{bmatrix} u \\ v \\ w \end{bmatrix}$, $\Omega$ is a positive-definite diagonal matrix representing the sizes of the control volumes associated with the discrete velocity field $\mathbf{u}_h$, $\mathbf{C}(\mathbf{u})$ the convective coefficient matrix which is skew-symmetric and $\mathbf{\bar{u}}$ is the mass fluxes $\bar{u}$, $\bar{v}$ and $\bar{w}$ which form these coefficients, $\mathbf{D}$ the discrete diffusive operator which is negative-definite matrix, $\mathbf{f}_h$ the discrete source vector and $\mathbf{M}$,$\mathbf{G}$ are the discrete divergence and gradient operators respectively (the superscript $t$ is the transpose).

The symmetry-preserving scheme and all the steps of the Time integration method have been used as in chapter 2 in three dimensional form where the Poisson equation now has the form

$$\mathbf{A}^{3d} \cdot \mathbf{x}^{3d} = \mathbf{b}^{3d}$$

(3.7)

where: $\mathbf{A}^{3d} \in \mathbb{R}^{N \times N}$ the coefficients matrix, it is a square symmetric and positive-definite matrix which has the coefficients of the internal nodes (the coefficients of the boundaries are included in the next internal coefficient nodes),

$\mathbf{x}^{3d} \in \mathbb{R}^N$ the vector of the unknowns that expresses the pressure of all the nodes,

$\mathbf{b}^{3d} \in \mathbb{R}^N$ the vector of the coefficients at the right hand side.

$N$ the total number of the nodes.

The matrix $\mathbf{A}^{3d}$ is a sparse matrix contains coefficients and a lot of zeros, it has been saved using three arrays, one array saves the non-zero values of the matrix and the others save their corresponding indices, one for $i$, another for $j$. The conjugate gradient algorithm [37] is used to solve eq. 3.7.

### 3.4 Comparisons

The code has been tested by performing additional simulation in purpose of comparing with Benchmark results. Two dimensional numerical simulation of air flow in square differentially heated cavity at the side walls and adiabatic horizontal walls, has done and compared with [12] at $Ra = 10^3$. Moreover three dimensional simulation of a cubic cavity with the same conditions before and adiabatic spanwise walls has compared with [38]. Both simulation present good agreement with Benchmark where the overall Nusselt number is calculated as an important parameter (see tables 3.1 3.2) like

$$Nu_{2D} = \frac{1}{L} \int_0^L - \frac{\partial T}{\partial z} \, dx$$

(3.8)

$$Nu_{3D} = \frac{1}{W \cdot L} \int_0^W \int_0^L - \frac{\partial T}{\partial z} \, dxdy$$

(3.9)

Where: $L = L/H$ and $W = W/H$. (note that the results given in tables 3.1 3.2 can be taken by rotate the cavity i.e. set $\theta = 90^\circ$ or change the boundary conditions).

$Nu_{\text{max}}$ and $Nu_{\text{min}}$ are the maximum and minimum local Nusselt numbers near the hot wall and at the mid-plane in the 3-D simulation. The velocities presented also agree very well with those defined in the references.
Chapter 3. Direct numerical simulation of laminar and turbulent Rayleigh-Bénard convection

### 3.5 Results and Discussion

**Two dimensional simulation** The influence of the cavity length, has been studied by varying the aspect ratio $\Gamma_1 = L/H$ among 0.0625, 0.125, 0.1875, 0.25, 0.5, 0.75 and 1. It’s found that at the smallest aspect ratio, the air does not find enough space to move and it flows in so small levels of velocities resulting to Nusselt number equals to one as a conduction heat transfer (see figure 3.2.a). With increasing the aspect ratio $\Gamma_1 = 0.1875$ the air starts to roll in significant values of velocities transforming its large buoyancy forces to kinetic energy near the isothermal surfaces and making Nusselt number rise rapidly at $\Gamma_1 = 0.25$. After these configurations of aspect ratio the air receives more space to flow and rotate, the buoyancy forces will be greater and the values of velocities arise due to amplifying the isothermal surfaces but at the same time these grown forces are consumed in rolling the air in the center area making the Nusselt number increasing slowly with small values of increment (see figure 3.2.b).

Figure 3.4 shows that all the aspect ratios studied share the same structure of air flow in forming one central rotating cell, except the last one where the air starts to form two cells, all the cells viewed with inclination, and that appears clearly in the large aspect ratios.

<table>
<thead>
<tr>
<th>2-D square cavity</th>
<th>present work</th>
<th>benchmark</th>
</tr>
</thead>
<tbody>
<tr>
<td>mesh 62x62</td>
<td>1.1178</td>
<td>1.117</td>
</tr>
<tr>
<td>$N_{u_{2D}}$</td>
<td>1.506</td>
<td>1.505</td>
</tr>
<tr>
<td>$z$</td>
<td>0.0854</td>
<td>0.092</td>
</tr>
<tr>
<td>$N_{u_{min}}$</td>
<td>0.692</td>
<td>0.692</td>
</tr>
<tr>
<td>$z$</td>
<td>0.997</td>
<td>1</td>
</tr>
</tbody>
</table>

| $u_{max}$         | 3.641        | 3.649     |
| $(x, z)$          | (0.5, 0.806) | (0.5, 0.813) |

<table>
<thead>
<tr>
<th>3-D cubic cavity</th>
<th>present work</th>
<th>present work</th>
<th>benchmark</th>
</tr>
</thead>
<tbody>
<tr>
<td>mesh 32x32x32</td>
<td>1.0713</td>
<td>1.0712</td>
<td>1.070</td>
</tr>
<tr>
<td>$N_{u_{3D}}$</td>
<td>1.424</td>
<td>1.423</td>
<td>–</td>
</tr>
<tr>
<td>$z$</td>
<td>0.0018</td>
<td>0.0979</td>
<td>–</td>
</tr>
<tr>
<td>$N_{u_{min}}$</td>
<td>0.726</td>
<td>0.727</td>
<td>–</td>
</tr>
<tr>
<td>$z$</td>
<td>1</td>
<td>1</td>
<td>–</td>
</tr>
</tbody>
</table>

| $u_{max}$         | 3.526        | 3.530        | 3.543     |
| $(x, y, z)$       | (0.473, 0.528, 0.815) | (0.486, 0.514, 0.825) | (0.5166, 0.5, 0.817) |
| $v_{max}$         | 0.165        | 0.174        | 0.173     |
| $(x, y, z)$       | (0.528, 0.733, 0.528) | (0.514, 0.744, 0.514) | (0.5, 0.7521, 0.5) |
| $w_{max}$         | 3.511        | 3.535        | 3.544     |
| $(x, y, z)$       | (0.815, 0.528, 0.473) | (0.825, 0.514, 0.486) | (0.823, 0.5, 0.5032) |

Table 3.1: Comparison with the benchmark results in square 2-D differentially heated cavities at $Ra = 10^3$.

Table 3.2: Comparison with the benchmark results in cubic 3-D differentially heated cavities at $Ra = 10^3$. 

---

52
3.5. Results and Discussion

**Figure 3.2:** $Nu_{\text{local}}$ along the isothermal hot plate (a) and $Nu_{2D}$ number (b) at all the aspect ratio values.

**Figure 3.3:** Temperature maps at aspect ratios $\Gamma_1 = \{0.0625, 0.125, 0.1875, 0.25, 0.5, 0.75, 1\}$ from left to right respectively.

**Figure 3.4:** Stream line maps that correspond to figure 3.3.
Figure 3.5 shows profiles of the vertical velocity $w$ (3.5.a) along $x$ direction from the right adiabatic end side till the mid-length plane, for all the aspect ratios. At the right it showed the temperature profiles (3.5.b.down) in the boundary layer of height $0.1H$, and the horizontal velocity $u$ profiles (3.5.b.up) along $z$ direction from the hot plate till the mid-height plane. The peaks of $w$ profiles depart a way of the wall with increasing the aspect ratio, and the air rolls in the center area when more space is available. Also $u$ near the hot plate increases with the aspect ratio, and the gradient of temperature becomes greater resulting to a rise in the Nusselt number. It noticed that at the beginning distances of the length especially at high values of $\Gamma_1$, the peaks of $u$ profiles become up warded, and this is returned to the inclination where the slope respect to the horizontal makes the rotating cells inclined.

![Image](image1.png)

**Figure 3.5:** (a) profiles of $w$ velocity along $x$ direction from the right adiabatic end side till the mid-length plane, (b.down) Temperature profiles in the boundary layer of height $0.1H$, (b.up) profiles of $u$ along $z$ direction from the hot plate till the mid-height plane (the numbers beside lines refer to $\Gamma_1$).

In table 3.3, the $N_{u2D}$ results are presented with a study of mesh independence for a basic case and found that the $N_{u2D}$ does not change significantly and there is no need to use so finned mesh for this laminar flow. It should refer also to that the effect of the initial temperature has been checked and found an unique steady solution at different initial values of temperature.

**Three dimensional simulation** 3-D simulations have been implemented in two insights, the first considers the spanwise walls as periodic conditions that can define the infinite general
Table 3.3: 2-D numerical simulation results of natural convection in tilted cavity filled of air, at different values of aspect ratios $\Gamma_1$ and $Ra = 2.47 \times 10^5$.

case and neglect the effect of side walls. The second considers them as adiabatic side wall.

Adiabatic side boundaries. In these outlines the flow of air has been investigated at three different values of $\Gamma_1 = L/H$ between 0.125, 0.1875 and 0.25, and at each value by different cases of $\Gamma_2 = W/H$. As a main case the configuration at aspect ratios $\Gamma_1 = 0.25$ and $\Gamma_2$ varies to 0.125, 0.25, 0.5, 0.75, has chosen in investigate the influence of the initial temperature, the results of $Nu_{3D}$ are presented in table 3.4.

Table 3.4: 3-D numerical simulation results of natural convection in tilted cavity filled of air, at different values of aspect ratios $\Gamma_1$ and $\Gamma_2$, $Ra = 2.47 \times 10^5$ and adiabatic side walls.
Chapter 3. Direct numerical simulation of laminar and turbulent Rayleigh-Bénard convection

It can noted that the configuration of the cavity of $\Gamma_1 = 0.1875$ and $\Gamma_2 = 0.1875$ yields to $Nu_{3D} = 1.08$ that can be a good design for the honey comb cell which supports these high conditions of thermal difference and reduces effectively the heat transfer by convection, we can see that the heat transfer is reduced two times than the case at aspect ratios $\Gamma_1 = 0.25$ and $\Gamma_2 = 0.25$ which has $Nu_{3D} = 2.38$.

Now Three different initial temperature $T^0 = 0, T^0 = 1$ and $T^0 = 0.5$ have been tested in the previous main case. At $\Gamma_2 = 0.125, 0.25$, the air rolls in two dimensional form leading to Nusselt number values in the steady state, smaller than the 2-D simulation results which can be returned to the effect of the side walls in damping the flow by friction. Its found also that all the initial temperature leads to the same values of $Nu_{2D}$ and velocities with the same flow direction.

At $\Gamma_2 = 0.5$ and larger the flow starts to be three dimensional and the air tends to rotate along the diagonal of the cavity expanding at the edges to fill the whole span width and forming consequently a permanent rotating shift over an inclined axis in the spanwise direction, as showed in figures 3.6.e and 3.6.f, the values of $Nu_{2D}$ and velocities do not change here also, but the flow at $T^0 = 0$ becomes to rotate in opposed sense as that in the case of $T^0$ is different from zero (see figures 3.6.c and 3.6.d).

![Figure 3.6: Streamlines at configurations $\Gamma_1 = 0.25$ and $\Gamma_2$ varies to: (a)0.125 (b)0.25 (c)0.5 and $T^0$=0 (d)0.5 and $T^0$=0.5 (e)0.5 diagonal side, (f)0.5 opposite diagonal side.](image)

![Figure 3.7: Isothermal planes at configurations $\Gamma_1 = 0.125$ and $\Gamma_2$ varies to: (a)0.125 (b)0.25 (c)0.375 (d)0.5.](image)
3.5. Results and Discussion

Figure 3.8: Isothermal planes at configurations $\Gamma_1 = 0.1875$ and $\Gamma_2$ varies to: (a)0.1875 (b)0.375 (c)0.5625.

