

NUMERICAL SIMULATION OF NATURAL CONVECTION IN A DIFFERENTIALLY HEATED TALL ENCLOSURE USING A SPECTRAL ELEMENT METHOD

Diogo B. Pitz
John W. Chew

University of Surrey - Guildford, Surrey, GU2 7XH, United Kingdom
d.bertapitz@surrey.ac.uk, j.chew@surrey.ac.uk

Abstract. Natural convection in differentially heated enclosures is a benchmark problem used to investigate the physics of buoyant flows and to validate numerical methods. Such configurations are also of interest in engineering applications such as cooling of electronic components and air flow around buildings. In this work a spectral element method is used to carry out direct numerical simulations of natural convection in a tall enclosure of aspect ratio 4 with isothermal vertical walls and adiabatic horizontal walls. Spectral element methods combine the flexibility of classical finite element methods with the high accuracy and efficiency of single-domain spectral methods. The flow is solved in a three-dimensional domain with periodic boundary conditions imposed in the third direction. The numerical results are compared with solutions available in the literature and with numerical results obtained using a commercial software that employs a low-order finite volume method. Good agreement with previous work is obtained for the value of the Rayleigh number investigated, $Ra = 2.0 \cdot 10^9$, which is greater than the critical value of Ra where transition to an unsteady, chaotic state is known to occur. The results are presented in terms of the time-averaged flow structure, Reynolds stresses and modal energies. Although the time-averaged velocity and temperature fields obtained with a commercial finite volume code are in general good agreement with the results obtained with the spectral element code, it does not give accurate predictions of second-order statistics.

Keywords: natural convection, differentially heated cavity, spectral element method, direct numerical simulation

1. INTRODUCTION

Natural convection flows occur in a large range of natural phenomena and in engineering applications. This phenomenon has been investigated extensively both numerically and experimentally over the past decades, usually adopting canonical flow configurations with simplified geometries and boundary conditions, such as rectangular cavities heated from the side or heated from below. The latter configuration becomes unstable at a relatively low Rayleigh number, since the direction of the heat flux is opposed to that of gravity. Buoyant flows in differentially heated cavities, on the other hand, become unstable at much higher values of the Rayleigh number. For low aspect ratios, the instabilities depend on the boundary conditions imposed on the horizontal walls, but when the aspect ratio is greater than or equal 4, the transition to an unstable state is initiated on the vertical boundary layers, independently of the boundary conditions imposed on the horizontal walls (Xin and Le Quéré, 1995). In differentially heated tall cavities, the flow is governed by the aspect ratio, by the Prandtl number and by the Rayleigh number based on the cavity height. To obtain accurate solutions for turbulent natural convection flows it is necessary to conduct unsteady, three-dimensional numerical simulations of the Navier-Stokes equations using high-order numerical methods. The use of direct numerical simulation (DNS) to study turbulent natural convection flows is important to investigate instability mechanisms and to improve turbulence modelling.

Xin and Le Quéré (1995) carried out unsteady two-dimensional numerical simulations for natural convection in a differentially heated cavity of aspect ratio 4, using a spectral algorithm employing Chebyshev polynomials as expansion bases and a second-order semi-implicit time integration scheme. This flow configuration is known to become unstable for $Ra > 10^8$, so the authors studied the cases $Ra = 6.4 \cdot 10^8$, $Ra = 2.0 \cdot 10^9$ and $Ra = 10^{10}$. The calculations revealed that the vertical boundary layers remain laminar over most of the cavity height, even for the largest value of Ra studied. For the lowest value of Ra studied, the flow is weakly turbulent, with the instabilities concentrated in the downstream part of the boundary layers; the cavity core remains stratified, as would occur in a laminar solution. As the Rayleigh number is increased, the flow in the core of the cavity becomes unsteady and it is no longer stratified for $Ra = 10^{10}$. In a more recent work, Trias *et al.* (2007) performed two- and three-dimensional numerical simulations for this problem, considering the flow to be periodic in the third direction. These authors used second- and fourth-order spectro-consistent spatial discretisation and a second-order time scheme. The results are in general good agreement with those reported by Xin and Le Quéré (1995), but it was observed that in the two-dimensional cases the fluctuation levels are over-predicted in comparison with the three-dimensional simulations. This occurs because the third coordinate allows the energy to

be dissipated to the smallest scales, whilst in 2D the energy is transferred to the large scales. The three-dimensional calculations revealed that even for $Ra = 10^{10}$ the instabilities are concentrated at the lower and upper parts of the cavity, with the core remaining stratified. Trias *et al.* (2010a,b) observed that significant changes in the flow structure occur for $Ra > 10^{10}$ as the transition point moves upstream the boundary layers.

