Turbulent natural convection in a differentially heated cavity of aspect ratio 5 filled with non-participating and participating grey media

R. Capdevila\textsuperscript{1}, O. Lehmkühl\textsuperscript{2}, F. X. Trias\textsuperscript{1}, C. D. Pérez-Segarra\textsuperscript{1} and G. Colomer\textsuperscript{2}

\textsuperscript{1}Centre Tecnològic de Transferència de Calor (CTTC), Lab. Termotècnia i Energètica, Universitat Politècnica de Catalunya (UPC), C/Colom, 11, E08222 Terrassa, Barcelona, Spain
\textsuperscript{2}Termofluids, S. L., Magí Colet 8, E08204 Sabadell, Barcelona, Spain

E-mail: \textsuperscript{1} cttc@cttc.upc.edu, \textsuperscript{2} termofluids@termofluids.com

Abstract. In the present work, turbulent natural convection in a tall differentially heated cavity of aspect ratio 5:1, filled with air (Pr = 0.7) under a Rayleigh number based on the height of $4.5 \cdot 10^10$, is studied numerically. Two different situations have been analysed. In the first one, the cavity is filled with a transparent medium. In the second one, the cavity contains a grey participating gas. The turbulent flow is described by means of Large Eddy Simulation (LES) using symmetry-preserving discretizations. Simulations are compared with experimental data available in the literature and with Direct Numerical Simulations (DNS). Surface and gas radiation have been simulated using the Discrete Ordinates Method (DOM). The influence of radiation on fluid flow behaviour has also been analysed.

1. Introduction

Experimental measurements of this case were conducted by Cheeswright \textit{et al.} (1986). Measured temperatures at floor and ceiling differ from adiabatic conditions. The authors attributed this fact to the heat losses through the adiabatic horizontal walls and through the vertical front and back walls.

Several numerical studies, using either low Reynolds number two-equation eddy-viscosity turbulent models (Ince \textit{et al.}, 1986; Henkes & Hoogendoorn, 1986; Capdevila \textit{et al.}, 2006; Albets \textit{et al.}, 2008) or LES (Barhaghi & Davidson, 2007), have been presented in the literature. In general, they had difficulties in reproducing experimental results, even using thermal resistance coefficients to reproduce the heat losses (Ince \textit{et al.}, 1986). It has been traced that these discrepancies could be also due to both surface and gas radiation. If gases contained inside the cavity are participating in radiation, both surface and gas radiation affect the fluid flow. In this case, the governing equations of the fluid motion are coupled to the integro-differential equation of radiative transport (RTE) through a new term, the divergence of radiative heat transfer. This term appears in the energy equation relating the net gain of energy due to radiation inside the fluid. The RTE represents an energy balance associated with radiation, and depends not only on spatial position, but also on direction of propagation of radiation.
2. Mathematical formulation

The turbulent flow is described by means of LES using symmetry-preserving discretizations (Rodriguez et al., InPress, 2011). The spatial filtered Navier-Stokes equations can be written as,

$$M_{c} \frac{\partial \bar{u}_{c}}{\partial t} + \mathbf{C}_{c}( \bar{u}_{c}) \bar{u}_{c} + \nu \mathbf{D}_{c} \bar{u}_{c} + \rho^{-1} \mathbf{G}_{c} - \mathbf{T}_{c} = \mathbf{C}( \bar{u}_{c}) \bar{u}_{c} - \mathbf{C}( \bar{u}_{c}) \bar{u}_{c} \approx -M_{c} \mathbf{T}_{c}$$  \hspace{1cm} (2)

where $M$, $C$, $D$ and $G$ are the divergence, convective, diffusive and gradient operators, respectively, $\Omega$ is a diagonal matrix with the sizes of control volumes, $\bar{u}_{c}$ is the filtered velocity, $f_{c}$ is the body force term (in this paper the Boussinesq approximation for the density has been adopted), $M_{c}$ represents the divergence operator of a tensor, and $T_{c}$ is the SGS stress tensor, which is defined as,

$$T_{c} = -2\nu_{sgs} \bar{S}_{c} + (T_{c} : I) I / 3$$  \hspace{1cm} (3)

where $\bar{S}_{c} = \frac{1}{2}\left[ \mathbf{G}( \bar{u}_{c} ) + \mathbf{G}^{*}( \bar{u}_{c} ) \right]$. To close the formulation, a suitable expression for the subgrid-scale viscosity, $\nu_{sgs}$, must be introduced. In the present work three different SGS models have been tested: the wall-adapting local eddy-viscosity (WALE) SGS model (Nicoud & Ducros, 1999), the variational multiscale method - WALE (VMS) SGS model and the QR model (Verstappen, 2009). The main issues related to these models are given below.