Figure 3.9: Isothermal planes at configurations $\Gamma_1 = 0.25$ and $\Gamma_2$ varies to: (a)0.125 (b)0.25 (c)0.5 (d)0.75.

Periodic side boundaries  Again various configurations of the cavity with periodic spanwise conditions, have been studied in order to see the air flow behavior at infinite limits of the spanwise. Table 3.5 represents the obtained results of $Nu_{3D}$ at each studied case of $\Gamma_1$ and $\Gamma_2$. Five values of $\Gamma_1$ 0.0625, 0.125, 0.1875, 0.25 and 0.5 have been tested and each one by different values of $\Gamma_2$ (the periodic distance). Again a basic studied configuration is chosen to investigate the effect of the span width, that is $\Gamma_1 = 0.25$ and different span widths.

Its found that a distance of cut in the periodic direction should be greater than one ($\Gamma_2 > 1$) to formulate two completed rolls, in the limits of $\Gamma_1 = 0.25$. In figure 3.11, the wavy arrangement in the isothermal planes that correspond to the Bénard cells are showed. The stream lines at two span width distances are presented in figures 3.10.b, 3.10.c, it can be seen that $\Gamma_2 = 0.75$ is not sufficient to formulate two completed rolls adapted with the $\Gamma_1 = 0.25$, whereas $\Gamma_2 = 1$ accomplishes it.
### Table 3.5: 3-D numerical simulation results of natural convection in tilted cavity filled of air at different values of aspect ratios $\Gamma_1$ and $\Gamma_2$, $Ra = 2.47 \times 10^5$ and periodic side boundaries.

<table>
<thead>
<tr>
<th>mesh $N_x \times N_y \times N_z$</th>
<th>$\Gamma_1$</th>
<th>$\Gamma_2$</th>
<th>$Nu_{1D}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>32x162x82</td>
<td>0.0625</td>
<td>2</td>
<td>1.68589</td>
</tr>
<tr>
<td>32x82x82, 62x162x162</td>
<td>0.125</td>
<td>1</td>
<td>3.25900</td>
</tr>
<tr>
<td>32x162x82</td>
<td>0.125</td>
<td>2</td>
<td>3.25964</td>
</tr>
<tr>
<td>32x82x82, 62x162x162</td>
<td>0.1875</td>
<td>0.5</td>
<td>2.41475</td>
</tr>
<tr>
<td>32x82x82, 62x162x162</td>
<td>0.1875</td>
<td>2</td>
<td>4.461480</td>
</tr>
<tr>
<td>82x82x162</td>
<td>0.25</td>
<td>0.25</td>
<td>3.44400</td>
</tr>
<tr>
<td>82x102x162</td>
<td>0.25</td>
<td>0.25</td>
<td>3.44400</td>
</tr>
<tr>
<td>42x62x82</td>
<td>0.25</td>
<td>0.75</td>
<td>4.39277</td>
</tr>
<tr>
<td>62x82x82</td>
<td>0.25</td>
<td>1</td>
<td>5.51187</td>
</tr>
<tr>
<td>62x102x82, 62x162x162</td>
<td>0.25</td>
<td>1.25</td>
<td>5.63706</td>
</tr>
<tr>
<td>62x102x82, 62x162x162</td>
<td>0.25</td>
<td>1.5</td>
<td>5.65738</td>
</tr>
<tr>
<td>62x162x82</td>
<td>0.25</td>
<td>2</td>
<td>5.51182</td>
</tr>
<tr>
<td>62x212x82</td>
<td>0.25</td>
<td>3</td>
<td>5.65727</td>
</tr>
<tr>
<td>62x332x82, 62x82x82</td>
<td>0.25</td>
<td>4</td>
<td>5.64917</td>
</tr>
<tr>
<td>62x82x82</td>
<td>0.5</td>
<td>1</td>
<td>5.13973</td>
</tr>
<tr>
<td>62x162x82</td>
<td>0.5</td>
<td>2</td>
<td>5.13973</td>
</tr>
</tbody>
</table>

$Ra = 2.47 \times 10^5$ and periodic side boundaries.

### Figure 3.10: Bénard cells at configurations $\Gamma_1 = 0.25$ and $\Gamma_2$ varies: (a)4, (b)1, (c)0.75 with periodic side boundaries.
Figure 3.11: Isothermal planes at configurations $\Gamma_1 = 0.25$ and $\Gamma_2$ varies to: (a)0.25, 0.5, 0.75, 1 (b)1.5, 2 (c)3 (d)4.

The Bénard cells arises when a long thin horizontal fluid layer heated from below, has a non-uniform temperature distribution with a static vertical temperature gradient large enough to develop a form of convection instability. It represents a good example of self-organizing and spontaneous establishment of spatial ordering, studying non-linear systems. At small vertical temperature gradient the viscosity of the fluid and its thermal diffusivity prevent the convection motion and the system will have a structure of thermal conductivity with linear heat transfer form, afterward and when the fluid is heated sufficient, the temperature of the layer is increased and a stage is reached (critical temperature) where the
buoyancy driven forces in the fluid overcome its viscosity (the internal friction which opposes movement) and begins to undergo bulk motion in unstable form ended by circulating patterns that called the Bénard cells. These cells are stable and alternate from clock-wise to counter-clockwise horizontally (see figure 3.10.a), they are metastable that means a small perturbation will not be able to change the rotation of the cells, but a larger one could affect the rotation and microscopic perturbations of the initial conditions are enough to produce a non-deterministic macroscopic effect.

From table 3.5, it can be noted that the $Nu_{3D}$ number of the basic case is repeated at aspect ratios $\Gamma_2 = 1, 2$ and at $\Gamma_2 = 1.5, 3$, and linked with identical formulation of Bénard cells observed in the isothermal planes i.e. the air flow is repeated each particular periodic distance until arrives to a constant formulation at $\Gamma_2 = 4$. The heat transfer by convection is so small in the configuration $\Gamma_1 = 0.0625$, $\Gamma_2 = 2$ where $Nu_{3D} = 1.686$, that can be another good design of honey comb cells which installed in the flat plate collector, they can be arranged in closed parallel rows in the air gap to reduce the heat losses produced by the natural convection.

Now if we compare the flow pattern in the basic case at $\Gamma_2 = 0.5$ (figure 3.11.a) and the aspect ratio case $\Gamma_1 = 0.1875$, $\Gamma_2 = 0.5$ (figure 3.14), we can see that the flow still two dimensional in the basic case while becomes three dimensional in the other, this can be due to the fact that the air in wider space tends to rotate in the inclination direction and needs more enough distance -in the periodic direction- to become 3-D. In figure 3.15 when $\Gamma_1 = 0.5$, the flow remains two dimensional even at $\Gamma_2 = 2$. The isothermal planes in figures 3.12, 3.13, 3.14, 3.11 that have the value of $\Gamma_1$ is varying increasingly, they show more curvature in the shape and more influence by the inclination, when $\Gamma_1$ is being increase.

These observations are bounded by the limits used here of $\Gamma_1$, $\Gamma_2$ and $Ra$ where the flow is laminar, while at enough higher values of $\Gamma_1$ the flow becomes transitional or turbulent and pass into its three dimensionality form with smaller width distances.

![Figure 3.12: Isothermal planes at configurations $\Gamma_1 = 0.0625$, $\Gamma_2 = 2$.](image)
3.6. Turbulent Rayleigh-Bénard convection flow in very large aspect ratio cavity

We are interested in studying the total air gap that corresponds to the previous studied case but at aspect ratios $\Gamma_1 = 50$ and $\Gamma_2 = 25$. We want to investigate the evolution of Bénard cells at very large aspect ratio like the air gap in flat plate collector, as well as analyze the
thermal plumes that detach permanently from the thermal boundary layers and expand in the core region, producing high spatial intermittent thermal dissipation rate distributed over the convective cell. The thermal dissipation rate was found to be an important quantity that plays a main role in heat transport mechanism in turbulent Rayleigh-Bénard convection, its statistics tend to be non-Gaussian reflecting the small-scale intermittency in this kind of turbulent convection. Similarly to the passive scalar, the probability density function of the thermal dissipation rate in the bulk region was found to be fitted to stretched exponential in correspondence with (Overholt & Pope [45]; Schumacher & Sreenivasan [46]), its high-amplitude values are found close to isothermal walls in the thermal boundary layers and in the lateral wall regions, these high magnitudes are associated with the mushroom-like thermal plumes in the bulk and the thermal boundary layers. The plumes are generated in the thermal boundary layer close to the bottom or the top wall and are driven to the opposite one by buoyancy, they transform their thermal energy into kinetic energy in the core region forming a connected network of heat transport across the whole cavity.

In our study we have recaptured the events of Turbulent Rayleigh-Bénard convection as a turbulent flow in these configurations and $Ra$ number. The thermal plumes have observed in inclined view and the PDF of the thermal dissipation rate has been evaluated from various gathered instantaneous snapshots. Profiles of fine-scale statistics like the skewness and the flatness, of the temperature fluctuation along the center vertical line have drawn and found different distribution between the bulk and the boundary layer regions, moreover the PDFs of temperature have plotted with increasing the height and found that PDFs are being increasingly symmetric toward the center plane that means an equal distribution of the hot and cold temperature is detected in the center zone because of the mixing of plumes, the tails of PDFs found to be stretched because of the intermittency of temperature. A sufficient averaged time is given and noted to be so long because of the configuration and weak turbulent convection, where three different probe points have chosen to observe the temporal advancing of the flow features, the histograms of flow feature were found unidirectional distributed over the time. Study of mesh density to establish the kolmogorov turbulent scales has done upon the Grötbach criterion [40] when $Pr < 1$, in addition the periodic spanwise has been investigated to minimize the domain in that direction where $\Gamma_2$ is so high and can be considered as infinite boundaries, its found that $\Gamma_2 = 4$ is sufficiently expresses the domain and that verified by evaluating the two-points correlations of flow features in the probe points. The velocity and temperature field are found to be correlated within the thermal boundary layer and tend to be uncorrelated in the bulk region, and that by evaluating the correlation coefficients $C_{1w'T'}, C_{2v'T'}$.

### 3.6.1 Resolution requirements

In order to resolve the smallest relevant turbulent scales in DNS, the resolution of the flow field must be sufficiently high. Therefore, the grid points are clustered in the vicinity of the effective walls using hyperbolic-tangent function refinement, so that about 12 grid points in the high density mesh are within the thermal boundary layer and grid spacing $h_{vi} = \sqrt[3]{\Delta x \Delta y \Delta z}$ in the bulk region satisfies the Grötbach [40] estimate for the kolmogorov scales $\eta_k$.

$$h_{vi} \leq \eta_{k,Gr.} \equiv \frac{\pi}{\sqrt{\frac{Pr^{1/2}}{(Nu - 1)Ra}^{1/4}}}$$

(3.10)

Where $\Delta x, \Delta y$ and $\Delta z$ are the dimensionless distances of the largest control volume in the bulk region. Two mesh sizes have been tested at different values of spanwise periodic distance and the averaged overall Nusselt number $<Nu_{\alpha,D}>$ is evaluated in all the case as an important comparison parameter ($< >$ denotes the time averaging), using the scaling law of Nusselt number that presented by Shishkina & Wagner [33], who did DNS of turbulent...
Rayleigh-Bénard convection in cylindrical sample of aspect ratio $\Gamma = D/H$ equals to 10 and $Ra = 10^5, 10^6, 10^7$ and Prandtl number $Pr = 0.7$. $Nu = 0.128Ra^{0.303}$ that is in general agreement with this reported by Niemela et al. [41]. In our case $Ra = 2.47 \times 10^5$ that yields to $Nu \simeq 5.4$ and we have Prandtl number $Pr = 0.7$, using the equation 3.10 the Grötbach estimate of the scale takes the value $\eta_{k,Gr.} = 8 \times 10^{-2}$ Table 3.6 displays the tested probes with the results of $<Nu_{3D}>$. 

