Tenth International Conference on Computational Fluid Dynamics (ICCFD10), Barcelona, Spain, July 9-13, 2018

ICCFD10-2018-xxxx

Towards Real-time CFD Simulation of Indoor Environment

N. Morozova, R. Capdevila, F.X. Trias and A. Oliva
Corresponding author: nina@cttc.upc.edu

Heat and Mass Transfer Technological Center (CTTC), Universitat Politècnica de Catalunya - BarcelonaTech (UPC), ESEIAAT, C/ Colom 11, 08222 Terrassa (Barcelona), Spain

Keywords: Indoor Environment Simulation, Large Eddy Simulation, Reynolds Averaged Navier-Stokes

Heating, ventilation, and air conditioning (HVAC) systems create thermally comfortable environment with acceptable indoor air quality. In order to regulate indoor air parameters (temperature, humidity, etc.), it is essential to have suitable tools to predict HVAC performance in buildings. It can be simulated using airflow network models, zonal models and Computational Fluid Dynamics (CFD).

Airflow network models represent the building as a network of well-mixed zones with uniform temperature, pressure and velocity, connected by the airflow paths. These models are based on Bernoulli equation, so momentum effect is neglected. Because of their simplicity, these models have severe limitations, which can produce significant errors in the results.

Zonal models are considered the intermediate between airflow networks and CFD. They divide a room into a limited number of cells and solve mass end energy balance equations in each cell. Zonal models do not solve momentum equation in order to reduce computing costs, so in case the flow momentum is strong, their accuracy would drop considerably. In order to improve accuracy, regions with strong flow momentum are treated specially, which significantly increases the complexity of the method. In many cases, the time for preparing data input for a zonal model may be longer than a coarse-grid CFD simulation [1].

CFD numerically solves a set of partial differential equations for the conservation of mass, momentum, energy and turbulent quantities. Building or room is divided into control volumes and the equations are solved for every mesh element. The solution provides a complete set of air parameters at every point of the building. Even though CFD has become more and more popular in indoor environment simulations, the huge computational cost makes many engineers and researchers seek for faster and yet accurate CFD models.

Main objectives of this study are to choose a reliable and robust model to perform CFD simulations of indoor environment with minimal computational cost and adequate accuracy. As well as to investigate capabilities of CFD to perform real-time simulations of indoor environment. Real-time simulations taking into account instantaneous changes in the external weather conditions and occupants behavior would make it possible to adjust parameters of HVAC system in order to maintain the lowest energy consumption, while keeping acceptable indoor air quality.

Flow inside the buildings is driven by the natural and forced convection. We have chosen two characteristic configurations that mimic typical airflow patterns inside the building. First test case is a turbulent ($Ra = 1.2 \times 10^{11}$) air-filled differentially heated cavity of height aspect ratio 3.84 which resembles a highly-stratified indoor environment with natural convection, such as an atrium or a hall. The cavity is subject to a temperature difference across the vertical isothermal walls. The geometry of the problem is displayed in Figure 1 (left). Second test case, turbulent ($Ra = 2.4 \times 10^9$) mixed convection in a ventilated square cavity represents a ventilated room with thermal exhausts coming from its lower part. Air is supplied from a thin inlet slot at the ceiling level and the outlet slot is located at the floor level on the opposite wall. The floor is heated, while the other walls are maintained at the temperature of the cold inlet jet (Figure 2 - left).

The incompressible Navier-Stokes equations coupled with the temperature transport equation are considered. Equations are discretized on a collocated Cartesian grid in a finite-volume framework. A Fractional step method is used to solve the velocity-pressure coupling along with a second-order symmetry-preserving

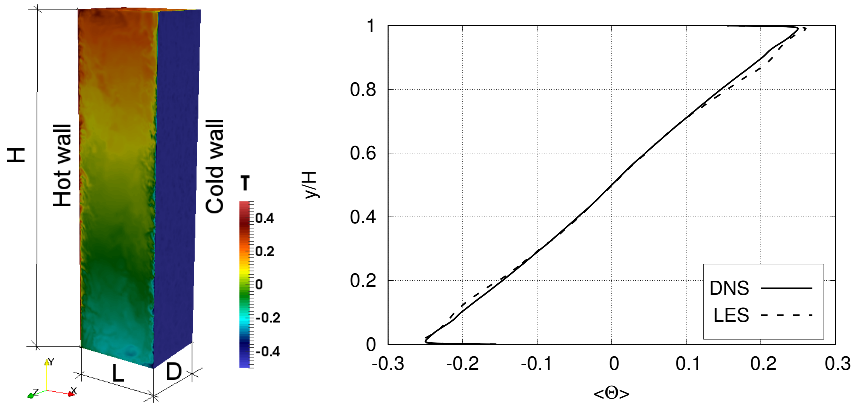

Figure 1: Natural convection case, Left: schematic view of the test case and instantaneous isotherms, Right: averaged temperature profile at the cavity mid-width.

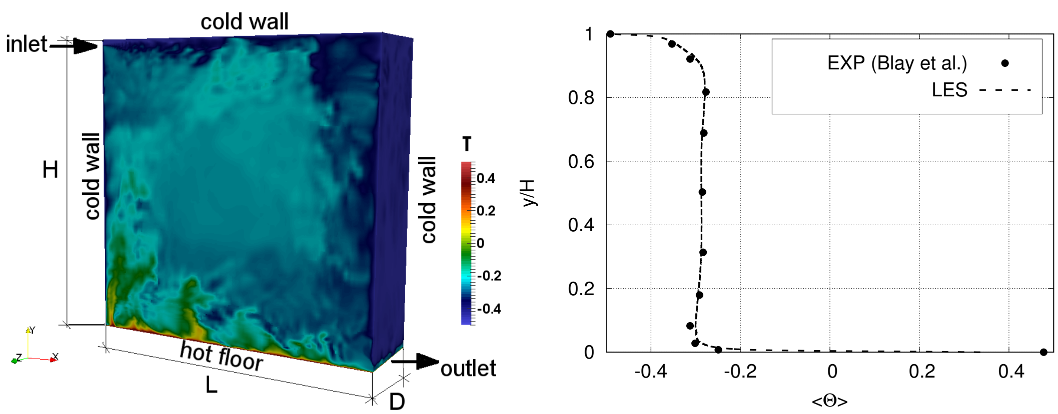

Figure 2: Forced convection case, Left: schematic view of the test case and instantaneous isotherms, Right: averaged temperature profile at the cavity mid-width.

numerical scheme. Direct Numerical Simulation (DNS), Large Eddy Simulation (LES) and Reynolds Averaged Navier-Stokes (RANS) techniques are used to model the turbulent flow.
Both cases are being tested on a wide range of computational grids along with different turbulent models such as LES-WALE, LES-QR and k-ε (RNG). Some preliminary results obtained for LES-WALE model are compared to the DNS simulation in case of natural convection (Figure 1), and to the experimental data obtained by Blay *et al.* [2] (Figure 2) in case of forced convection.
Global quantities of interest like average Nusselt number, average global kinetic energy and total mass conservation are calculated for different numerical set-ups (numerical methods and grid sizes). A comparison between accuracy of the results, reliability of the models and computational cost will be presented in order to find its optimal combination.

References

[1] Q. Chen. Ventilation performance prediction for buildings: A method overview and recent applications. *Building and Environment*, 44(4):848–858, 2009.

[2] D. Blay, S. Mergui, and J.L. Tuhault amd F. Penot. Experimental turbulent mixed convection created by confined buoyant wall jets. *First European Heat Transfer Conference, UK*, pages 821–828, 1992.