The filtered temperature transport equation is

$$\Omega_{c} \frac{\partial \bar{T}_{c}}{\partial t} + \mathbf{C}( \bar{u}_{c}) \bar{T}_{c} + \frac{\nu}{Pr} \mathbf{D}_{c} \bar{T}_{c} - \nabla \cdot \mathbf{q}_{rad} = \mathbf{C}( \bar{u}_{c}) \bar{T}_{c} - \mathbf{C}( \bar{u}_{c}) \bar{T}_{c} \approx -M_{c} \mathbf{T}_{c}$$  \hspace{1cm} (4)

in the $T_{c}$ term, $\nu_{sgs}$ is substituted by $\nu_{sgs}/Pr_{t}$, where $Pr_{t}$ is the turbulent Prandtl (0.4 in this paper).

The $\nabla \cdot \mathbf{q}_{rad}$ is obtained from the RTE. In the present study RTE is solved for an emitting-absorbing non-scattering grey medium:

$$\frac{d \bar{I}}{ds} = -\kappa \bar{I} + \kappa \bar{I}_{b}$$  \hspace{1cm} (5)

The filtered RTE can be written as

$$\frac{d \bar{T}}{ds} = -\kappa \bar{T} + \kappa \bar{T}_{b} = -\pi \bar{T} + (\kappa \bar{T} - \pi \bar{T}) + \pi \bar{T}_{b} + (\kappa \bar{T}_{b} - \pi \bar{T}_{b}) \approx -\pi \bar{T} + \pi \bar{T}_{b}$$  \hspace{1cm} (6)

The simplest way to close the filtered RTE is used, i.e. the terms in parenthesis are ignored and $\bar{T}_{b} \approx I_{b}(T)$. The filtered radiative source term is approximated using the same criteria as in the filtered RTE,

$$\nabla \cdot \mathbf{q}_{rad} = 4\pi \kappa \bar{T}_{b} - \int_{4\pi} \kappa \bar{T} d\omega \approx 4\pi \kappa \bar{T}_{b} - \int_{4\pi} \pi \bar{T} d\omega$$  \hspace{1cm} (7)

2.1. WALE subgrid scale model (Nicoud & Ducros, 1999)

Subgrid scale model based on the square of the velocity gradient tensor,

$$\nu_{sgs} = (c_{w}l)^{2} \frac{\left( \bar{V}_{c} \cdot \bar{V}_{c} \right) ^{2}}{\left( \bar{S}_{c} : \bar{S}_{c} \right) ^{2} + \left( \bar{V}_{c} \cdot \bar{V}_{c} \right) ^{2}} ;$$ \hspace{1cm} (8)

$$c_{w} = 0.5$$

$$\bar{V}_{c} = \frac{1}{2}[\mathbf{G}( \bar{u}_{c} )^{2} + \mathbf{G}^{*}( \bar{u}_{c} )^{2}] - \frac{1}{3} (\mathbf{G}( \bar{u}_{c} )^{2} I)$$
2.2. QR subgrid-scale model (Verstappen, 2009)

Subgrid scale model based on the invariants of the rate-of-strain tensor

\[ \nu_{sgs} = (c_{qr}l)^2 \frac{\nu}{q} \quad (9) \]

\[ c_{qr} = \frac{1}{\pi} + \frac{1}{24} \]

\[ | \mathcal{S}_c | = (2\mathcal{S}_c \mathcal{S}_c)^{1/2} \]

\[ q = \frac{1}{4} | \mathcal{S}_c |^2 \]

\[ r = -\text{det} \mathcal{S}_c \]