<table>
<thead>
<tr>
<th>probe</th>
<th>Periodic span width $\Gamma_2$</th>
<th>$N_x$</th>
<th>$N_y$</th>
<th>$N_z$</th>
<th>$\gamma_x$</th>
<th>$\gamma_y$</th>
<th>$\gamma_z$</th>
<th>$h_{DNS}$</th>
<th>$&lt;Nu_{3D}&gt;$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1.5</td>
<td>802</td>
<td>42</td>
<td>42</td>
<td>0</td>
<td>0</td>
<td>1.5</td>
<td>$4.54 \times 10^{-2}$</td>
<td>5.047</td>
</tr>
<tr>
<td>2</td>
<td>3</td>
<td>802</td>
<td>82</td>
<td>42</td>
<td>0</td>
<td>0</td>
<td>1.5</td>
<td>$4.45 \times 10^{-2}$</td>
<td>4.986</td>
</tr>
<tr>
<td>3</td>
<td>4</td>
<td>802</td>
<td>110</td>
<td>42</td>
<td>0</td>
<td>0</td>
<td>1.5</td>
<td>$4.52 \times 10^{-2}$</td>
<td>4.965</td>
</tr>
<tr>
<td>4</td>
<td>4</td>
<td>1202</td>
<td>110</td>
<td>62</td>
<td>0</td>
<td>0</td>
<td>1.5</td>
<td>$3.34 \times 10^{-2}$</td>
<td>4.953</td>
</tr>
</tbody>
</table>

**Table 3.6:** Results $<Nu_{3D}>$ in different performed probes with the information of mesh size used where $\gamma_n$ is the concentration parameter in the corresponding direction $n$.

We can see that the $<Nu_{3D}>$ has no significant change at $\Gamma_2$ between 3 and 4, that means $\Gamma_2 = 4$ is sufficiently reflects the infinite spanwise sides. An additional proof of adequacy of the computational domain at $\Gamma_2 = 4$, two-points correlation in the homogeneous direction $y$ and for all the features of flow has evaluated to ensure that the turbulent fluctuation are uncorrelated at separation of one-half distance. Three different locations of coordinates $(x, z)$, $P1(25, 0.5)$, $P2(49, 0.05)$ and $P3(46, 0.8)$ have monitored as three different form of flow regions. As known that a high degree of correlation exists between the velocity at two points in space if the distance between them is smaller than the diameter of the eddy. Conversely if the points are far enough in between that many eddy diameters can be found, a little correlation can be expected. the two-points correlation of variable fluctuation in statistically stationary state can be given like

$$R_{\phi\phi}(x, r, z) = \frac{\langle \phi'(x, y, z) \phi'(x, y + r, z) \rangle}{\langle \phi'^2 \rangle}$$

(3.11)

Figures 3.16 shows that the correlation values in all the three points fall to zero or so small magnitude at separation one-half distance and even lower than one-half for some features, which verifies that the computational domain of $\Gamma_2 = 4$ is sufficiently large.
Another request could be attained to ensure that the smallest relevant turbulent scales are well-resolved. To do so, one-dimensional energy spectra of temperature and velocity fields at the periodic $y$ direction and the center position $(25, y, 0.5)$ have performed (see figure 3.17), where it can be defined as

$$E_{\phi\phi}(\omega) = \langle \hat{\phi}_\omega \hat{\phi}_\omega^* \rangle$$

(3.12)

$$(.)^*$ denotes the complex conjugate and $\omega$ the frequency.

Its observed that the temperature spectra match the Bolgiano exponent [47] of $-7/5$, but lack the inertial subrange which is supposed to follow the buoyancy subrange. According to the Bolgiano dynamics the velocity spectra would show a $11/5$ decrease within the buoyancy subrange, however, only the kolmogorov-law $-5/3$ is observed. This is in agreement with results Verzicco and Camussi [36] who argued that this might be the case, when most of the thermal energy is injected into the large scales through the wind. on the other hand it has
to be taken in account that Bolgiano dynamics assume a stable stratified fluid layer which is not given in Rayleigh-Bénard convection where energy is injected into fluid by means of thermal plumes. Its therefore reasonable that the velocity spectra follow the kolmogorov law, since there is no energy extracted from the velocity field and stored as potential energy as suggested by Bolgianos theory. It can be observed that the inertial subrange is so difficult to distinguish and becomes larger at high $Ra$ number as shown by Kaczorowski & Wagner [34] who founded so small inertial subrange at $Ra = 3.5 \times 10^5$, no energy pile-up at high wave numbers has noted and the energy density drops several order of magnitudes at larger wave number that means the grid resolutions are enough to resolve the smallest relevant turbulent scales.

### 3.6.2 Temporal analysis

Due to the high aspect ratio ($\Gamma_2$) of the cavity with $Ra$ number approximately low, the period of converging the statistics is found to be so long. As mentioned before, three different locations in the cavity have monitored as probe points to observe the developing of flow features, an averaging operator has been implied to check the adequacy of time averaging period in the monitoring points $P_1(25, 2, 0.5), P_2(49, 2, 0.05)$ and $P_3(46, 2, 0.8)$ where the sufficient averaging period is detected to be $5000[TU]$, reminding that the normalized time scale is $H/U$ where $U$ the free-fall velocity, $U = \sqrt{\beta g \Delta T H}$. Some Authors like I. Rodríguez, O.Lehmkul [39] in their DNS and J.J. Niemela, L. Skrbek [41] in their experiments, of turbulent Rayleigh-Bénard convection, showed existence of periodic alternating circulation large-scale flow motion under a shift of time the histogram of the velocity obtains a bimodal shape. On account of this we would like to check out this phenomena in our case by plotting the vertical velocity $w$ and temperature histograms in the monitoring points (see figures 3.18 3.19), its showed that all the diagrams present almost Gaussian distribution over the time of simulation referring to alternating the descend and the ascend flow by small-scale motion, in passing through the bulk region where the hot and cold thermal plumes are mixed well, a little deviation from the normal Gaussian distribution found to be getting smaller toward the center point $P_1$, this will be proofed later by calculating the skewness and the flatness of temperature along the center line as functions of the height.
Figure 3.18: Vertical velocity histogram at: (a) P1(25, 2, 0.5), (b) P2(49, 2, 0.05) and (c) P3(46, 2, 0.8)
3.6. Turbulent Rayleigh-Bénard convection flow in very large aspect ratio cavity

3.6.3 Averaged Turbulent components

Two-dimensional maps averaged over time and periodic direction of turbulent flow features with the profile of averaged local Nu number, have presented in figures (3.20 → 3.29). In purpose of not losing the details of flow features in the upstream and downstream regions since the aspect ratio is so high and showing the map as one figure could modify the details of flow, the maps are presented as three highlighted parts downstream (left), middle (bottom) and upstream (right) flow where the features of flow are similar along the center cavity and no importance to show all the center zone. A general description can be given from the first-order averaged statistics (figs.3.20 and 3.21) of coherent separated circulation of hot and cold fluid in the downstream and upstream vicinities due to the inclination and side walls however, a linear thermal stratification along the center area can be noted with weak circulation cells because of the long aspect ratio, and that give the isothermal planes trivial wavy form. The air starts to flow in laminar sense in the boundary layer at the upstream part, compressed by the coherent circulation there, until arrives to an inflection point where the coherent side vortex is separated from the center area. This point can be detected in the local averaged Nusselt number profile at the hot wall (see figure 3.22) where a down wardsed cusp appears in the upstream area, another up warded one arises in the downstream area where the flow rotation at the separation point is downward. These positions can be recognized also in the averaged temperature maps by a big wave between the center and side areas (see figure 3.20), behind these separation points in the center area the flow is formed by two up descend and down ascend coherent flow driven by the inclination, the interaction between the two streams leads to create small and weak circulation cells distributed along the cavity as can be seen in the averaged stream line figure 3.21.

The distribution of turbulent kinetic energy $< k' > = 0.5 < u'u' >$, the turbulent heat flux components $< w'T' >$ and $< u'T' >$, the temperature variance $< T'T' >$ and three of the four non-zero components of Reynolds stress have been presented in figures 3.23 → 3.29 respectively. All the statistics present simple distribution within the domain in the side regions since the more complex form is found in the center area where the plumes are mixed. High values of turbulent kinetic energy are noted in the previous inflection points or the cusps positions in averaged local Nusselt profile (between 0.06$\Gamma_1$ – 0.08$\Gamma_1$), that associated with high values of turbulent heat flux which likewise have a significant distribution in the center area. The highest values of temperature variance are observed near the isothermal
walls at the thickness of the thermal boundary layer where the stems of plumes arose, so low fluctuation events are observed near the adiabatic side walls and generally the highest fluctuations are distributed where the thermal plumes.

Figure 3.22: Local averaged Nusselt number $N_{u_{avg}}$ profile at the hot wall.
3.6. Turbulent Rayleigh-Bénard convection flow in very large aspect ratio cavity

**Figure 3.23:** Turbulent kinetic energy $< k' >$ maps in three highlighted zones: left (downstream), right (upstream) and bottom (middle).

**Figure 3.24:** The vertical component of turbulent heat flux $< w'T' >$ maps in three highlighted zones: left (downstream), right (upstream) and bottom (middle).

**Figure 3.25:** The horizontal component of turbulent heat flux $< u'T' >$ maps in three highlighted zones: left (downstream), right (upstream) and bottom (middle).
Figure 3.26: Variance Temperature $<T'T'>$ maps in three highlighted zones: left (downstream), right (upstream) and bottom (middle).

Figure 3.27: Variance $y$ velocity component $<v'v'>$ maps in three highlighted zones: left (downstream), right (upstream) and bottom (middle).

Figure 3.28: Variance $z$ velocity component $<w'w'>$ maps in three highlighted zones: left (downstream), right (upstream) and bottom (middle).
3.7 Temperature and thermal boundary layer

Figure 3.29: Variance $x$ velocity component $<u'u'>$ maps in three highlighted zones: left (downstream), right (upstream) and bottom (middle).

3.7 Temperature and thermal boundary layer

Figure 3.30.a shows the probability density function (PDF) of the dimensionless Temperature $T$ at different heights $z$ from the hot wall till the center in the bulk region. It's noted that the (PDF) of temperature becomes increasingly symmetric toward the center where the hot rising and the cold falling flows are good mixing (thermal plumes mixing) this has been observed also by the Authors like Emran & Schumacher [44] in turbulent Rayleigh-Bénard convection cases where the tails of (PDF) used to be stretched distribution reflecting the spatial intermittency of temperature. Our distribution tends to be Gaussian in the upward parts however the stretched tails can not be Gaussian, and that corresponds with the observations of the temperature spatial distribution in turbulent Rayleigh-Bénard convection where the distribution becomes more sharper with peaked top by increasing the Rayleigh number.

The thermal boundary layer is resolved with 12 grid points clustered near the isothermal walls as displayed in figure 3.30.b, where we plot the vertical mean square of temperature fluctuation profile $<T'T'>$ along the center vertical line of the 2-D time and periodic direction averaged features plane. The thermal boundary layer thickness can be estimated by $\delta_t = H/2 <Nu_{3D}>$ and detected here by $\delta_t = 0.1$ which is always close to the position of the maximum of $<T'T'>$ as showed clearly in figure 3.30.b, that means, our simulation corresponds very well with the estimated correlation of detecting the thickness of thermal boundary layer, afterward in the bulk region the increasing dissipation of thermal plumes leads to a reduction in temperature fluctuations and the profile slopes toward the small values of $<T'T'>$.

For further study of the temperature height-dependence, the skewness and the flatness of the temperature fluctuation have been plotted over the height in figure 3.31, they can be given respectively as

$$S_{T'} = \frac{<T'^3>}{<T'^2>^{3/2}}$$  \hspace{1cm} (3.13)

$$F_{T'} = \frac{<T'^4>}{<T'^2>^2}$$  \hspace{1cm} (3.14)

the $S_{T'}$ measures the temporal deviation of $T'$ from asymmetry, its the third moment around the mean normalized by the cube of the rms. Zero value of skewness indicates that the distribution of the variable is normal and symmetric, however the negative or positive values
Chapter 3. Direct numerical simulation of laminar and turbulent Rayleigh-Bénard convection

Figure 3.30: Probability density function (PDF) of Temperature $T$ at different heights of $z$ (a), and the mean square of temperature fluctuations profile $<T'T'>$ extracted along the vertical center line of 2-D time and periodic direction averaged results plane (b), the data of the temperature PDF is taken of gathering a sequence of independent snapshots.