Ghaisas *et al.* (2013) evaluated the performance of various large-eddy simulation (LES) techniques to solve the problem of turbulent natural convection in a differentially heated tall cavity. Excellent agreement was obtained for the velocity and temperature fields in comparison with the DNS results from Trias *et al.* (2007). The second-order statistics, on the other hand, were not very accurately predicted by any of the LES models. LES models are desirable since they have the potential to provide accurate solutions with a much lower computational cost when compared to DNS. DNS solutions are necessary, though, to provide results for a thorough validation of LES models, specially in terms of high-order statistics.

In this work the code Semtex (Blackburn and Sherwin, 2004), which employs a spectral element method, is used to conduct direct numerical simulations of turbulent natural convection in a tall differentially heated enclosure. The flow is solved in a three-dimensional, periodic domain for $Ra = 2.0 \cdot 10^9$. The solutions will be compared against existing DNS data and with results obtained with the commercial, low-order finite volume code Ansys Fluent. The fact that the flow is periodic in one of the directions allows the use of Fourier expansions, which provides a favourable framework for parallelisation.

2. GOVERNING EQUATIONS AND NUMERICAL METHOD

2.1 Governing equations

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

$$Pr = \frac{\nu}{\alpha} = 0.7 \quad (1)$$

$$Ra = \frac{g\beta\Delta TH^3}{\nu\alpha} = 2 \cdot 10^9 \quad (2)$$

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

The governing equations are solved in incompressible form using the Boussinesq approximation to account for changes in density only in the term that is multiplied by the acceleration of gravity. The continuity, Navier-Stokes and energy equations are then given by:

$$\nabla \cdot \mathbf{u} = 0 \quad (3)$$

$$\rho \left(\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} \right) = -\nabla p + \mu \nabla^2 \mathbf{u} - \rho \beta (T - T_0) \mathbf{g} \quad (4)$$

$$\frac{\partial T}{\partial t} + \mathbf{u} \cdot \nabla T = \alpha \nabla^2 T \quad (5)$$

