CFD Modeling Approach for HVAC Systems Analysis

R. Mahu, F. Popescu and I.V. Ion

„Dunarea de Jos" University of Galati, 47 Domneasca St. 800008, Galati, Romania,
email: razvanmahu@yahoo.com, florin.popescu@ugal.ro, iion@ugal.ro

Abstract: A heating, ventilation and air conditioning (HVAC) system consisting of a simplified building with one room was modelled and simulated considering both the internal and the external conditions. The numerical model was designed to concurrently take into account: (1) the internal buoyancy-driven convective heat transport, (2) the conductive heat flow through the solid walls, (3) the external convective cooling induced by the lower temperature wind, and (4) the internal and external radiative heat transfer. In order to avoid making approximations by specifying an average value for the external heat transfer coefficient (HTC), the external boundary of the numerical model was modified to include a significant external volume around the building. Thus, the flow around the building was directly simulated and the external and internal HTCs were computed on-the-fly. The coupled fluid flow – thermal transfer simulation was performed to evaluate the internal environment comfort level.

Keywords: HVAC, numerical modelling & simulation, CFD, RANS.

1. Introduction

The HVAC systems design for modern buildings in civil engineering, simultaneously satisfying the requirements of comfort and optimal working conditions, and also optimizing the energy consumption, has become increasingly difficult. The ever increasing height and complicated architectural shapes, along with the more and more stringent specifications for energetic efficiency and also the continuously shorter concept-to-real building timeframe demands, have led the engineers towards the virtual environment simulation methods.

Up to the 1990s, most researches in fluid mechanics were achieved by expensive experimental studies. With the increasing processing power of computers, a third approach in fluid mechanics research has become increasingly available: the computational fluid dynamics (CFD), that synergistically completes the theoretical approach and experiment studies, providing the architects and engineers with a powerful tool for building design (Figure 1).
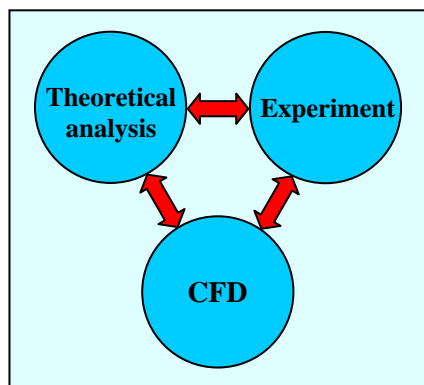

Figure 1. Synergy between theory, experiment and numerical modelling

There are several strong motivations for the use of CFD: short turnaround times, lower analysis costs compared with experimental studies in laboratories, wind tunnels or water channels, comprehensive data availability, including time-accurate, on all physical quantities for a given model in any region of the field.

2. Problem description

The case modelled is a complex problem of air flow and heat transfer inside and outside an enclosure taking into account outdoor climatic conditions. Basically, this is a one room building (Figure 2), with dimensions of 6 m (length) x 4 m (width) x 3 m (height), with brick walls of 0.35m thick, a glass window and a wooden door. The room is illuminated by six light sources each having a thermal output of 34 W. Inside the building there is an 80 kg person working at a computer with a thermal power of 173 W. The human heat input as equivalent to a person wearing a cotton clothes line. Thermal regulation is performed by a free convection heat source, with a power of 3000 W. External wind speed is 5 m/s, and the upstream airflow temperature is -15 °C. Based on this initial data a complex numerical model is needed, capable of simultaneously simulating practically all types of heat transfer:

- The internal heat transfer by natural convection and radiation determined by internal heat sources;
- The outside wind forced convection and the external radiative heat transfer;
- The conductive heat flow through the building walls.