Figure 3.31: High dependence of (a) the skewness and (b) flatness of the temperature fluctuations along a central mid-length mid-width line of the cavity (the vertical line through $P1(25, 2, 0.5)$).

indicate that the distribution is skewed right or left of its average. Figure 3.31.a displays the changing profile of the skewness with the height where a small relatively value of skewness can be detected. The high negative and positive skewness around $\delta_t$ implies that rising and falling plumes are observed at the edge of the bottom and top thermal boundary layers. Simultaneously the colder and hotter pockets of temperature are generated in the vicinity of the thermal plumes that detach from the thermal boundary layer, afterward up to height $z = 0.2$ the skewness decreases monotonically before declines to zero at the center in the bulk region where the plumes are mixed. The displacement of skewness from negative values to positive ones or in reverse, corresponds to a rising plume or falling plume from the hot
or cold wall with temperature higher or lower than the averaged time temperature, then the both plumes start to extend and mix in the bulk region leading to normal Gaussian distribution in the center where the skewness is zero.

The magnitude of the deviations from Gaussianity can be measured by flatness $F_T'$. In figure 3.31.b, the changing profile of the excess kurtosis (deviation of 3) or flatness with the height is presented. A relatively small and negative value distribution of the flatness can be observed, with high negative value in the thermal boundary layers start afterward to increase in the bulk region till arrives near zero at the center where becomes more Gaussianity. these observations of varying the flatness between the boundary layer and the bulk regions, again agree well with views of the Authors like Ching [48], Emran & Schumacher [44].

### 3.8 Thermal dissipation rate and thermal plumes

The thermal dissipation rate was found to be an important quantity that plays a main role in heat transport mechanism in turbulent Rayleigh-Bénard convection, its value indicates the magnitude of gradients of the turbulent temperature field by

$$\varepsilon_T = \alpha (\nabla T)^2$$  \hspace{1cm} (3.15)

Where $T$ is the dimensional temperature, the ensemble (space-time average) of the thermal dissipation rate field can be directly related to $Nu$ number, this relation is given by

$$<\varepsilon_T>_{t,v} = \frac{\Delta T^2}{H^2}Nu$$ \hspace{1cm} (3.16)

Now in non-dimensional form, the thermal dissipation rate can be derived as

$$\varepsilon_T = \frac{\alpha (\Delta T)^2}{H^2}(\nabla T)^2 = \frac{(\Delta T)^2U}{H} \frac{1}{\sqrt{RaPr}}(\nabla T)^2$$ \hspace{1cm} (3.17)

which gives

$$\varepsilon_T = \frac{1}{\sqrt{RaPr}}(\nabla T)^2$$ \hspace{1cm} (3.18)

and $<\varepsilon_T>_{t,v}$ from eqs. 3.16 and 3.18 can be evaluated as

$$<\varepsilon_T>_{t,v} = \frac{Nu}{\sqrt{RaPr}}$$ \hspace{1cm} (3.19)

Both equations 3.15 and 3.18, indicate that the statistics of the fluctuating thermal dissipation field are connected with the local fluctuations of the conductive heat transfer, as given by the local currents, $\mathbf{j}(\mathbf{x}, t) = -\alpha(\nabla T)$, here $\mathbf{x} = \begin{bmatrix} x \\ y \\ z \end{bmatrix}$.

In all the studies the thermal dissipation rate is found to present highly non-Gaussian statistics reflecting the small-scale intermittency in turbulent Rayleigh-Bénard convection. Its high-amplitude are found associated with the thermal boundary layers and the thermal plumes that detach permanently from them, thermal plumes can be defined as a fragment of the thermal boundary layers that resourced from amplifications in the fluctuation of vertical temperature gradient where alternative displacements of differentially heated fluid take a place, their roots are born in the thermal boundary layers where a pushing is created by the caps of the plumes arose from the opposite boundary layer. Their stems arise at the intersections of the hot and cold roots, they are driven to the opposite wall by buoyancy and propagate in the bulk region forming their caps and taking the mushroom-like form. By its
expanding in the bulk region the plumes transform their thermal energy into kinetic energy constructing a connected network of heat transport across the whole cavity. The thermal plumes also can be considered as local events in which a vertical upward (downward) vertical velocity fluctuation and a positive (negative) temperature fluctuation are directly correlated i.e. $w' T' > 0$ the positive values of turbulent heat flux (referred to [49]), in outline of these sights we plot the profiles of correlation coefficients $C1_{w'T'}, C2_{w'T'}$ over the height of cavity (see figures 3.32.a and 3.32.b) these correlation coefficients can be given by

\begin{align}
C1_{w'T'} &= \frac{<w'T'>}{\sqrt{<w'^2><T'^2>}} \\
C2_{v'T'} &= \frac{<v'T'>}{\sqrt{<v'^2><T'^2>}}
\end{align}

It can be observed from the figures that the two turbulent fields are strongly correlated

\begin{figure}[h]
\centering
\includegraphics[width=\textwidth]{figure3.32.png}
\caption{Correlation coefficients of the turbulent fluctuations of the velocity and temperature fields at the same vertical line that define in figure 3.31.}
\end{figure}

within the boundary layers and tend to decorrelate in the bulk region where the plumes are mixed. that correspond well with the observations by Kaczorowski & Wagner [34] in their simulation results.

Figure 3.33 displays a zoomed 3-D section of the thermal plumes taken from instantaneous snapshot in range of isothermal planes $0.5 \rightarrow 1$ (3.33.a) and $0.8 \rightarrow 1$ (3.33.b) with the corresponding thermal dissipation rate $\varepsilon_T$ (3.33.c) as a clip of height $0.3H$ because the two isothermal walls are so closed and the patterns of $\varepsilon_T$ can be overlapped. An inclined large-scale structure view of thermal plumes is observed in figure 3.33.a and the ridges (stems) of the thermal plumes that detach from hot thermal boundary layer can be recognized in figure 3.33.b, their pattern are recaptured almost identically in figure 3.33.c of the thermal dissipation rate.

The highest values of $\varepsilon_T$ have been detected close to both isothermal walls where the high interactions between the caps of the hot rising plumes that enter the cold top boundary layer, and the roots of the falling cold plumes. the same event occurs between the descending cold plumes with the hot roots in the bottom hot boundary layer. It found also high values of $\varepsilon_T$ in the side wall regions, as shown in figure 3.34 where three contour plots of instantaneous temperature and the corresponding thermal dissipation rate $\varepsilon_T$ at different three heights
3.8. Thermal dissipation rate and thermal plumes

Figure 3.33: Extracted instantaneous zoomed section of the hot thermal plumes at temperature range \(0.5 \rightarrow 1\) (a), their stems \(0.8 \rightarrow 1\) (b) and the corresponding thermal dissipation rate \(\varepsilon_T\) through a clip of height 0.3\(H\) (c).

\[z = \delta_t, \quad z = 0.5H\] and \(z = H - \delta_t\), are presented, the coherent circulation of the hot upward flow and the cold downward one in the both side regions induces high gradient temperature in the top and bottom thermal boundary layers in that zones and therefore high amplitudes of \(\varepsilon_T\). In the center region where the plumes are dominating, the interaction locations can be highlighted by the high values of \(\varepsilon_T\) at heights \(\delta_t, H - \delta_t\), whereas its values are decreased toward the bulk as showed at height 0.5\(H\).

In order to investigate the distribution of the thermal dissipation rate through the cavity, the probability density function of the normalized thermal dissipation rate by \(\langle \varepsilon_T \rangle_{t,v}\), has been plotted in the thermal boundary layer, the bulk and the whole cavity. The data are gathered from a sequence of independent snapshots and plotted over a log–log and log–linear scales.

Figure 3.35 shows the PDF of the normalized thermal dissipation rate \(\varepsilon_T / \langle \varepsilon_T \rangle_{t,v}\) in the entire volume of the cavity. It exhibits non-Gaussian spatial distribution with stretched exponential tails reflecting the spatial intermittency effect. We can distinguish between three main regions of the PDF distribution:

1. Turbulent background region that can be fitted to Gaussian distribution with respect to \(\log(\varepsilon_T / \langle \varepsilon_T \rangle_{t,v})\), it corresponds to so small values of normalized thermal dissipation rate that detected in the bulk region, its values follow the contributions \((\varepsilon_T / \langle \varepsilon_T \rangle_{t,v}) < 0.1\) where an inflection point in the PDF can be observed.

75
Chapter 3. Direct numerical simulation of laminar and turbulent Rayleigh-Bénard convection

Figure 3.34: Three contour plots of instantaneous temperature (up) and the corresponding thermal dissipation rate $\varepsilon_T$ (down) at different three heights $z = \delta_t$ (top group), $0.5H$ (middle group) and $H - \delta_t$ (bottom group).

(2) Thermal plumes region where a deviation from the behavior of the passive scalars presented by Schumacher & Sreenivasan [46], occurs because of the thermal plumes. Our PDF is found to be in agreement with the observations of the passive scalar in the turbulent background region where the Gaussian distribution falls below of our PDF (this will be obvious in the highlighted PDF of the bulk region) and the differences compared to the passive scalar convection PDFs was found in range $(\varepsilon_T/ < \varepsilon_T >_{t,v}) > 0.1$ that considered to be a result of the thermal plumes and boundary layers. The values of the thermal plumes regions follow the contributions $0.1 < (\varepsilon_T/ < \varepsilon_T >_{t,v}) < 10$ where another inflection point in the PDF can be noted.

(3) Conductive sublayer region that corresponds to the highest values of normalized thermal dissipation rate in the thermal boundary layer where so high spatial intermittency is evolved. Its values follow the range $(\varepsilon_T/ < \varepsilon_T >_{t,v}) > 10$ they are fitted by an exponential function with constant coefficient $\sim 1/\sqrt{RaPr}$ i.e. the non-dimensional thermal conductivity.

In figure 3.36, the contours of the three regions of normalized thermal dissipation rate (white lines) have been demonstrated in a highlighted central section at the mid-spanwise plane of instantaneous temperature field, moreover the turbulent background region contours are plotted through the side wall areas to show that this region is dominated in these parts of the cavity, its found also to be located in the vicinities between the plumes. The thermal plumes region is showed clearly corresponding with the plumes, and the conductive sublayer region is located where highly gradient temperature existed so close to the isothermal walls (where the interactions between the caps and roots as explained before).
3.8. Thermal dissipation rate and thermal plumes

**Figure 3.35**: PDF of the normalized thermal dissipation rate $\varepsilon_T / <\varepsilon_T >_{t,v}$ in log-log scales, the data are taken of gathering a sequence of independent snapshots.

The general division of the domain is to bulk and thermal boundary layer

**Thermal dissipation rate in the boundary layer** We plot the PDF of $\varepsilon_T / <\varepsilon_T >_{t,v}$ in the thermal boundary layer $z = 0 \rightarrow \delta_t$ (see figure 3.37), a fat stretched tail can be observed reflecting the spatial intermittency effect in the boundary layer that is relatively greater than the one in the bulk because of the higher values of $\varepsilon_T$ in this region. This region coincides with the region 3 and follows an exponential scaling of the form

$$ P((\varepsilon_T / <\varepsilon_T >_{t,v}) > 10) = B_1 \exp(-B_2(\varepsilon_T / <\varepsilon_T >_{t,v})) $$  (3.22)

$B_1 = 2.2 \times 10^{-3}$ and $B_2 = -0.19$ we can note that the value of $B_1 \sim 1/\sqrt{RaPr} \equiv 2.423 \times 10^{-3}$ that is the non-dimensional thermal conductivity.

**Thermal dissipation rate in the bulk** Likewise, we plot the PDF of $\varepsilon_T / <\varepsilon_T >_{t,v}$ in the bulk region $z = 0.2H \rightarrow 0.8H$ (see figure 3.38). It found that this region contains all the small values of $\varepsilon_T$, the detailed view of this region shows a picture similar to the results obtained in passive scalar convection (see [46]), the probability density functions of the thermal dissipation rates are neither symmetric nor Gaussian. For very small $\varepsilon_T / <\varepsilon_T >_{t,v}$, the fitted Gaussian function falls below the PDFs (see figure 3.38.b where the proper proposed Gaussian distribution can be seen without plotting), reflecting strong intermittency effects of the small scales. The PDF of this region follows a stretched exponential function given as

$$ P(\varepsilon_T / <\varepsilon_T >_{t,v}) = \frac{A_1}{\sqrt{\varepsilon_T / <\varepsilon_T >_{t,v}}} \exp(-A_2(\varepsilon_T / <\varepsilon_T >_{t,v})^{\alpha/2}) $$  (3.23)
Chapter 3. Direct numerical simulation of laminar and turbulent Rayleigh-Bénard convection

Figure 3.36: (left) Contours of the three regions of $\frac{\varepsilon_T}{<\varepsilon_T>_{t,v}}$ as white lines in a highlighted central section at the mid-spanwise plane of instantaneous temperature field, top: conductive sublayer, middle: thermal plumes and bottom: turbulent background, (right) contours of turbulent background region in the side walls areas.