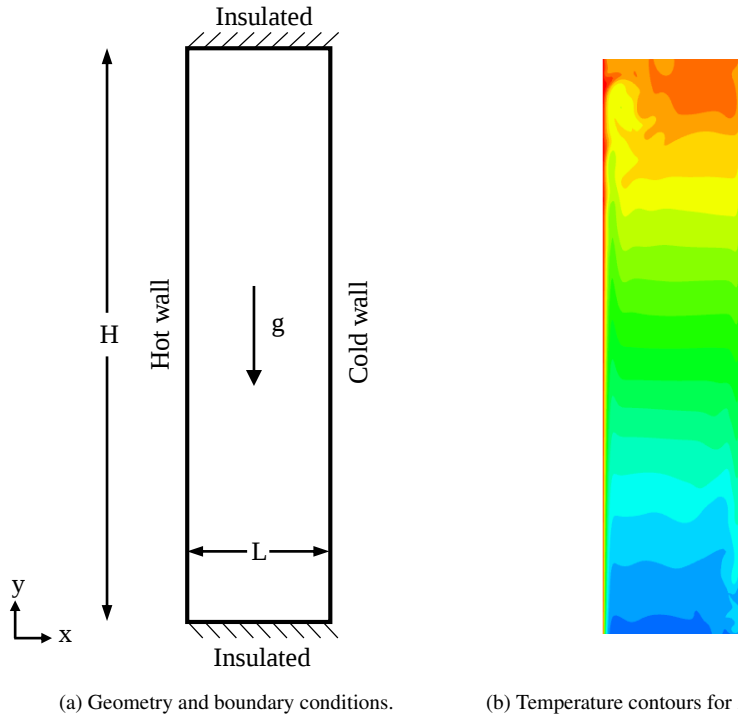
2.2 Numerical Method

The problem is solved using the DNS code Semtex, a Navier-Stokes solver that employs Fourier expansions to solve three-dimensional problems with periodicity in one direction. The two remaining directions are discretised using a spectral element approximation, which combines the flexibility of the Galerkin, finite element method with the high accuracy and efficiency of spectral methods. Using Fourier expansions, a given variable ϕ is projected in a set of two-dimensional Fourier modes, as shown in Eq. 6, where k is the wavenumber in the periodic direction.

$$\hat{\phi}_k(x, y, t) = \frac{1}{2\pi} \int_0^{2\pi} \phi_k(x, y, z, t) e^{-ikz} dz \quad (6)$$

The spectral element approximation employs Lagrange polynomials $h_p(x)$ through the zeros of the Gauss-Lobatto-Legendre (GLL) polynomials. In one dimension the Lagrange interpolant is written in the form,

$$\mathcal{I}\phi(x) = \sum_{p=0}^P \phi(x_p) h_p(x) \quad (7)$$



(a) Geometry and boundary conditions.

(b) Temperature contours for $Ra = 2 \cdot 10^9$.

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

where P is the polynomial order. If we choose $P = 5$, for example, we will have six Lagrange polynomials $h_0(x)$, $h_1(x)$, $\dots$, $h_5(x)$ of order 5 interpolating the function $\phi(x)$. In the spectral element method the approximations are written for a canonical domain $\xi = [-1, 1]$ (in one dimension) and the solution is obtained through a global assembling. Using the GLL points ξ_p the Lagrange interpolant is given by Eq. 8, where L_P is the Legendre polynomial of order P .

$$h_p(\xi) = \begin{cases} 1 & \xi = \xi_p \\ \frac{(1 - \xi^2)L'_P(\xi)}{P(P+1)L_P(\xi_p)(\xi - \xi_p)} & \text{otherwise.} \end{cases} \quad (8)$$

Extension to the two-dimensional case is straightforward as the approximation is written as the tensor product of two one-dimensional expansion bases in the form $h_{pq}(\xi_1, \xi_2) = h_p(\xi_1)h_q(\xi_2)$, where the canonical region is now the square $[-1, 1] \times [-1, 1]$, with $-1 \leq \xi_1, \xi_2 \leq 1$. Using the spectral element method the numerical grid can be refined by either decreasing the size h of the elements or by increasing the polynomial order p of the interpolant within each element (the so-called hp refinement).

The code runs in parallel using MPI (Message Passing Interface). The number of processors is limited by the number of Fourier modes used, since the parallel algorithm partitions the domain in the periodic direction only. The time integration is carried using an explicit second-order accurate scheme. For further details on the numerical method and its implementation, the reader is referred to Blackburn and Sherwin (2004) and Karniadakis and Sherwin (2013).

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

3. RESULTS AND DISCUSSION

The solution was initiated from a pure-conduction temperature profile and zero velocity. In Semtex a random perturbation was imposed to obtain transition to the unstable state. Transition to the final state was monitored by analysing the value of the Nusselt number on the hot wall along the calculation time. After the value of Nu started to oscillate around

a constant value, statistics were collected until a statistically steady state was obtained. Figure 2 shows the distribution of the modal energy $E(k)$ as a function of the Fourier mode k , normalised with the energy of the first mode, not shown in the graph as it is much larger than the energy for the rest of the modes. It is noted that the energy decays more than four orders of magnitude between the second and the last modes, indicating that a large range of scales is resolved. The lowest modes contain most of the energy of the flow, whilst the highest modes contain little energy and are responsible for energy dissipation.