2.3. VMS, WALE model with a variational multiscale framework (Hughes et al., 2000)

In VMS three classes of scales are considered: large, small and unresolved scales. The first two classes are solved with LES, whereas the unresolved scales are modelled. In this model, the subgrid small-scale term \( T'_{c} \), is modelled instead of \( T_{c} \),

\[ T'_c = -2\nu_s S'_{ij} + \frac{1}{3} T'_e \delta_{ij} \quad (10) \]

\[ \nu_{sgs} = (C_{w}^{wms} \Delta)^2 \frac{(V'_{ij} : V'_{ij})^{3/2}}{(S'_{ij} : S'_{ij})^{3/2} + (V'_{ij} : V'_{ij})^{3/2}} \]

\[ S'_{ij} = \frac{1}{2} \left[ G(u'_c) + G^*(u'_c) \right] \]

\[ V'_{ij} = \frac{1}{2} \left[ G(u'_c)^2 + G^*(u'_c)^2 \right] - \frac{1}{3} (G(u'_c)^2)^2 \]

where \( C_{w}^{wms} = 0.325 \) and is the equivalent of the WALE coefficient for the small-small VMS approach.

3. Numerical method

Numerical results are carried out by using the CFD code Termofluids which is an intrinsic 3D parallel CFD object-oriented code applied to unstructured-collocated meshes (Lehmkuhl et al., 2007). Fully conservative second-order schemes for spatial discretization and third order explicit time integration are used. The pressure-velocity linkage is solved by means of an explicit finite volume fractional step procedure.

The generated domain is periodic in depth. Therefore the system of equations is reduced to a set of 2D systems by means of a Fourier diagonalization method. These systems are solved using a Direct Schur complement-based domain Decomposition method in conjunction with a Fast Fourier Transform (Borrell et al., Corrected Proof, 2011).

In the code, radiation can be simulated using either the Discrete Ordinates Method (DOM) (Fiveland, 1984; Capdevila et al., 2010) or the Finite Volume Method (FVM) (Raithby & Chui, 1990; Capdevila et al., 2010). In both methods the space is discretized by means of finite volumes and the difference lies on the angular discretization procedure. In DOM, the directional variation of the radiative intensity is represented by a discrete number of ordinates, and integrals over solid angles are approximated by numerical quadratures which are usually designed to preserve symmetry and satisfy several moments. The FVM ensures the conservation of radiative energy by means of integrating the RTE not only in finite volumes (spatial domain) but also in control angles (angular domain). In the present work, some results using DOM with a \( S_6 \) angular discretization and the step scheme, which is the analogous of the upwind scheme in CFD, are
The periodic boundary condition is performed by means of setting the entering intensity at one side of the periodic boundary, \( I_{\text{in}}(x_i, y_i, z_i) \), the value of the outgoing intensity at the same location of the other side of the periodic boundary \( I_{\text{out}}(x_i, y_i, z_o) \) (see figure 1).

The discretized algebraic radiative transfer equations are solved using a parallel sweep solver (Colomer et al., 2010). Parallel computation is performed in a Gigabit Ethernet networked Beowulf cluster of PCs.

### 3.1. Test case:

The studied case corresponds to a cavity of aspect ratio 5:1 (width \( L_x = 0.5 \) and height \( L_y = 2.5 \)) filled with air (Prandtl number evaluated at mean temperature of the isothermal walls is 0.7) and a Rayleigh number based on the height of \( 4.5 \cdot 10^{10} \). In the first studied situation the medium is considered transparent to the radiation. In the second situation the medium is participating in radiation, with a Planck number (defined as \( \Pi = \frac{\sigma T_o^4}{(\lambda \Delta T_o/H)} \)) of 1253.8 and a temperature ratio \( \delta = \frac{\Delta T_o}{T_o} \) of 0.1399. The four walls are considered black. The optical thickness \( (\tau = \sigma H) \) of the medium is 1.0 and the scattering albedo \( (\varpi = (1 + \kappa/\sigma_s)^{-1}) \) is 0.

Baraghani & Davidson (2007) stated that a spanwise width of 0.2 was large enough to capture the required turbulent scales. The same value, a spanwise depth of \( L_z = 0.2 \), has also been chosen in this work.

### 4. Results

Different meshes have been tested, however only results for the four different structured meshes are presented (see table 1).