Figure 3.37: PDF of the normalized thermal dissipation rate $\frac{\varepsilon_T}{<\varepsilon_T>_{t,v}}$ in the boundary layer together with least-squares fits (purple line), the data are plotted in log-linear scales and again are taken of gathering a sequence of independent snapshots.
3.9. Conclusions

Which was analytically derived for passive scalar turbulence in the limit of large $Pr$ and pecket numbers by Chertkov, Falkovich & Kolokolov [50] and Gamba & Kolokolov [51] who found $\alpha = 2/3$. The coefficients obtained through least-squares fits are: $A_1 = 6.5 \times 10^{-3}$, $A_2 = -3.8$ and $\alpha = 1$. We should draw an attention to that all the statistics in all the region agree very well with the results of Kaczorowski & Wagner [34] who investigated higher $Ra$ numbers and found that $\alpha$ tends to approach the theoretically predicted passive scalar scaling at high $Ra$.

Figure 3.38: PDF of the normalized thermal dissipation rate $\varepsilon_T / < \varepsilon_T >_{t,v}$ in the bulk together with least-squares fits (purple line), the data are plotted in log-linear scale (a) and log-log scale (b), and taken of gathering a sequence of independent snapshots.

3.9 Conclusions

2-D and 3-D direct numerical simulations of laminar air flow in tilted rectangular cavity heated from below and cooled from above, have been studied at different aspect ratio and different boundary and initial conditions. The Nusselt number is calculated as a main parameter that limits the studied cases to reduce heat transfer by convection. The 2-D sights show that the $Nu_{2D}$ increases rapidly when $\Gamma_1$ changes from 0.1875 to 0.25 since the air goes from a throttle state to more free one. No influence of initial temperature was found in ours 2-D simulation while its noted in the 3-D simulation, its effect is limited in changing the rotation direction when the flow becomes three dimensional in the adiabatic side boundaries case. Bénard cells are formed along the horizontal periodic spanwise direction at particular distance and repeated at relative larger ones until become stable. To express the infinite general spanwise distance that represented by set the periodic side boundaries, its found that the minimum span width should be enough to formulate two completed rolls suited to $\Gamma_1$, which leads to permanent and not changed Nusselt number with increasing
the periodic cut distance, this span width is detected to be of aspect ratio $\Gamma_2 > 1$ when $\Gamma_1 = 0.25$ and $Ra = 2.47 \times 10^5$. Also found that the air tends to circulate in the inclination direction when more space is available (greater values of $\Gamma_1$), then begins to flow in the periodic third direction. Two arrangement design of the plastic cells in the air gap of the solar flat plate collector have proposed, in designing the small cells (adiabatic side boundaries) or the infinite parallel rows (periodic side boundaries). These configuration have been found so costly especially the high-temperature supported materials thus if we put a plastic plate horizontally in the air gap that could divide the $Ra$ number and consequently the $Nu$ number, can be a future idea of this work.

Afterward at very large aspect ratio $\Gamma_1 = 50$, the flow has been evolved to turbulent form. Thermal plumes as those appeared in the cases of turbulent Rayleigh-Bénard convection in no titled convective cell, have been detected in the center area and coherent circulations are clustered near the side walls. Temporal study have been performed to investigate the Gaussianity of the flow in different part of turbulent domain by finding the histogram skewness and kurtosis of the temperature at different heights between the isothermal walls. Its found high deviation from Gaussianity at the boundary layers and reduced gradually in the bulk to become zero at the center where a smaller correlation between the turbulent fields of temperature and velocity is detected comparing with the boundary layer areas. High amplitudes and intermitency of the thermal dissipation rate were found near the isothermal walls and the thermal plume regions, where the PDF statistics of $\varepsilon_T$ were found to be tail stretched exponential distribution, and the tail becomes fatter in the boundary layers compared with the bulk. A partitioning of the domain was presented according the PDF of the normalized thermal dissipation rate, to

1. Gaussian or turbulent background region (so small values of thermal dissipation rate distributed in the bulk (mixing zones) and side wall areas)
2. thermal plumes region (high relatively values of thermal dissipation rate distributed in the boundary layer and the bulk where the plumes arose (ridges) and propagated (caps))
3. conductive sublayer region (very high values of thermal dissipation rate located in the boundary layers so close to the isothermal walls in the areas of interactions between the arising roots and descending caps).

Similar observations of statistics were found in the bulk to those of the passive scalar convection. A study of mesh density and adequacy of the periodic side distance have done by testing the Grötbach criterion [40] that estimates the Kolmogorov dissipation scale and evaluating the two-points correlation to verify the satisfactory of the periodic distance, its found that $\Gamma_2 = 4$ is sufficient and the used mesh is enough to resolve the smallest relevant turbulent scales where the one-dimensional energy spectra in the periodic direction at a center point was performed and detected the Kolmogorov law $-5/3$ in velocity energy spectra and Bolgiano exponent $-7/5$ in the temperature one but its so difficult to distinguish.
Chapter 4

Conclusions

This work can be classified as a practical application of using the computational capacities in broadening the theoretical comprehension available about the physical phenomena of heat transfer and fluid motion. It offers a great aid in studying the complex phenomena that evolve a lot of difficulties in experiment studies like the accuracy and the small scale motion flow, furthermore it helps in improving the available designs of Industrial applications without losing a lot of money consumed in fabricating the models.

The so famous Navier-Stokes equations that govern the flow motion and the first principle of thermodynamic that governs the conservation of energy, have been numerically solved over small control volumes that discretise the studied domain. The finite volume method (FVM) has been utilized as a main method in all the work, and the governing equations have been resolved explicitly using the fractional step methods. First-order and second-order numerical scheme dissipative and conservative for the kinetic energy have been implemented in evaluating the non-linear convective term of Navier-Stokes equations, and different solvers for large linear system equations like Poisson equation have been used in 2-D and 3-D views using one CPU and more in parallel mode.

Primarily a forced laminar driven flow without heat exchange has been numerically simulated in two-dimensional sense in the driven cavity and backward facing step problems. They can present basically two models of the internal and external (over object) incompressible flow. Afterward, a natural driven flow by buoyancy forces has been numerically simulated in laminar two dimensional mode in a square differentially heated cavity and in turbulent mode in a cavity of aspect ratio 4. They present so wide practical models in engineering applications of heating and cooling fields. All these simulation has been performed by own developed code that is verified against the Benchmark and the literature results with good agreement. Later an improvement of the code to solve three dimensional cases and simulate a turbulent Rayleigh-Bénard convection in a cavity of aspect ratio 0.5 has done with validation to the code against the Benchmark. However the limits of using just one CPU prevented using smoother grids and the parallelization was the solution. A simple parallel code has performed to solve huge linear system equations by more than one CPU, applying the MPI parallel mode.

Understanding the phenomena of Bénard cells in laminar and turbulent regimes and its widely appearance in many engineering applications like the solar collector, was the main aim of this work where a parallel special code developed by the center (Heat and Mass transfer tecnological center) has been used to perform three dimensional numerical simulation of the air flow in inclined cavity heated from below and cooled from above at various aspect ratios. It represents a direct simulation for improving the design of the plastic honey comb cells that installed in the flat plate solar collectors, where high thermal resistance material can be used and installed in relatively small volume. The evolution of Bénard cells has been
noted and efficient constrains in investigating the Bénard flow have been presented like the sufficient periodic distance and the effect of the initial temperature. On the other hand, two arrangement design of the plastic cells have been proposed, in designing the small cells (adiabatic side boundaries) or the infinite parallel rows (periodic side boundaries). However, they have been found so costly and placing a plastic plate horizontally in the air gap that divides the flow can be a future work in damping the air motion and reduce the heat losses by convection.

A turbulent Rayleigh-Bénard convection has evolved just by increasing the aspect ratio. Thermal mushroom-like plumes of relatively large scales have been arisen as a turbulent form of Bénard cells. The thermal dissipation rate as a main important parameter in turbulent Rayleigh-Bénard heat transport convection, has been studied in details where its found to be following the passive scalar behavior in the bulk region and presents high intermittent magnitudes reflecting the small-scale intermittency that has its importance in DNS turbulent studies. Its found that the thermal dissipation rate has its highest values near the isothermal walls and the side wall regions and upon its values the thermal plumes have been extracted and separated from the turbulent background area. Studying cases with higher Rayleigh number at this configuration can be a so efficient future advancing work in investigating the turbulent Rayleigh-Bénard convection that have many applications in solar energy field.
Bibliography


[43] Peter S. Pacheco “A User’s Guide to MPI”.


Appendix A

Turbulent three dimensional Rayleigh-Bénard convection

As an extension of the 2-D simulation of turbulent flow in differentially heated cavity, we present here a 3-D simulation of Rayleigh-Bénard convection turbulent flow in a closed cavity of aspect ratio $\Gamma = L/H = 0.5$. The cavity is heated from below and cooled from the top with adiabatic lateral walls, the Rayleigh number is $Ra = 2 \times 10^9$, and the Prandtl number equals to 0.71, a second-order symmetry preserving scheme reported by R.W.C.P. Verstappen and A.E.P. Veldman [8] is used in the 3-D finite volume method of numerical simulation, comparisons with the benchmark results of 3-D cubic cavity presented by E. Tric, G. Labrosse, M. Betrouni [38] who used the Chebyshev pseudo-spectral algorithm, at Rayleigh number $Ra = 10^3$, has done and found an acceptable agreement to validate the code used.

Non-dimensional form of Navier-Stokes and energy equations (eq.A.2 and eq.A.3) are approximately solved using the fractional step methods in structured staggered grids, the overall Nusselt number at the hot wall was evaluated, preconditioned conjugate gradient algorithm [37] is used to solve the Poisson equation that takes the form $A^{3d} \cdot x^{3d} = b^{3d}$, where the coefficients matrix $A^{3d}$ is saved in sparse way to save memory requirements, the one-leg second-order-explicit time integration scheme (k1L2) with the EigenCDCD method proposed by F.X.Trias and O.Lehmkuhl [9] have been used to maximize the time step without losing the accuracy of the solution and this is done by adapting the stability domain in the time integration scheme and bound the eigenvalues of the dynamical system.

\[
\nabla \cdot \mathbf{u} = 0 \tag{A.1}
\]

\[
\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla)\mathbf{u} = \sqrt{\frac{Pr}{Ra}} \nabla^2 \mathbf{u} - \nabla p + \mathbf{f} \tag{A.2}
\]

\[
\frac{\partial T}{\partial t} + (\mathbf{u} \cdot \nabla)T = \frac{1}{\sqrt{PrRa}} \nabla^2 T \tag{A.3}
\]

All quantities are expressed in characteristic units. Length scales are normalized with respect to the height of the cavity $H$, velocities with respect to the free-fall velocity, $U = \sqrt{\beta g \Delta T H}$, time scales with respect to $H/U$, the temperature is scaled by the difference between the hot and cold walls $\Delta T = T_h - T_c$, and the pressure with respect to $\rho U^2$. Rayleigh number is evaluated like $Ra = \beta g \Delta T H^3/\nu \alpha$ and the Prandtl number as $Pr = \nu/\alpha$, where $g$ is the gravity, $\nu$ the kinematic viscosity, $\beta$ the volumetric thermal expansion coefficient and $\alpha$ the thermal diffusivity. $\mathbf{u}$, $p$, $T$ and $t$ are dimensionless velocity vector, pressure, temperature and time respectively, the body forces vector $\mathbf{f} = (0, 0, T)$. 