3. Modeling and simulation approach

In order to increase the level of solution accuracy, a coupled fluid flow – thermal transfer solution was preferred. This implies that the numerical model must be able to simulate not only the internal and external heat transfer, but also the internal and external flow, at one time.
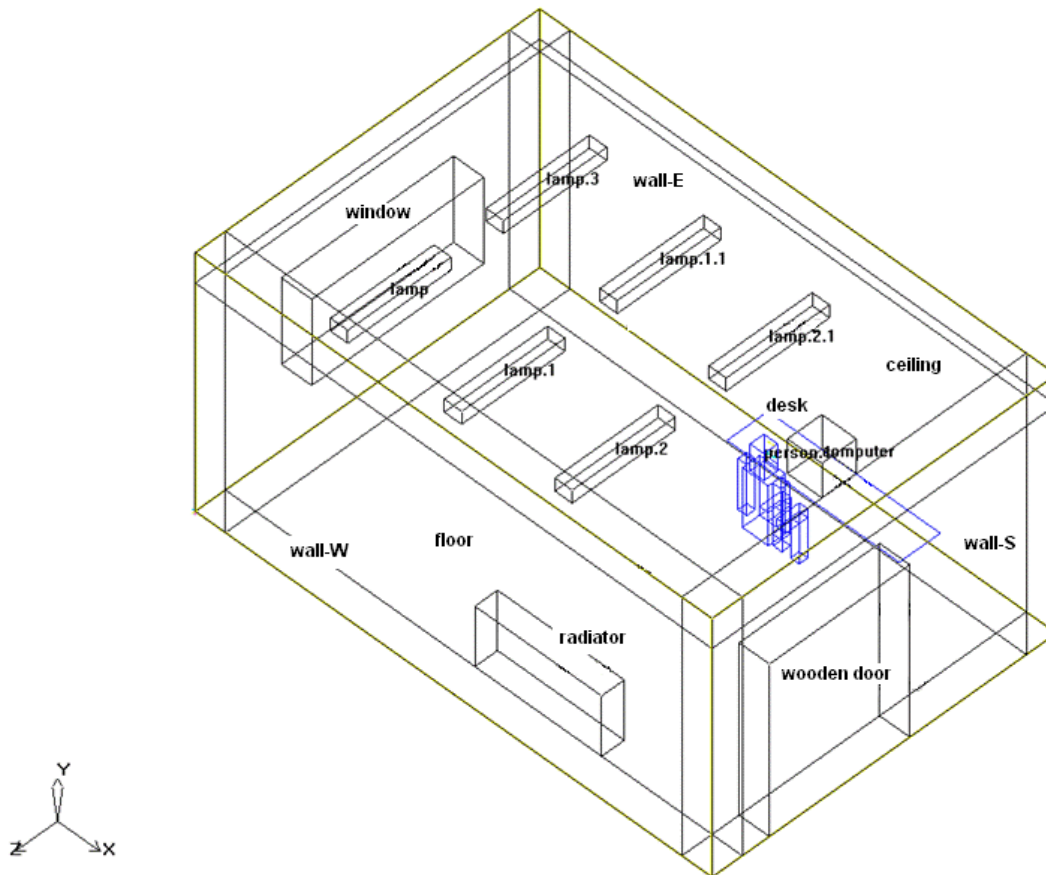
Figure 2. Building schematic with internal configuration

A combined approach of using ANSYS Airpak 3.0 for numerical model preparation and final postprocessing, and ANSYS Fluent 12.0 for solution calculation and initial postprocessing, proved to be optimal. The predefined geometry construction and meshing features of Airpak software greatly facilitated the preprocessing step, while the parallel processing capabilities of Fluent software offered a very short solution turnaround time.

A fully structured, multiblock discretization was created using the semi-automated tools in Airpak. The unsteady-state Reynolds-Averaged Navier-Stokes (RANS) model was selected, with RNG k-epsilon turbulence modeling. No solar heat input was considered. All material data was taken from the available standard Airpak material libraries.

The RNG k-ε turbulence model is similar to the standard k-ε model, but uses an improved derivation based on a statistical technique, called "renormalization group theory". Several terms in the k and ε transport equations have been modified, while the model constants, empirical for the standard k-ε model, have been analytically derived.

The transport equations for k and ε are presented below:

$$\rho \frac{Dk}{Dt} = \frac{\partial}{\partial x_i}\left(\mathrm{Pr}_k\, \lambda_{ef}\, \frac{\partial k}{\partial x_i}\right) + G_k + G_b - \rho\varepsilon - Y_M \quad (1)$$

$$\rho \frac{D\varepsilon}{Dt} = \frac{\partial}{\partial x_i}\left(\mathrm{Pr}_\varepsilon\, \lambda_{ef}\, \frac{\partial \varepsilon}{\partial x_i}\right) + C_{1\varepsilon}\frac{\varepsilon}{k}(G_k + C_{3\varepsilon}G_b) - C_{2\varepsilon}^*\rho\frac{\varepsilon^2}{k} \quad (2)$$

where G_k represents the generation of turbulence kinetic energy due to the mean velocity gradients, G_b is the generation of turbulence kinetic energy due to buoyancy, and Y_M represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate, calculated in the same way as for the standard k-ε model.

