On the Feasibility of CFD for Transient Airflow Simulations in Buildings

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 Airflow Simulation, Large Eddy Simulation, Transient Building Simulation, Unsteady Reynolds Averaged Navier-Stokes

1 Aim and Approach

The aim of this work is to investigate the capabilities of steady CFD simulations to be applied for the design of HVAC systems and transient CFD simulations for the Model Predictive Control (MPC) for HVAC systems. For this purpose, the possibility of performing real-time and faster-than-real-time reliable CFD simulations using a reasonable amount of computational resources is studied. Different approaches including Large Eddy Simulation (LES) and Unsteady Reynolds Averaged Navier-Stokes (URANS), mesh resolutions and discretization techniques (collocated and staggered) are tested in order to choose reliable and robust approaches to perform CFD simulations of indoor environment with minimal computational cost and adequate accuracy. Two characteristic configurations which mimic typical airflow patterns inside buildings are considered in this study. These are cavities driven by natural or forced convection forces. The quality of the simulations is evaluated as the transient time evolution of the global airflow quantities, which describe the overall heat transfer and flow characteristics. The computational speed is evaluated as the computational time ratio, R, the ratio between the wall-clock time and the physical time of the simulations. The estimation of the feasibility of using CFD simulations for HVAC applications within the next years is done using Moore's law [1].

2 Scientific Innovation and Relevance

The fast and accurate computation of indoor airflow is an important point for the design of effective HVAC setups. Indoor airflow could be calculated using multizone, zonal or CFD models. CFD provide the most detailed information in cost of high computational effort, which implies that CFD is not usually used in HVAC design or MPC systems, since they require real-time and faster-than-real-time CFD simulations, performed using limited computational resources, all these makes CFD unavailable for HVAC applications nowadays. Moreover, MPC systems require transient airflow simulations in order to adapt the HVAC performance to the changes in the system, such as occupants behavior, electrical load changes or opening of doors and windows. However the growing computational capacity makes CFD an attractive approach. Some works have studied the feasibility of CFD for indoor environmental simulations, but the question of computational cost of the near future applications and the possibilities of transient simulations have not yet been investigated. The growing computational capacity altogether with the efficient turbulence models and discretization techniques are helping to reduce the computational cost of CFD simulations and make it feasible to maintain the quality of life and minimize the energy use of buildings.

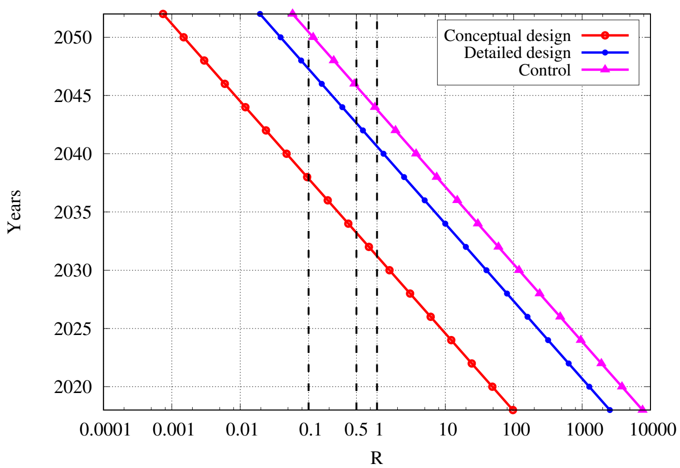

Figure 1: Potential of accessing real-time (R = 1) and faster-than-real-time (R < 1) CFD simulations over the next years for an arbitrary $1000m^3$ building [4].

3 Preliminary Results and Conclusions

Two characteristic configurations which mimic typical airflow patterns inside buildings are considered: A tall turbulent ($Ra = 1.2 \times 10^{11}$) air-filled deferentially heated cavity, which resembles a highly-stratified indoor environment with natural convection, such as an atrium or a hall, and turbulent ($Ra = 2.4 \times 10^{9}$) mixed convection in a ventilated square cavity, represents a ventilated room. Air is supplied from an inlet slot at the ceiling and the outlet 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. Both cases resemble experiments performed by Saury *et al.* [2] and Blay *et al.* [3], respectively. In the previous work [4] the steady results have been obtained for the first test case. They show that CFD simulations are not yet affordable neither for design nor for control of indoor environment. But according to the Moore's law [1] taking into account growing computational capacity CFD would be feasible for design purposes on office workstations within the next decade for the investigated cavity and within two decades for an arbitrary residential building. While HVAC predicting control systems equipped with CFD simulation tools will get affordable in approximately two-three decades (Figure 1).

References

[1] G. E. Moore. The future of integrated electronics. Technical report, Palo Alto, CA, USA, 1965.

[2] D. Saury, N. Rouger, F. Djanna, and F.Penot. Natural convection in an air-filled cavity: Experimental results at large Rayleigh numbers. *International Communications in Heat and Mass Transfer*, 38(6):679–687, 2011.

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

[4] N. Morozova, R. Capdevila, F.X. Trias, and A. Oliva. Towards Real-Time CFD Simulation of Indoor Environment. In *Proceedings of 10th International Conference on Computational Fluid Dynamics*, July 9-13, 2018. http://www.iccfd.org/iccfd10/papers/ICCFD10-344-Paper.pdf.