87
Appendix A. Turbulent three dimensional Rayleigh-Bénard convection

<table>
<thead>
<tr>
<th></th>
<th>present work mesh 32x32x32</th>
<th>present work mesh 62x62x62</th>
<th>benchmark Ref. [38]</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \overline{N_u} )</td>
<td>1.0713</td>
<td>1.0712</td>
<td>1.070</td>
</tr>
<tr>
<td>( N_u )</td>
<td>1.424</td>
<td>1.423</td>
<td>–</td>
</tr>
<tr>
<td>( z )</td>
<td>0.0918</td>
<td>0.0979</td>
<td>–</td>
</tr>
<tr>
<td>( N_u ) min</td>
<td>0.726</td>
<td>0.727</td>
<td>–</td>
</tr>
<tr>
<td>( z )</td>
<td>1</td>
<td>1</td>
<td>–</td>
</tr>
<tr>
<td>( u_{max} )</td>
<td>3.526</td>
<td>3.530</td>
<td>3.543</td>
</tr>
<tr>
<td>( (x, y, z) )</td>
<td>(0.473, 0.528, 0.815)</td>
<td>(0.486, 0.514, 0.825)</td>
<td>(0.5166, 0.5, 0.817)</td>
</tr>
<tr>
<td>( v_{max} )</td>
<td>0.165</td>
<td>0.174</td>
<td>0.173</td>
</tr>
<tr>
<td>( (x, y, z) )</td>
<td>(0.528, 0.733, 0.528)</td>
<td>(0.514, 0.744, 0.514)</td>
<td>(0.5, 0.7521, 0.5)</td>
</tr>
<tr>
<td>( w_{max} )</td>
<td>3.511</td>
<td>3.535</td>
<td>3.544</td>
</tr>
<tr>
<td>( (x, y, z) )</td>
<td>(0.815, 0.528, 0.473)</td>
<td>(0.825, 0.514, 0.486)</td>
<td>(0.8223, 0.5, 0.5032)</td>
</tr>
</tbody>
</table>

Table A.1: Comparison with the benchmark results in cubic 3-D differentially heated cavities.

The origin of Bénard cells starts when a long thin horizontal fluid layer heated from below, has a non-uniform temperature distribution with a static vertical temperature gradient large enough to develop a form of convection instability. An arbitrary fluctuation takes a place and a small parcel of hotter fluid than the neighboring experiencing a buoyancy force begins to rise, the fluid ascends from underneath to fill the void left by the rising part inducing an amplification of temperature fluctuation, as the fluid coming up from under the rising parcel is from the bottom and is hotter than the fluid above. By drawing up more fluid from the hot region, the original temperature fluctuation is amplified and the plume formulated of rising fluid becomes stronger with time.

As the temperature difference between the effective walls exceeds at high Rayleigh number, the cellular flow becomes more complicated and the two dimensional rolls break up into three dimensional cells that develop to mushroom-like forms, these mushroom-like are the thermal plumes that can be identified as fragments of the thermal boundary layer that detach permanently and move into the bulk, their stems have relatively high amplitude contributions of the local heat flux and the vertical velocity field, also they can be assigned with high-amplitude events of thermal dissipation rate. The plumes are generated in the thermal boundary layer close to the bottom or the top wall and are driven to the opposite one by buoyancy, they transform their thermal energy into kinetic energy in the bulk region forming a connected network of heat transport across the whole cavity.

At higher Rayleigh number the event of ejection plumes from the boundary to the core regions has larger frequency and smaller plumes are ejected rapidly leading to break down unstable spokes to form more independent large-scale flow structures driven by buoyancy from the boundary to the core.

The Nusselt number has been detected about \( \overline{N_u} = 80.28 \) that is close to the results of \( Nu \) reported by I. Rodríguez, O. Lehmkul, R. Borrell and C.D. Pérez-Segarra [39] who studied the same case but with cylindrical configuration.

The grid size in the bulk must establish the kolmogorov turbulent scale that can be estimated according to the Grötzbach criterion [40] for turbulent Rayleigh-Bénard convection by the correlation:

\[
\eta_v \leq \pi \sqrt{Pr/(NuRa)}^{1/4}
\]

for \( Pr \leq 1 \)

\[
\eta_v = \text{max} \sqrt[4]{\Delta x \Delta y \Delta z}
\]
\( \Delta x, \Delta y, \Delta z \) are the maximum values of the control volume dimensions. In our case and by using the Niemela \[41\] approximation to evaluate the Nusselt number \( \text{Nu} = 0.124 R^0.309 \), \( \eta_v = 4 \times 10^{-3} \) while we have \( \eta_{DNS} = 0.02 \) and we will need more nodes to resolve the smallest vortices, whereas our goal in this work is implementing the symmetry preserving scheme in three dimensional coordinates and understanding -through an essentially view- the turbulent flow in Rayleigh-Bénard convection.

<table>
<thead>
<tr>
<th>Ra</th>
<th>( N_x )</th>
<th>( N_y )</th>
<th>( N_z )</th>
<th>( \gamma_x )</th>
<th>( \gamma_y )</th>
<th>( \gamma_z )</th>
<th>Total time [TU]</th>
<th>Average time [TU]</th>
<th>Symmetry scheme</th>
</tr>
</thead>
<tbody>
<tr>
<td>2 \times 10^9</td>
<td>62</td>
<td>62</td>
<td>102</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>3100</td>
<td>1320</td>
<td>2\textsuperscript{nd}</td>
</tr>
</tbody>
</table>

**Table A.2:** Nusselt number and the used parameters of work.

The averaged temporal operator has implemented to get the structure patterns in the statistically stationary state, where the averaged patterns of temperature and the second order statistics as the velocity and temperature variance, turbulent kinetic energy and turbulent heat flux, through the two diameter planes, are shown in figures A.1, A.3 respectively.

A symmetric arrangement over the diameter line of the diagonal plane, has been noted in all the first and second order averaged patterns, the averaged stream lines also exhibits a symmetry over the diameter diagonal lines and the flow chooses one diagonal plane to rotate by its maximum velocity suffering of small symmetric inclination (see figure A.2).

**Figure A.1:** Averaged temperature maps taken at two opposite sides (a), and in the two diagonal planes (b).
Figure A.2: Averaged stream lines maps taken at four opposite sides.

Its noted that the thermal plumes are dragged with the high fluctuation values of the vertical velocity that takes its maximum values in the diagonal planes (see figure A.3 $<u'u'>$, $<w'T'>$) making the high turbulent heat flux to be dominated diagonally and takes its low values in the core region.

The main form of flow presents high fluctuation of kinetic energy in the diagonal planes allocated in the opposite sides of one diameter. More complex form of fluctuation is noted in the core region for all the statistics, the temperature represents its high variance at the limit of the thermal boundary layer near the corners of the cavity.

Thermal dissipation rate plays a central role in mechanisms of heat transport in turbulent Rayleigh-Bénard convection, its value indicates the magnitude of gradients of the turbulent temperature field by

$$\varepsilon_T = \alpha (\nabla T)^2$$  \hspace{1cm} (A.4)

Where $T$ is the dimensional temperature, the ensemble (space-time average) of the thermal dissipation rate field can be directly related to $Nu$ number, this relation is given by

$$<\varepsilon_T>_{v,t} = \alpha \frac{\Delta T^2}{H^2} Nu$$  \hspace{1cm} (A.5)

Now in non-dimensional form, the thermal dissipation rate can be derived as

$$\varepsilon_T = \alpha \frac{(\Delta T)^2}{H^2}(\nabla T)^2 = \frac{(\Delta T)^2 U}{H} \frac{1}{\sqrt{RaPr}} (\nabla T)^2$$  \hspace{1cm} (A.6)

which gives

$$\varepsilon_T = \frac{1}{\sqrt{RaPr}} (\nabla T)^2$$  \hspace{1cm} (A.7)

Figure A.4 shows the probability density function (PDF) of the normalized thermal dissipation rate $\varepsilon_T/ <\varepsilon_T>_{v,t}$ in log–log scales, where $<\varepsilon_T>_{v,t}$ from eqs. A.5 and A.7 can be evaluated as

$$<\varepsilon_T>_{v,t} = \frac{Nu}{\sqrt{RaPr}}$$  \hspace{1cm} (A.8)
Figure A.3: Second-order averaged statistics patterns through two diagonal cross sections.

It can be noted the Gaussian distribution in the lower values of thermal dissipation rate that located in the bulk region where the thermal plumes are highly mixed forming the turbulent background areas in the core, the deviation part from Gaussian distribution presents
the ascended and descended plumes that detach from thermal boundary layers at the cold and hot plates where the large values of $\epsilon_T$ represents those layers. It found that spatial intermittency of the thermal dissipation rate increases toward the top and bottom boundary layers as its cleared in fatter stretched exponential tail of the (PDF).

Figure A.4: Probability density function of the thermal dissipation rate $\epsilon_T$ normalized by $<\epsilon_T>_{v,t}$ in log – log scales, the data are taken from one snapshot.
Appendix B

MPI parallel implementation for solving large linear systems

B.1 Introduction

Many experts believe that now we are close to the upper performance limits that can be achieved by single processor computers, since the growing needs to faster computational processing, have been a regarded importance, and the one or two CPUs working becomes not effective and requires very long run time such as weeks or months. Furthermore the requirements of memory in CFD field as DNS simulation of turbulent flow where the mesh could be millions, become so huge and with just one CPU is almost impossible to achieve. The main obstacle in DNS simulation and other fields of engineering application like weather prediction and electronic network analysis, is encountered in solving very large systems of linear equations that takes the biggest portion of runtime.

These requirements attained a big attention to carry out the computational calculations simultaneously, operating on the principle that large problems can often be divided into smaller ones, which are then solved in parallel mode and the total required memory can be distributed between many CPUs. Parallel computers can be roughly classified according to the level at which the hardware supports parallelism, with multi-core and multi-processor computers having multiple processing elements within a single machine, while clusters, MPPs, and grids use multiple computers to work on the same task. Specialized parallel computer architectures are sometimes used alongside traditional processors, for accelerating specific tasks.

Communication and synchronization between the different subtasks are typically some of the greatest obstacles to getting good parallel program performance, the most recommended communication way is using MPI or the message passing interface. The MPI programming model, as its name implies, is based on message passing. In a message-passing system, different concurrently-executing processes communicate by sending messages from one to another over a network. Unlike multi-threading, where different threads share the same program state, each of the MPI processes has its own, local program state that cannot be observed or modified by any other process except in response to a message. Therefore, the MPI processes themselves can be as distributed as the network permits, with different processes running on different machines or even different architectures. MPI implementations typically contain optimized algorithms for collective operations that take advantage of knowledge of the network topology and hardware, even taking advantage of hardware-based implementations of some collective operations.

Returning to the essential problem, that is solving the linear equation $A \cdot x = b$, where
the size of $A$ could be millions, even supercomputers takes a prodigious amount of time to find a sufficiently accurate solution vector. Many different linear system solution methods have been invented, some of these solution techniques are hundreds of years old, whereas, others have been created fairly recently. Each algorithm has its own strengths and weakness, one of the earliest routines is called Cramer’s rule which is a direct solution method, Gaussian elimination with back substitution, LU factorization and Gauss-Jordan reduction are three of the most popular direct solution methods too, with the direct procedures, the programmer can know in advance how many additions and multiplications will be required to solve a given problem, they change the linear system into an equivalent linear system of a form that can be more easily solved than the original equations, the changes in the original system are usually quite extensive. An infinite precision arithmetic can never be attained for large and/or ill-conditioned linear systems, and all the direct solution methods suffer from round-off error due to the finite precision arithmetic available on all computers. The indirect solution methods are a group of algorithms that start off with an initial guess of solution vector and progressively refine this estimated solution using iterative process. Jacobi iteration, Successive over-relaxation and Gauss-Seidel iteration are three of the better known indirect methods, the iterative techniques do not converge to the true solution in all instances, some of these procedures guarantee convergence only for certain types of linear systems as for Jacobi and Gauss-Seidel algorithms which converge only for linear system has diagonally dominant coefficients matrix $A$ that means the magnitude of the diagonal element in each row is greater than the sum of the magnitudes of all other elements in its row. Other recent methods is the Conjugate Gradient algorithm and Preconditioned Conjugate Gradient that is one of the best known iterative techniques for solving sparse symmetric positive definite linear systems. The projection iterative methods also, which use a canonical way for extracting an approximation to the solution of a linear system from a subspace, and various other methods [42], the iterative methods are usually slower than the direct techniques, but the iterations are self-correcting so that infinite precision arithmetic is not required for acceptably accurate answers, for this reason, the indirect procedures are usually preferred (where they can be applied) for large and/or ill-conditioned problems.