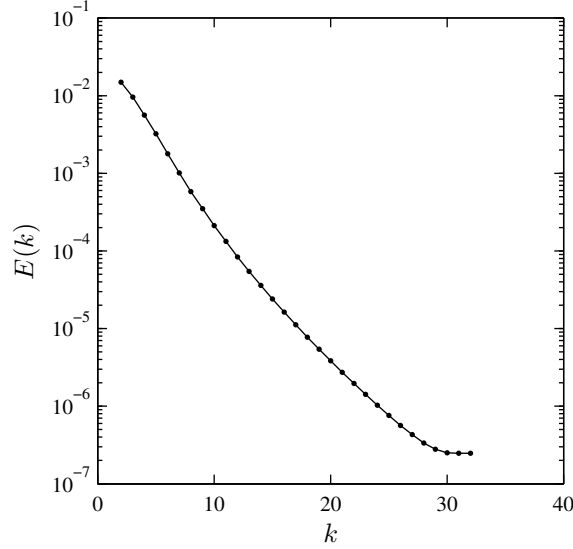


Figure 2: Distribution of the modal energies as a function of the Fourier mode k .

Instantaneous temperature contours obtained with the spectral element code are illustrated in Fig. 1, where the general behaviour of the flow can be understood. Hot fluid is transported towards the upper part of the cavity along the hot vertical wall. The boundary layer is laminar in most of the vertical length, and becomes unstable in the downstream part, near the top of the cavity, where strong mixing occurs. On the right hand side, on the cold wall, a similar behaviour is observed, but with cold fluid moving down in a laminar boundary layer which only becomes unstable in the downstream part, i.e., in the bottom part of the cavity. Note that the core of the cavity remains stratified, in agreement with the results reported by Trias *et al.* (2007). Figure 3 shows iso-surfaces of the Q criterion, where it is noted that small-scale structures are formed downstream the boundary layers and that the regions of turbulent flow are located in the top and bottom of the cavity.



Figure 3: Iso-surfaces of the Q criterion for $Ra = 2 \cdot 10^9$.

Figure 4 shows profiles for the horizontal velocity and for the $\overline{v'v'}$ component of the Reynolds stress tensor at the mid vertical position, $y^* \equiv y/H = 0.5$ obtained with both codes. These results were obtained averaging the flow field

both in time and along the periodic direction, and are normalised with U_s and U_s^2 , respectively. The horizontal velocity profile reveals that there is no motion in the horizontal direction in the cavity core, with the horizontal velocity having a small component in the boundary layers only. Note that $v'v'$ has the same order of magnitude as the horizontal velocity, indicating that at this location the vertical motion dominates the flow structure. The vertical velocity at this location, not shown here, is approximately three orders of magnitude larger than the horizontal velocity. For this location both Semtex and Fluent give very good predictions for the horizontal velocity component in comparison with the DNS results from Trias *et al.* (2007). The turbulent fluctuation $\overline{v'v'}$, on the other hand, is under-predicted by Fluent. As expected, the fluctuation is much higher near the walls than in the cavity core.

