8th. World Congress on Computational Mechanics (WCCM8)
5th European Congress on Computational Methods in Applied Sciences and Engineeering
(ECCOMAS 2008)
June 30 –July 5, 2008
Venice, Italy

ON THE TURBULENT NATURAL CONVECTION IN THERMAL STORAGE TANKS SUBMITTED TO HEAT LOSSES TO THE ENVIRONMENT

*** Ivette Rodríguez [1], Assensi Oliva [1], C. D. Pérez-Segarra [1], Oriol Lehmkuhl [1,2]**

[1] Centre Tecnològic de Transferència de Calor (CTTC)
Universitat Politècnica de Catalunya (UPC)
C. Colom 11, 08222 Terrassa (Barcelona, Spain)
cttc@cttc.upc.edu, http://www.cttc.upc.edu

[2] Termofluids, S. L.
Magi Colet 8,
08204 Sabadell (Catalonia, Spain)

Key Words: *Turbulent natural convection, storages tanks, LES models, coarse DNS, unstructured meshes*

ABSTRACT

The thermal performance of an energy storage system is significantly affected by the internal natural convective flow due to the heat transfer to the environment during its static mode of operation. Considering the importance of the natural convection for many technological applications including cooling of electronics, HVAC, solar energy (solar collectors and storage tanks), among others, the modeling of the thermal and fluid-dynamic phenomena involving natural convective flows have been widely studied in the literature. However, most of the studies carried out have dealt with laminar flows on rectangular enclosures, turbulent natural convection on differentially heated rectangular cavities such as the works conducted by [1, 2], and only a few have considered turbulent-flow modelling on cylindrical enclosures [3].

Regarding to the mathematical modelling of the natural convection phenomena in storage tanks, turbulence modelling must be considered. During its static mode of operation, when there are large temperature differences between the fluid and the environment, turbulent cooling process might occur. However, there is a lack of information about the transition point from laminar to turbulent regimes (e.g. the critical Rayleigh number) to be used as a reference in this kind of transient situations. On the other hand, the study of such complex phenomena can be aborded by means of detailed numerical simulations of heat transfer and fluid flow using Computational Fluid Dynamics (CFD) codes as a powerful tool for solving this kind of phenomena. This is especially true considering the emergence of low cost and loosely coupled parallel computers and the improvement in parallel algorithms that tolerate slow networks.

In this work, the transient cooling process of a fluid initially at rest inside a cylindrical storage tank is investigated by means of CFD simulations. In a previous study [4], it was shown that

under these conditions, the governing parameters of the phenomena inside the tank are the aspect ratio (H/D), the Rayleigh number (Ra), the non-dimensional wall conductance $(\widehat{U})$ and the Prandtl number (Pr).

Considering the state-of-the-art in turbulence modelling, a comparison between different LES models are taken into account in order to choose an appropriate modelisation of the phenomena involved: the dynamic Smagorinsky model, the regularization model and a coarse DNS. The selected models should be capable of catch the development of the turbulent structures at the beginning of the cooling process as well as to describe the relaminarisation in the core of the tank as the time marches.

The numerical study is carried out by using TermoFluids CFD code [5]. This code is an unstructured and parallel object-oriented CFD code for accurate and reliable solving of industrial flows. Governing equations are solved by means of finite-volume techniques on unstructured grids. Coupling between pressures and velocities are solved by means of a fractional-step based algorithm, while for temporal discretisation explicit Adams-Bashforth, semi-implicit Crank-Nicolson or a fully-implicit schemes are used depending on the modelisation considered. The pressure equation is solved by means of a parallel Schur decomposition solver which is an efficient direct solver for loosely coupled PC clusters.

The physical phenomena taking place during the transient cooling process of the fluid inside the tank will be presented, as well as the comparison between the different turbulent models considered. Attention is focused on tank aspect ratios and Rayleigh numbers. The computations carried out will shown the capability of the different models to describe the turbulent nature and relaminarisation of the fluid inside the enclosure, also giving criteria about the computational effort/precision required to solve this kind of problems.

REFERENCES

[1] X. Trias, M. Soria, D. Pérez-Segarra, and A. Oliva. Direct Numerical simulations of two and three dimensional turbulent natural convection flows in a differentialy heated cavity of aspect ratio 4. *Journal of Fluid Mechanics*, 586:259–293, 2007.

[2] S. Peng and L. Davidson. Large Eddy simulation for turbulent buoyant flow in a confined cavity. *International Journal of Heat and Fluid Flow*, 22:323–331, 2001.

[3] E. Papanicolaou and V. Belessiotis. Transient natural convection in a cylindrical enclosure at high Rayleigh numbers. *International Journal of Heat and Mass Transfer*, 45:1425–1444, 2002.

[4] I. Rodríguez, A. Oliva, and C.D. Pérez-Segarra. Numerical study of the transient convection inside solar domestic hot water storage tanks. In *Proceedings of the 6th ISES Europe Solar Congress (EUROSUN 2006)*, Glasgow, 2006.

[5] O. Lehmkuhl, C. D. Perez-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 *Proceedings of the 2005 Parallel Computational Fluid Dynamics Conference*, pages 1–8, 2007.