In this work, an application of using MPI parallelization for solving a group of integrated diffusive heat transfer equations by conduction, as a sample of large linear system problem, is represented, we illustrate two forms of parallelizations supported by figures and use Jacobi and Gauss-Seidel iterative algorithms in finding the solution as a general and wide popular used algorithms. Problems of deadlock have been arisen and solved in distinct decomposition of the domain.

B.2 problem description and Domain decomposition

Two dimensional heat transfer per conduction with vertical effective lateral boundaries and adiabatic horizontal boundary conditions problem, is one example of large linear system of form $A \cdot x = b$, once eq. B.1 is integrated partially over a high number of control volumes.

$$\rho c_p \frac{\partial T}{\partial t} = \frac{\partial}{\partial x} \left( \lambda \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y} \left( \lambda \frac{\partial T}{\partial y} \right) \quad (B.1)$$

Assuming that the heat transfer is independent of time that means we want to get the solution of steady state. The integrated form of eq.B.1 becomes as:

$$\lambda \Delta y \left( \frac{T_E - T_P}{\Delta X_e} - \frac{T_P - T_W}{\Delta X_w} \right) + \lambda \Delta x \left( \frac{T_N - T_P}{\Delta Y_n} - \frac{T_P - T_S}{\Delta Y_s} \right) = b \quad (B.2)$$

(see figure B.1.b ), after rearranging the equation it becomes:

$$a_P T_P = a_E T_E + a_W T_W + a_N T_N + a_S T_S + b \quad (B.3)$$
where \( b \) equals \( T_w_1 \), \( T_w_2 \) at the lateral boundaries, and equals \( T_S \), \( T_N \) at the horizontal up and down boundaries respectively. The coefficients take values:

\[
\begin{align*}
    a_E &= \lambda \frac{\Delta y}{(\delta X)_e}, \\
    a_W &= \lambda \frac{\Delta y}{(\delta X)_w}, \\
    a_N &= \lambda \frac{\Delta x}{(\delta Y)_n}, \\
    a_S &= \lambda \frac{\Delta x}{(\delta Y)_s}, \\
    a_P &= a_E + a_W + a_N + a_S
\end{align*}
\]

finally eq.B.3 for the whole domain can be expressed as a linear system of form: \( A \cdot x = b \), where \( A \) is the coefficients matrix. \( b \), and \( x \) are the known and unknown vectors respectively.

## B.3 Domain decomposition

Solving this linear system on just one CPU for millions of equations, takes long run time and requires a great capacity of memory related to the kind and how speed is the solution algorithms used. Thus, dividing the problem into smaller ones and solve them simultaneously with implementing communication techniques among them in parallel mode could be a good solution for saving time and computational cost in general.

Each CPU works on a part of the domain that is extended in order of one control volume creating the halos nodes in each communicate direction (see figure B.1.a). The number of nodes (control volumes) is distributed between the processors in a way that all the CPUs receive the same number of nodes initially and then the rest of the division is distributed over the CPUs starting from the zero rank processor where the rank is the number of processor, in this way all processors receive almost the same load with a difference no more than one node.

Halo nodes in each subdomain support the data of neighbors to be used in this last. A uniform mesh is used to facilitate the partitioning of the domain and provide previous information of the halos size.

### One dimensional parallelization

The partitioning of the domain has done in just one direction \((x)\). Just three types of subdomains exist :inner, boundary left and boundary right (see figure B.2.a), the information are transported in just one direction that means all the processors have two neighbors at the left and the right except the zero and \( N_p - 1 \) rank processor, where the \( N_p \) is the number of processors or CPUs.

All the processors should have two integers that decide the rank numbers of neighbors (left...
two dimensional parallelization In this kind, the partitioning will be in two directions ($x$) and ($y$), nine types of subdomains arise: four types at the four edges, another four types each one between two adjacent edges and one internal type in the center area (see figure B.2.b), the information will be exchanged in four directions (left, right, up and down).

The processors will be distributed over the domain and each one should have its own type according the location, consequently each rank processor has a type (an integer) that decides the rank number of neighbors by four integers and defines the information according the location by four integers also (is boundary or not).

![Diagram of domain decomposition](image)

(a) One dimensional parallelization and the basic three types of subdomains.

(b) Two dimensional parallelization and the basic nine types of subdomains.

**Figure B.2:** Domain decomposition.

Jacobi and Gauss-Seidel iterative solution algorithms As mentioned in part 1, that the direct algorithms change the linear system into a form that can be more easily solved and this change is usually quite extensive. Furthermore the direct solution methods have a round-off error due to the finite precision arithmetic available on computers, whereas the iterative methods are self-correcting and the infinite precision arithmetic is not required for acceptably accurate solutions, but they are slower than direct methods. In reality there are many particularly fast linear systems algorithms that are suited only for special types of matrix $A$, such as sparse matrices, sparse symmetric and positive definite, banded systems and others. These algorithms are not suitable for use on problems with more random or dense structures. Programs for general linear systems are available, but tend to be slower than specialized algorithms. Concurrent processing, should alleviate the speed problem of the general programs. Since a high degree of accuracy was also required, it was decided that an indirect solution method should be used in preference to a direct method. Another requirement of the parallel linear systems program was that it should be as simple and straightforward running as possible.

The iterative Gauss-Seidel and Jacobi algorithms are implemented here as a good solution methods can be use primarily. In each subdomain (as CPU1) (see figure B.1.a), the results of unknowns at the decomposition limit are transported to fill the halo nodes of the next subdomain that shares the same limit (CPU2), which in turn sends its results to fill the halo nodes of the previous subdomain (CPU1). In the following iteration the halos nodes in
B.3. Domain decomposition

each subdomain will be used without any modification, to evaluate the internal nodes at the decomposition limit and thereby the two CPUs will have been communicated. The results can be sensed and received among CPUs as messages, that have their own distinguishing tags and this is done by using the routines MPI_Send and MPI_Recv. MPI_Allreduce routine is used to collocate the error from all CPUs and find the maximum one each iteration (for information about the MPI routines the reader is referred to [43]).

In Jacobi iteration, any newest iterated variable value should not have been used in the immediately following iteration and wait until the end of a complete iteration cycle utilizing the values of the previous iteration, while in Seidel iteration, the recent iterated variable value is used in the immediately running iteration to improve the estimated solution and consequently use less number of iterations.

If we denote the unknown recent variable $T$ as in our example, and the unknown variable of the previous iteration as $T^*$ the Jacobi and Gauss-Seidel algorithms can be implemented as:

<table>
<thead>
<tr>
<th>Jacobi algorithm</th>
<th>Gauss-Seidel algorithm</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\varepsilon = 10^{-6}$</td>
<td>$\varepsilon = 10^{-6}$</td>
</tr>
<tr>
<td>$T_P = (a_E T^<em>_E + a_W T^</em>_W + a_N T^<em>_N + a_S T^</em>_S + b)/a_P$</td>
<td>$T_P = (a_E T_E + a_W T_W + a_N T_N + a_S T_S + b)/a_P$</td>
</tr>
<tr>
<td>if : $</td>
<td>T_P - T^*_P</td>
</tr>
<tr>
<td>repeat with : $T^*_P = T_P$</td>
<td>repeat with : $T^*_P = T_P$</td>
</tr>
<tr>
<td>else finish</td>
<td>else finish</td>
</tr>
</tbody>
</table>

Table B.1: Gauss-Seidel and Jacobi algorithms.

It can be observed in table B.1, that the number of iterations become bigger with using more CPUs in Seidel method and that by reason of the information that transported between two subdomains in one iteration and then will be used in the next iteration after updating, they could have been transported and updated is just one iteration in the case of the two subdomains were one combined subdomain. In the Jacobi method the number of iterations does not change because no iterated variable values are used in the recent iteration, just uses the result values of iteration anterior, therefore it does not matter how many subdomains are used.

<table>
<thead>
<tr>
<th>number of CPUs</th>
<th>number of iterations by Gauss-Seidel</th>
<th>number of iterations by Jacobi</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>124364</td>
<td>229169</td>
</tr>
<tr>
<td>8</td>
<td>124792</td>
<td>229169</td>
</tr>
<tr>
<td>16</td>
<td>125218</td>
<td>229169</td>
</tr>
<tr>
<td>32</td>
<td>126074</td>
<td>229169</td>
</tr>
<tr>
<td>64</td>
<td>126933</td>
<td>229169</td>
</tr>
</tbody>
</table>

Table B.2: Comparison between Gauss-Seidel and Jacobi iterative solvers for the 2-D simulation using mesh of 256x256.

The contour map of temperature in the two parallelization modes, are presented in figure B.3 as results of our example.
The basic blocking send and receive routines MPI_Send, MPI_Recv are used in the one dimensional parallelization. In these routines the program execution will be suspended until the message buffer is safe to use, no problems of deadlock appear by following a good order of send and receive messages (the boundary processors send first and the internal receive).

In the two dimensional parallel programing, the problem of deadlock arises because of exist two directions of exchange concurrently and the order of send and receive could not be possible, to solve this problem two ways are proposed: one way by set the even rank processors to be sending first during the odd ones are receiving at the same time, this method proposes that all the adjacent processors are in odd rank numbers as the main is in even one, sometimes and at four decomposition of the domain two adjacent subdomains will be in even rank and the send and receive operations will be idle that means, one processor is waiting for a message from another one that never gets sent, in this case another method can be applicable regardless the send or receive buffer is empty and that by using the non-blocking send and receive routines MPI_Isend, MPI_Irecv. These routines return almost immediately and don’t wait for any communication events to be complete, like overlapping the communications with computation. They are more faster and here need MPI_Wait routine for organizing the send and receive operations.

Figure B.3: Contour map of temperature in the two parallel mode decompositions.
List of Figures

1.1 numerical Quick scheme. ........................................ 10
1.2 maps of the streamlines in the cavity at $Re = 100, Re = 400$. ........ 10
1.3 maps of the streamlines in the cavity at $Re = 1000, Re = 3200$. ........ 10
1.4 maps of the streamlines in the cavity at $Re = 5000, Re = 7500$. ........ 11
1.5 maps of the streamlines in the cavity at $Re = 10000$. .................... 11
1.6 Values of $u$ along the vertical middle line of the cavity at Re=100. .... 12
1.7 Values of $v$ along the horizontal middle line of the cavity at Re=100. ... 12
1.8 Values of $u$ along the vertical middle line of the cavity at Re=1000. .... 13
1.9 Values of $v$ along the horizontal middle line of the cavity at Re=1000. ... 13
1.10 Values of $u$ along the vertical middle line of the cavity at Re=5000. ... 14
1.11 Values of $v$ along the horizontal middle line of the cavity at Re=5000. ... 14
1.12 Backward-facing step channel geometry with the definition of separation length $x_4$ and the reattachment lengths $x_1, x_5$. ........................................... 16
1.13 Refined mesh $100x90$. ........................................... 17
1.14 Streamwise velocity profiles at the $z = 0$ symmetric plane for the cases of (a) $Re=100$, (b) $Re=389$. ........................................... 18
1.15 Streamwise velocity profiles at the $z = 0$ symmetric plane for the case of $Re=1000$. ........................................... 18
1.16 A comparison of the present and the Guerrero and Cotta two-dimensional streamwise velocity profiles for the case of $Re=1000$. ........................................... 19
1.17 Streamline maps for cases of Reynolds number (a)$Re=100$, (b)$Re=389$, (c)$Re=800$, (d)$Re=1000$. ........................................... 20
2.1 30x30 nodes. ........................................... 22
2.2 Rayleigh number $Ra = 10^3$ and mesh $80x80$. ........................................... 25
2.3 Rayleigh number $Ra = 10^4$ and mesh $80x80$. ........................................... 26
2.4 Rayleigh number $Ra = 10^5$ and mesh $60x60$. ........................................... 27
2.5 Rayleigh number $Ra = 10^6$ and mesh $80x80$. ........................................... 28
2.6 The deviation error of $Nu_{2D}$ from Benchmark results at different meshes and different $Ra$. ........................................... 29
2.7 Rayleigh number $Ra = 10^3$, $Ra = 10^4$ in mesh $80x80$. .................... 29
2.8 Rayleigh number $Ra = 10^5$ in mesh $80x80$, $Ra = 10^6$ in mesh $100x100$. 30
2.9 Schema of two-dimensional differential heated cavity of aspect ratio 4 with adiabatic horizontal walls. ........................................... 32
2.10 Symmetry-preserving discretization method. ........................................... 34
2.11 Averaged temperature (left) and stream line (right) at: (a) $Ra = 6.4x10^{8}$, (b) $Ra = 2x10^{9}$, (c) $Ra = 10^{10}$. ........................................... 39
2.12 Averaged temperature (left) and vertical velocity (right) profiles at $y = 0.1, 0.2, 0.3, 0.4, 0.5, 0.6, 0.7, 0.8, 0.9$. Each vertical subdivision represents 0.5 units of temperature and 0.2 units for vertical velocity. For plots on the left, the abscissa scale factor is $4x$ and for plots on the right, is $4xRa^{1/4}$, $Ra$ values as figure 2.13.d. ...................................... 39