The two different meshes used in the DNS solutions (Trias et al., 2010) are also indicated in table 1. Meshes m1, m2, m3 and DNS are uniform in the vertical \( (y) \) and periodic \( (z) \) directions and distributed using a hyperbolic function in the horizontal \( (x) \) direction. Meshes m2 and m3 are obtained refining m1 in vertical and horizontal directions, respectively. Mesh m4 is used for the computation of the case where radiation is taken into account, thus it is also concentrated near the horizontal walls in order to capture the steep gradients and avoid instabilities due to the radiation heat fluxes. Therefore, in the 20% of the height located near the horizontal walls 80 CV’s are concentrated using an hyperbolic function. In the remaining 60% of the height, which is located in the central region, 180 CV’s are distributed uniformly. In meshes m1, m2, m3 and m4 the distribution in the horizontal direction has been designed taking into account that the thickness of the thermal boundary layer predicted by Grötzbach (1983) and Patterson & Imberger (1980) is \( \delta_0 = 5.15 \cdot 10^{-3} \) and \( \delta_0 = 5.43 \cdot 10^{-3} \), respectively. Three control volumes have been placed inside the smallest thermal boundary layer value.

---

**Figure 1.** Periodic boundary condition in the RTE resolution. (a) Mesh (b) Sketch of the boundary condition.
Numerical details of the DNS simulations are out of scope of the present paper and can be found in Trias et al. (2010).

Table 1. Meshes used in the numerical experiments.

|   | Nx  | Ny  | Nz  | N_{tot}  |
|---|-----|-----|-----|----------|
| m1 | 40  | 340 | 16  | 2.17 \cdot 10^5 |
| m2 | 40  | 650 | 16  | 4.16 \cdot 10^5 |
| m3 | 55  | 340 | 16  | 2.99 \cdot 10^5 |
| m4 | 55  | 340 | 8   | 1.50 \cdot 10^5 |
| DNS coarse | 160 | 432 | 128 | 8.85 \cdot 10^6 |
| DNS fine    | 318 | 862 | 128 | 3.51 \cdot 10^7 |

4.1. No Radiation (transparent medium)

Natural turbulent convection inside cavities include regions of laminar, transitional and turbulent flow. From a numerical point of view the prediction of the transitional point is a difficult task.

From an experimental point of view it is also very difficult to establish insulated boundaries. Cheeswright et al. (1986) reported heat losses in their experiment. The destabilizing effect of heat losses (negative temperature gradient at the wall) through the ceiling causes the boundary layer to be turbulent. This implies vertical boundary layer to be turbulent along all the cold wall. As heat losses at the bottom wall are smaller, flow stabilizes along that wall and becomes laminar. Therefore, flow is laminar at the beginning of the hot wall until it becomes turbulent at approximately 20% of the height of the cavity. Traditionally numerical simulations have used perfect adiabatic boundary conditions and the aforementioned transition point around 20% of the height as the reference value. However, since the flow is extremely dependent on the boundary conditions, the experimental value seems not very reliable. This is confirmed by the DNS simulation with perfect adiabatic boundary condition (see figure 2), which predicts the transition at 70% of the height \((y = 1.7)\). In figure 2 (top left), it can be observed that DNS coarse and DNS fine meshes give almost the same Nusselt profile, just with slight differences in the location of the transition point. Therefore, from now on only results of the finest mesh will be shown. DNS results show a laminar flow zone up to \(\approx 65\%\) of the height, a transition flow zone between \(\approx 65\%\) and \(\approx 76\%\) and after that a fully developed turbulent flow zone.

The Nusselt profile obtained with meshes m1, m2 and m3 reproduces exactly the laminar and the turbulent flow zones in the boundary layer and main discrepancies take place around the transition zone. Although mesh m3 is not the finest one, it is the one that best reproduces the DNS results. So it seems more important the refinement in horizontal direction (m3) than in vertical direction (m2). Mesh m4 has been designed in a similar way as mesh m3, but refining it near the horizontal walls, where the radiative heat fluxes are introduced and steep gradients are produced. It can be observed that mesh m4 gives also good Nusselt profiles.

The average Nusselt numbers (shown in table 2) confirm that all meshes reproduce the DNS results with an error lower than 6%. In fact, meshes m2, m3 and m4 differ less than 3% with respect to the DNS. For a fixed mesh, WALE and VMS models are better than QR model. However, the accuracy of the solution is mainly dependent on the quality of the mesh.