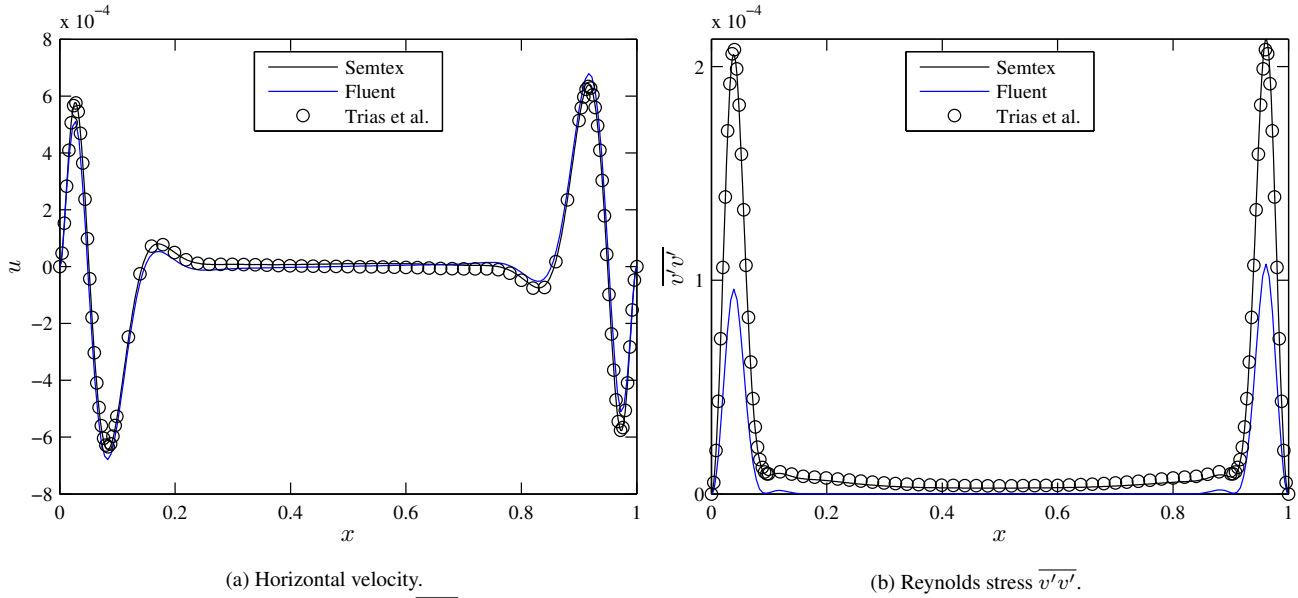


Figure 4: Horizontal velocity and $\overline{v'v'}$ at the cavity mid-height ($y^* = 0.5$). Comparison with Trias *et al.* (2007).

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

The mean Nusselt number on the vertical walls is very well predicted by both codes. Trias *et al.* (2007) reported a value of $Nu = 66.63$, whilst $Nu = 66.59$ and $Nu = 65.64$ were obtained with Semtex and Fluent, respectively, giving relative errors of 0.06% and 1.49%, respectively. In general it is observed that Fluent can give accurate predictions for global features of the flow, such as velocity profiles and heat transfer rates, but it fails to give accurate predictions for second-order statistics. The computing time per time step is almost 50% smaller with the spectral element code than with Fluent. It should be noted that the calculations performed with Semtex took much longer since the time step is much smaller when compared with Fluent, requiring a higher number of time steps.

4. CONCLUSIONS

In this work the performance of two codes was compared by performing numerical simulations of turbulent natural convection in a differentially heated cavity on meshes with similar numbers of degrees of freedom. Both codes provided very good predictions for the Nusselt number and for velocity and temperature profiles at the cavity mid-height. In a region in the upper part of the cavity, however, the results obtained with Fluent for the horizontal velocity profile did not agree well with previous results. The turbulent fluctuation of the vertical velocity was well predicted by the spectral element code in both regions analysed, whilst Fluent gave very poor predictions for the second-order statistics. It is