2.13 Local Nusselt number distribution at the hot wall: (a) $Ra = 6.4x10^8$, (b) $Ra = 2x10^9$, (c) $Ra = 10^{10}$ and the averaged temperature profile at the mid-length plane (d). ......................................................... 40

2.14 Isothermal instantaneous in the stationary state for: (a) $Ra = 6.4x10^8$, (b) $Ra = 2x10^9$, (c) $Ra = 10^{10}$. ................................................................. 41

2.15 Isothermal instantaneous snapshots for $Ra = 6.4x10^8$. ................................. 42

2.16 Isothermal instantaneous snapshots for $Ra = 2x10^9$. .................................. 42

2.17 Isothermal instantaneous snapshots for $Ra = 10^{10}$. .................................... 42

2.18 Second order statistics at $Ra = 6.4x10^8$ from left to right: $K$, $<T'T'>$, $<v'v'>$ , $<v'T'>$, $<v'u'>$. ................................................................. 43

2.19 Second order statistics at $Ra = 2x10^9$ from left to right: $K$, $<T'T'>$, $<v'v'>$ , $<v'T'>$, $<v'u'>$. ......................................................... 43

2.20 Second order statistics at $Ra = 10^{10}$ from left to right: $K$, $<T'T'>$, $<v'v'>$ , $<v'T'>$, $<v'u'>$. ......................................................... 44

2.21 Horizontal profiles at $y = 0.8$ of turbulent kinetic energy $K = <u'u'> + <v'v'>$ with scale multiplied by $10^{-3}$: (a) $Ra = 6.4x10^8$, (b) $Ra = 2x10^9$, (c) $Ra = 10^{10}$. ......................................................... 44

3.1 Discretization mesh, (a) 2-D, (b) 3-D. ................................................................. 50

3.2 $Nu_{local}$ along the isothermal hot plate (a) and $Nu_{2D}$ number (b) at all the aspect ratio values. ................................................................. 53

3.3 Temperature maps at aspect ratios $\Gamma_1 = \{0.0625, 0.125, 0.1875, 0.25, 0.5, 0.75, 1\}$ from left to right respectively. ........................................ 53

3.4 Stream line maps that correspond to figure 3.3. ............................................... 53

3.5 (a) profiles of $w$ velocity along $x$ direction from the right adiabatic end side till the mid-length plane, (b, down) Temperature profiles in the boundary layer of height 0.1H, (b.up) profiles of $u$ along $z$ direction from the hot plate till the mid-height plane (the numbers beside lines refer to $\Gamma_1$). ......................... 54

3.6 Streamlines at configurations $\Gamma_1 = 0.25$ and $\Gamma_2$ varies to: (a) 0.125 (b) 0.25 (c) 0.5 and $T^0=0$ (d) 0.5 and $T^0=0.5$ (e) 0.5 diagonal side, (f) 0.5 opposite diagonal side. ................................................................. 54

3.7 Isothermal planes at configurations $\Gamma_1 = 0.125$ and $\Gamma_2$ varies to: (a) 0.125 (b) 0.25 (c) 0.375 (d) 0.5. ................................................................. 56

3.8 Isothermal planes at configurations $\Gamma_1 = 0.1875$ and $\Gamma_2$ varies to: (a) 0.1875 (b) 0.375 (c) 0.5625. ................................................................. 57

3.9 Isothermal planes at configurations $\Gamma_1 = 0.25$ and $\Gamma_2$ varies to: (a) 0.125 (b) 0.25 (c) 0.5 (d) 0.75. ................................................................. 57

3.10 Bénard cells at configurations $\Gamma_1 = 0.25$ and $\Gamma_2$ varies: (a) 4, (b) 1, (c) 0.75 with periodic side boundaries. ......................................................... 58

3.11 Isothermal planes at configurations $\Gamma_1 = 0.25$ and $\Gamma_2$ varies to: (a) 0.25, 0.5, 0.75, 1 (b) 1.5, 2 (c) 3 (d) 4. ................................................................. 59

3.12 Isothermal planes at configurations $\Gamma_1 = 0.0625$, $\Gamma_2 = 2$. ......................... 60

3.13 Isothermal planes at configurations $\Gamma_1 = 0.125$ and $\Gamma_2$ varies to: 1 and 2. . . . 61

3.14 Isothermal planes at configurations $\Gamma_1 = 0.1875$ and $\Gamma_2$ varies to: 0.5, 1 and 2. . . . 61

3.15 Isothermal planes at configurations $\Gamma_1 = 0.5$ and $\Gamma_2$ varies to: 1 and 2. . . . 61

3.16 Two-points correlations at monitoring cavity locations (a) $P1(25, y, 0.5)$, (b) $P2(49, y, 0.05)$ and (c) $P3(46, y, 0.8)$. ......................................................... 64
3.17 One-dimensional energy spectra in the periodic y direction at center position (25, y, 0.5): (a)Temperature, (b)velocities. 64
3.18 Vertical velocity histogram at: (a)P1(25, 2, 0.5), (b)P2(49, 2, 0.05) and (c)P3(46, 2, 0.8) 66
3.19 Cumulative average statistics for temperatures (left) and its histogram (right), in the probe points: (a,b) P1(25, 2, 0.5), (c,d) P2(49, 2, 0.05) and (e,f) P3(46, 2, 0.8). 67
3.20 Averaged time and periodic direction of temperature maps in three highlighted zones: left (downstream), right (upstream) and bottom (middle). 68
3.21 Local averaged Nusselt number Nuavg profile at the hot wall. 68
3.22 Variance Temperature T maps in three highlighted zones: (downstream), right (upstream) and bottom (middle). 69
3.23 Turbulent kinetic energy < k' > maps in three highlighted zones: left (downstream), right (upstream) and bottom (middle). 69
3.24 The vertical component of turbulent heat flux < w'T' > maps in three highlighted zones: left (downstream), right (upstream) and bottom (middle). 69
3.25 Variance y velocity component < v'v' > maps in three highlighted zones: left (downstream), right (upstream) and bottom (middle). 70
3.26 Variance z velocity component < w'w' > maps in three highlighted zones: left (downstream), right (upstream) and bottom (middle). 70
3.27 Variance x velocity component < u'u' > maps in three highlighted zones: left (downstream), right (upstream) and bottom (middle). 70
3.28 Probability density function (PDF) of Temperature T at different heights of z (a), and the mean square of temperature fluctuations profile < T'T' > extracted along the vertical center line of 2-D time and periodic direction averaged results plane (b), the data of the temperature PDF is taken of gathering a sequence of independent snapshots. 71
3.29 High dependence of (a) the skewness and (b) flatness of the temperature fluctuations along a central mid-length mid-width line of the cavity (the vertical line through P1(25, 2, 0.5)). 72
3.30 Correlation coefficients of the turbulent fluctuations of the velocity and temperature fields at the same vertical line that define in figure 3.31. 74
3.31 Extracted instantaneous zoomed section of the hot thermal plumes at temperature range 0.5 → 1 (a), their stems 0.8 → 1 (b) and the corresponding thermal dissipation rate εT through a clip of height 0.3H (c). 75
3.32 Three contour plots of instantaneous temperature (up) and the corresponding thermal dissipation rate εT (down) at different three heights z = δt (top group), 0.5H (middle group) and H − δt (bottom group). 76
3.33 PDF of the normalized thermal dissipation rate εT/< εT >T,v in log−log scales, the data are taken of gathering a sequence of independent snapshots. 77
3.34 PDF of the normalized thermal dissipation rate εT/< εT >T,v in the boundary layer together with least-squares fits (purple line), the data are plotted in log−linear scales and again are taken of gathering a sequence of independent snapshots. 78
3.38 PDF of the normalized thermal dissipation rate $\varepsilon_T / < \varepsilon_T >_{t,v}$ in the bulk together with least-squares fits (purple line), the data are plotted in $\log - linear$ scale (a) and $\log - \log$ scale (b), and taken of gathering a sequence of independent snapshots. .............................. 79

A.1 Averaged temperature maps taken at two opposite sides (a), and in the two diagonal planes (b). ................................................................. 89
A.2 Averaged stream lines maps taken at four opposite sides. ................. 90
A.3 Second-order averaged statistics patterns through two diagonal cross sections. 91
A.4 Probability density function of the thermal dissipation rate $\varepsilon_T$ normalized by $< \varepsilon_T >_{v,t}$ in $\log - \log$ scales, the data are taken from one snapshot. .... 92

B.1 Information exchanging between two CPUs and resulted control volume discretization. ................................................................. 95
B.2 Domain decomposition. ................................................................. 96
B.3 Contour map of temperature in the two parallel mode decompositions. .... 98
List of Tables

1.1 A summary of the lengths $x_1$, $x_4$, and $x_5$. .............................. 17
2.1 The results at $Ra = 10^3$ ........................................................... 25
2.2 The results at $Ra = 10^4$ ........................................................... 26
2.3 The results at $Ra = 10^5$. ........................................................... 27
2.4 The results at $Ra = 10^6$. ........................................................... 28
2.5 Numerical simulation parameters. ............................................... 37
2.6 Summary of the averaged flow results. By rows from top to bottom, the magnitudes are: the overall averaged Nusselt number, the maximum averaged local Nusselt number and its position at the hot wall, the minimum averaged local Nusselt number and its position at the hot wall, the maximum averaged horizontal velocity $<u>_{max}$ and its position at the vertical mid-length plane, the maximum averaged vertical velocity $<v>_{max}$ and its position at the horizontal mid-height plane. ............................................ 37
2.7 Nusselt number and correlations. ............................................... 40
3.1 Comparison with the benchmark results in square 2-D differentially heated cavities at $Ra = 10^3$. ........................................................... 52
3.2 Comparison with the benchmark results in cubic 3-D differentially heated cavities at $Ra = 10^3$. ........................................................... 52
3.3 2-D numerical simulation results of natural convection in tilted cavity filled of air, at different values of aspect ratios $\Gamma_1$ and $Ra = 2.47 \times 10^5$. ........................................ 55
3.4 3-D numerical simulation results of natural convection in tilted cavity filled of air, at different values of aspect ratios $\Gamma_1$ and $\Gamma_2$, $Ra = 2.47 \times 10^5$ and adiabatic side walls. ........................................................... 55
3.5 3-D numerical simulation results of natural convection in tilted cavity filled of air at different values of aspect ratios $\Gamma_1$ and $\Gamma_2$, $Ra = 2.47 \times 10^5$ and periodic side boundaries. ........................................................... 58
3.6 Results $<Nu_{3D}>$ in different performed probes with the information of mesh size used where $\gamma_n$ is the concentration parameter in the corresponding direction $n$. ........................................................... 63
A.1 Comparison with the benchmark results in cubic 3-D differentially heated cavities. ........................................................... 88
A.2 Nusselt number and the used parameters of work. ........................................................... 89
B.1 Gauss-Seidel and Jacobi algorithms. ........................................ 97
B.2 Comparison between Gauss-Seidel and Jacobi iterative solvers for the 2-D simulation using mesh of 256x256. ........................................................... 97