Barthaghi & Davidson (2007), who used a fine mesh of 98x322x83 control volumes, suggested that, since the location of the transition between dynamic model results and their coarse DNS were pretty different, higher mesh resolutions were needed for natural convection boundary layer to obtain accurate results. Nevertheless, it can be stated from results of meshes m2, m3 and m4 in figure 2, that finer meshes are not needed. The key aspect is the discretization in the direction of the buoyancy flow. This is probably due to the fact that LES models are deactivated in the
laminar region. Then, it is necessary to use a high resolution mesh in this region in order to correctly solve it. Discretization in horizontal and periodic directions seem not being so decisive.

Table 2. Average Nusselt number.

|       | WALE    | QR       | VMS      |
|-------|---------|----------|----------|
| m1    | 146.9 (5.3%) | 147.6 (4.8%) | 148.4 (4.3%) |
| m2    | 154.6 (0.3%) | 151.1 (2.6%) | 157.1 (1.3%) |
| m3    | 154.2 (0.6%) | 154.2 (0.6%) | 154.6 (0.3%) |
| m4    | 150.9 (2.7%) | 152.8 (1.5%) | 151.1 (2.5%) |
| DNS fine | 155.06   |          |          |

In figure 3 average temperature and horizontal velocity at mid-width ($x/L_y = 0.1$) are given. Thermal stratification in the core of the cavity is a key aspect difficult to predict with LES
simulations. Dimensionless stratification is about 1, similar to other numerical simulations of natural convection inside cavities of different aspect ratios (Salat et al., 2004). It can be observed in the left side of figure 3 that, although mesh m1 predicted pretty well the Nusselt profile, it has difficulties in the prediction of the thermal stratification profile. Mesh m3 almost reproduces the DNS solution. Similar conclusions can be derived from the other models.

Horizontal velocity profile at mid-width is difficult to predict due to the fact that velocities are very small. The right side of figure 3 shows the velocity profiles for the different models and mesh m3. All of them reproduce the velocities near the adiabatic walls but fail in the core of the cavity. Again, better results are obtained with WALE and VMS models.

At the mid-height of the cavity ($y/L_y = 0.5$), the predicted vertical velocity and temperature profiles on the different meshes are depicted in figure 4. The differences between DNS and
LES solutions obtained with the four meshes are negligible. It can be also observed that the boundary layer for this solutions is very thin, due to the laminar behaviour of the flow at this height. Conversely the experimental data results show a thick boundary layer according to their turbulent behaviour in this region.

Figure 5 shows the vertical velocity and temperature profiles at 95% of the height of the cavity. The differences of the temperature between DNS and LES solutions obtained with the different meshes are very small. At this height the thickness of the boundary layer clearly indicated that the flow is turbulent. It can be also observed that experimental data differ by far from adiabatic boundary conditions.

4.2. Radiation (participating grey medium)
Numerical simulations of coupled convection and radiation (both surface and gas) have been obtained starting with the initial map of the stationary results without radiation.

Numerical results presented in this section have been obtained with mesh m4 and with VMS model, since in the previous section it has been been stated that it is the most suitable model for the simulation of the fluid flow of the present case.

Figure 6 depicts the temperature isotherms map for the cases with and without radiation. It can be observed that in the radiative situation, isotherms become closer the top of the hot wall and nearer the bottom of the cold wall. This phenomenon is coupled with a downward shift of the isotherm for the average temperature $((T - T_c)/\Delta T = 0.5)$ and a decrease in vertical gradient of the temperature. The latter effect can be also clearly observed in figure 7. Although numerical results with radiation are closer to the experimental data than the results without radiation, the differences suggest that experimental temperature measurements in the top and bottom walls should be used instead of assuming adiabaticity hypothesis in order to reproduce experimental profiles.