Turbulent viscosity is determined by this differential equation from RNG theory:

$$d\left(\frac{\rho^2 k}{\sqrt{\varepsilon\mu}}\right) = 1.72\frac{\hat{v}}{\sqrt{\hat{v}^3 - 1 + C_v}}\,d\hat{v} \quad (3)$$

where $v = \mu_{ef}/\mu$ and $C_v \cong 100$.

The effective Prandtl numbers are computed using the following equation:

$$\left| \frac{\frac{1}{Pr} - 1.3929}{\frac{1}{Pr_0} - 1.3929} \right|^{0.6321} \left| \frac{\frac{1}{Pr} + 2.3929}{\frac{1}{Pr_0} + 2.3929} \right|^{0.3679} = \frac{\mu}{\mu_{ef}} \quad (4)$$

where $Pr_0 = 1.0$. In the high-Reynolds-number limit ($\mu \ll \mu_{ef}$), $1/Pr = 1.393$.

The $C_{2\varepsilon}$ coefficient is replaced by:

$$C_{2\varepsilon}^* = C_{2\varepsilon} + \frac{C_\mu \rho \eta^3 (1 - \eta/\eta_0)}{1 + \beta \eta^3} \quad (5)$$

where $\eta = Sk/\varepsilon$ $\eta_0 = 4.38$ and $\beta = 0.012$.

Model constants are: $C_{1\varepsilon} = 1.42$, $C_{2\varepsilon} = 1.68$.

The numerical model was exported to Fluent for computation and initial results postprocessing. Some specific elements, for example Predicted Mean Vote (PMV), were calculated using Airpak routines.

4. Results and Discussion

The results indicate that the flow field is highly non-uniform both on the inside and on the outside of the building. A complex pattern of airflow is present on the inside, mainly driven by the 3000W heat source on the western wall. The contribution of the other heat sources (computer and lamps) is, nevertheless, significant. On the outside, large recirculation regions form on all sides except for the northern face, which is a typical behaviour for a bluff body flow, including the unstable, vortex-shedding wake. This suggests a highly variable (space and time) HTC distribution on the external surfaces. Although not verified with the current model, it is most probable that this HTC distribution greatly depends on the wind direction, and a full wind spectrum should be simulated and results recorded in the case of a real building design process.

The wind-chill factor is significantly felt including on the internal surfaces of the building, more visibly on the lower insulating parts, for instance the door and window surfaces, which suggests that the heat loss contribution of these areas is very important and should be carefully taken into consideration during the design stage.

The temperature field distribution and values recorded around the human model can be considered acceptable, therefore under the simulated conditions, the thermal regulating inside heat source seems to be adequate.

Figures 3 to 6 demonstrate some of the available post-processing capabilities: data contours on planes or slices with various orientations, external or internal pathlines, coloured by any given variable, data contours on domain boundaries or internal surface, etc.
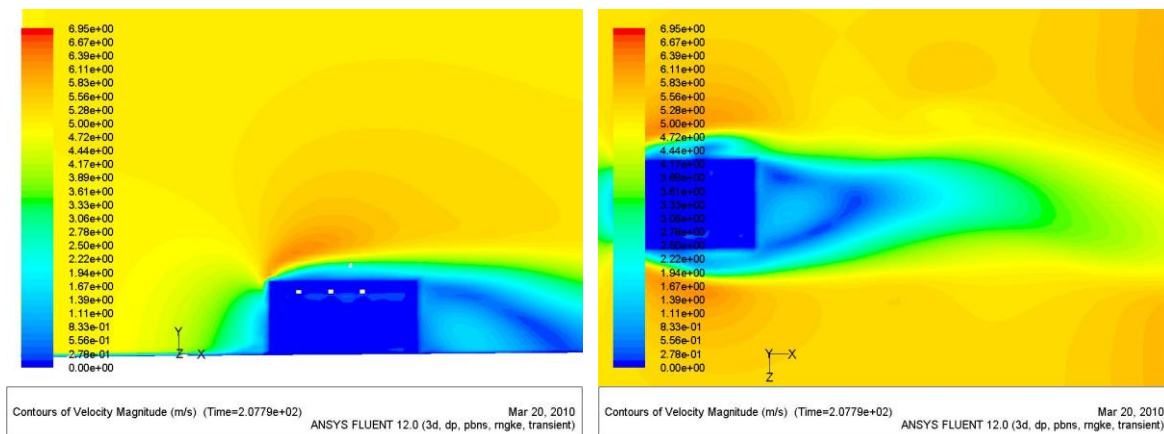