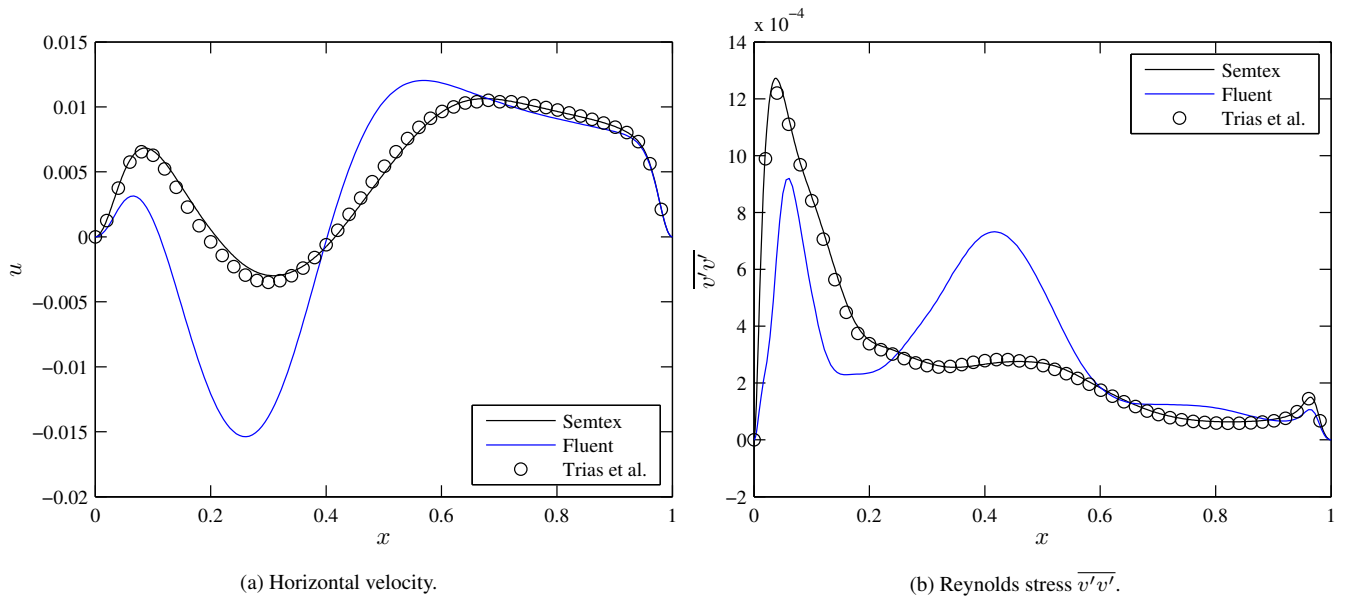


Figure 5: Horizontal velocity and $\overline{v'v'}$ at $y^* = 0.9$. Comparison with Trias *et al.* (2007).

noted that a similar number of degrees of freedom was used in both codes, which shows that the spectral element code requires less resolution for a good level of accuracy when compared with Fluent. Finally, it is concluded that Semtex is an attractive tool to carry fundamental investigations of turbulent flow. The fact that the code employs the spectral element method allows the use of complex geometries and unstructured meshes, as opposed with global spectral methods, which are useful only when simple geometries are used.

5. ACKNOWLEDGEMENTS

The authors acknowledge Professor Hugh Blackburn for supplying the Semtex code and advising on its implementation, as well as Dr Olaf Marxen and Prof Nick Hills for their comments. The financial support provided by the CAPES foundation and Rolls-Royce plc is also acknowledged.

6. REFERENCES

- Blackburn, H.M. and Sherwin, S., 2004. "Formulation of a galerkin spectral element-fourier method for three-dimensional incompressible flows in cylindrical geometries". *Journal of Computational Physics*, Vol. 197, No. 2, pp. 759–778.
- Ghaisas, N.S., Shetty, D.A. and Frankel, S.H., 2013. "Large eddy simulation of thermal driven cavity: Evaluation of sub-grid scale models and flow physics". *International Journal of Heat and Mass Transfer*, Vol. 56, No. 1, pp. 606–624.
- Karniadakis, G. and Sherwin, S., 2013. *Spectral/hp element methods for computational fluid dynamics*. Oxford University Press.
- Trias, F., Gorobets, A., Soria, M. and Oliva, A., 2010a. "Direct numerical simulation of a differentially heated cavity of aspect ratio 4 with rayleigh numbers up to 10^{11} - part i: Numerical methods and time-averaged flow". *International Journal of Heat and Mass Transfer*, Vol. 53, No. 4, pp. 665–673.
- Trias, F., Gorobets, A., Soria, M. and Oliva, A., 2010b. "Direct numerical simulation of a differentially heated cavity of aspect ratio 4 with rayleigh numbers up to 10^{11} - part ii: Heat transfer and flow dynamics". *International Journal of Heat and Mass Transfer*, Vol. 53, No. 4, pp. 674–683.
- Trias, F., Soria, M., Oliva, A. and Pérez-Segarra, C., 2007. "Direct numerical simulations of two-and three-dimensional turbulent natural convection flows in a differentially heated cavity of aspect ratio 4". *Journal of Fluid Mechanics*, Vol. 586, pp. 259–293.
- Xin, S. and Le Quére, P., 1995. "Direct numerical simulations of two-dimensional chaotic natural convection in a differentially heated cavity of aspect ratio 4". *Journal of Fluid Mechanics*, Vol. 304, pp. 87–118.

7. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.