An outstanding change is the spread of regions with higher turbulent viscosities near the floor and the ceiling. This phenomenon is accompanied with higher horizontal velocities (see figure 8) implying an increase of the intensity flow. Maximum velocity near the top and bottom walls are not equal in the radiation case. In general, it can be stated that radiation breaks the symmetry of the flow. Eventually, in figure 8 it can be observed that radiation thickens the boundary layers formed along the isothermal surfaces.
5. Conclusions

Turbulent natural convection in a tall differentially heated cavity of aspect ratio 5:1 filled with air, with a Rayleigh number based on the height of $4.5 \cdot 10^{10}$, has been studied numerically with LES models. It has been found that the mesh quality highly affects the quality of the LES solution in the turbulent boundary layer due to the fact that the mesh is used as a passive filter in the physical space LES implementations. Comparison between DNS and experimental data confirm that experiment differ by far from adiabatic boundary conditions. It has also been analysed the effect of radiation in participating medium on the fluid flow. Radiation breaks the symmetry of the case, increases the intensity of the flow and reduces the dimensionless stratification.
6. Acknowledgements

This work has been financially supported by the Ministerio de Educación y Ciencia, Spain (Project: “Development of high performance parallel codes for the optimal design of thermal equipments, reference ENE2010-17801”) and Termo Fluids S.L.

References

ALBETS, X., OLIVA, A. & PÉREZ-SEGARRA, C.D. 2008. Journal of Heat Transfer - Transactions of ASME 130, (7) pp. 1–11.

BARHAGHI, D.G. & DAVIDSON, L. 2007. Physics of Fluids 19 (12), 125106.

BORRELL, R., LEHMKUHL, O., TRIAS, F.X. & OLIVA, A. Corrected Proof, 2011 Journal of Computational Physics .

CAPDEVILA, R., PÉREZ-SEGARRA, C.D., COLOMER, G. & OLIVA, A. 2006. In Proceedings of the V International Symposium on Turbulence, Heat and Mass Transfer. pp. 535-538

CAPDEVILA, R., PÉREZ-SEGARRA, C.D. & OLIVA, A. 2010. Journal of Quantitative Spectroscopy and Radiative Transfer 111, (2) pp. 264–273.

CHEESWRIGHT, R., KING, K.J. & ZIAI, S. 1986. In Proceedings of Significant Questions in Buoyancy Affected Enclosure or Cavity Flows. pp. 75–81.

COLOMER, G., BORRELL, R., LEHMKUHL, O. & OLIVA, A. 2010. In Proceedings of International Heat Transfer Conference 14, IHTC14. Washington, DC, USA.

FIVELAND, W.A. 1984. Journal of Heat Transfer - Transactions of ASME 106, 699–706.

GRÖTZBACH, G. 1983. Journal of computational fluids 49, 241–264.

HENKES, R.A.W.M & HOOGENDOORN, C.J. 1986. In Proceedings of Eurotherm Seminar 22 - Turbulent natural convection in enclosures. A computational and experimental benchmark study (ed. R.A.W.M Henkes & C.J. Hoogendoorn), pp. 64–75.

HUGHES, T.J.R., MAZZEI, L. & HANZEN, K.E. 2000. Computing and Visualization in Science 3, (1) pp. 47–59.

INCE, N.Z., BETTS, P.L. & LAUNDER, B.E. 1986. In Proceedings of Eurotherm Seminar 22 - Turbulent natural convection in enclosures. A computational and experimental benchmark study (ed. R.A.W.M Henkes & C.J. Hoogendoorn), pp. 64–75.

LEHMKUHL, O., PÉREZ-SEGARRA, C.D., BORRELL, R., SORIA, M. & OLIVA, A. 2007. In Proceedings of the Parallel CFD 2007 Conference. pp. 1–8.

NICOU, F. & DUCROS, F. 1999. Flow, Turbulence and Combustion 62 (3), 183 – 200.

PATTERSON, J. & IMBERGER, J. 1980. Journal of Fluid Mechanics 100, 65–86.

RAITHBY, G.D. & CHUI, E.H. 1990. Journal of Heat Transfer 112, 415–423.

RODRIGUEZ, I., BORRELL, R., LEHMKUHL, O., PÉREZ-SEGARRA, C.D. & OLIVA, A. InPress, 2011 Journal of Fluid Mechanics .

SALAT, J., XIN, S., Joubert, P., SERGENT, A., PENOT, F. & LE QUERÉ, P. 2004. International Journal of Heat and Fluid Flow .

TRIAS, F. X., GOROBETS, A., SORIA, M. & OLIVA, A. 2010. International Journal of Heat and Mass Transfer pp. 665–673.

VERSTAPPEN, R. 2009. In Quality and Reliability of Large-Eddy Simulations II. Pisa, Italy.