# Numerical Simulation of Turbulent Natural Convection and Gas Radiation in Differentially Heated Cavities Using FVM, DOM and LES

**R.Capdevila***, **C.D. Pérez-Segarra**†, **O.Lehmkuhl**‡, **G.Colomer**‡

* † Centre Tecnològic de Transferència de Calor (CTTC), Universitat Politècnica de Catalunya (UPC), ETSEIAT, Colom 11, 08222 Terrassa, Barcelona, Spain
e-mail: cttc@cttc.upc.edu

‡Termofluids, S. L.
Magi Colet 8, 08204 Sabadell, Barcelona, Spain

**ABSTRACT**

Natural convection in differential heated cavities has been widely studied experimentally and numerically, however experimental results usually differ from numerical solutions. Since in most of those experimental enclosures absolute temperatures of the walls are relatively high, it has been traced that these discrepancies could be due to both surface or surface and gas radiation. Nevertheless, few studies have been done taking into account only surface radiation (non-participating medium) and much less taking into account surface and gas radiation (participating medium). When gases contained inside the cavity are participating in radiation the governing equations of motion of the fluid are coupled to the integro-differential equation of radiative transport (RTE) through a new term, the divergence of radiative heat transfer, affecting the fluid flow.

The aim of the present work is to analyse the effect of surface radiation as well as both surface and gas radiation on the turbulent natural convection in two different aspect ratio differentially heated cavities. The first case corresponds to a tall cavity of 28.6:1 aspect ratio and $1.92 \cdot 10^{10}$ Rayleigh number. The second case is a cavity of aspect ratio 5:1 with a Rayleigh number of $4.48 \cdot 10^{10}$. Both simulations are compared with experimental data available in the literature and the previous numerical results obtained by the authors in [1] using $k - \epsilon$ and $k - \omega$ low-Reynolds number two-equation eddy-viscosity turbulent models. Radiation is simulated using the Finite Volume Method (FVM) or the Discrete Ordinates Method (DOM) with low and high-order numerical schemes[2]. The turbulent flow is described by means of Large Eddy Simulation (LES) and Regularization Modelling (RGM) using simmetry-preserving discretizations [3].

Numerical results are carried out by using the CFD code TERMOFLUIDS which is an intrinsic 3D parallel CFD code applied to unstructured meshes[4]. In all cases a explicit finite volume fractional-step based algorithm is used. The pressure equation is solved by means of a parallel Schur decomposition solver. Discretized algebraic radiative transfer equations are solved using the parallel iterative solver GMRES.

## References

[1] R. Capdevila, C. D. Pérez-Segarra, G. Colomer, and A. Oliva. Numerical Simulation of Turbulent Natural Convection and Gas Radiation in Differentially Heated Tall Cavities Using Discrete Ordinates Method. In *Proc. V Int. Symp. Turb., Heat and Mass Transf.*, pages 535–538, 2006.

[2] R. Capdevila, C. D. Pérez-Segarra, and A. Oliva. Development and comparison of different spatial numerical schemes for the radiative transfer equation resolution using three-dimensional unstructured meshes. *Journal of Quantitative Spectroscopy and Radiative Transfer*, 111(2):264–273, 2010.

[3] C. D. Pérez-Segarra, O. Lehmkuhl, J. Jaramillo, G. Colomer, and A. Oliva. Turbulence modelling and numerical issues: from RANS to DNS and LES. In *Actas III Conf. Nac. Mecânica Fluidos, Termodinâmica e Energia (MEFTE - BRAGANÇA 09), resumos.*, 15-16, 2009.

[4] O. Lehmkuhl, C. D. Pérez-Segarra, R. Borrell, M. Soria, and A. Oliva. TERMOFLUIDS: A new Parallel unstructured CFD code for the simulation of turbulent industrial problems on low cost PC Cluster. In *Proc. Parallel CFD 2007 Conf.*, 1-8, 2007.