Figure 3. Contours of velocity magnitude, vertical plane (left) and horizontal plane (right)
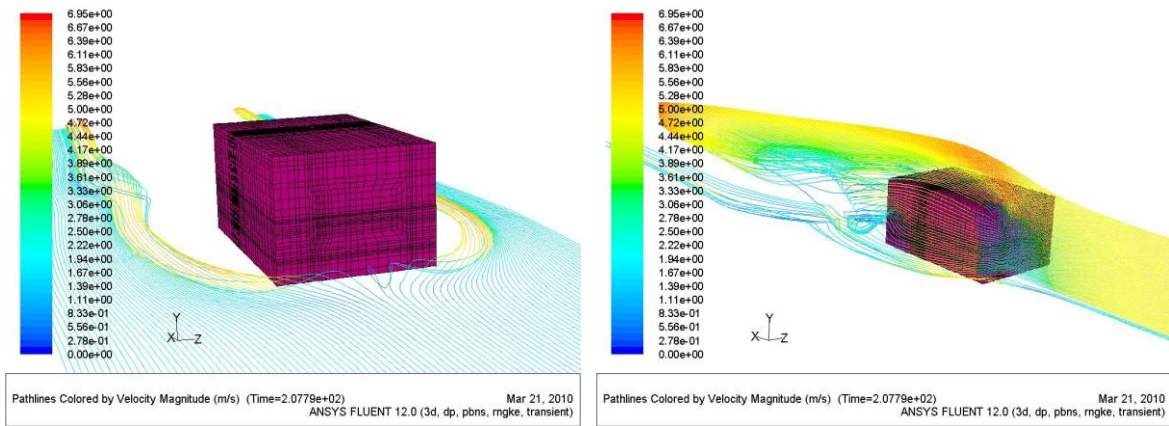
Figure 4. Outside air pathlines around the building, coloured by local velocity magnitude
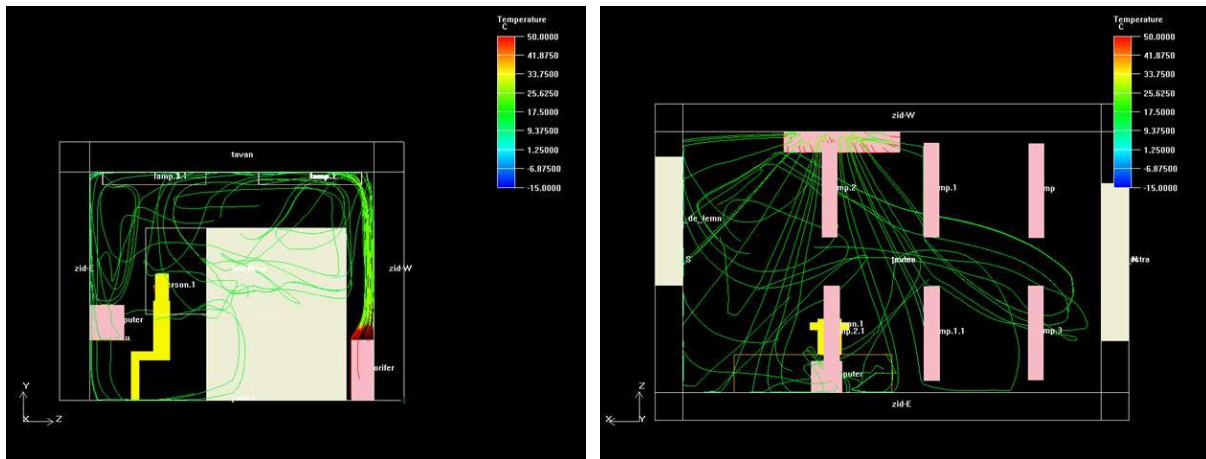

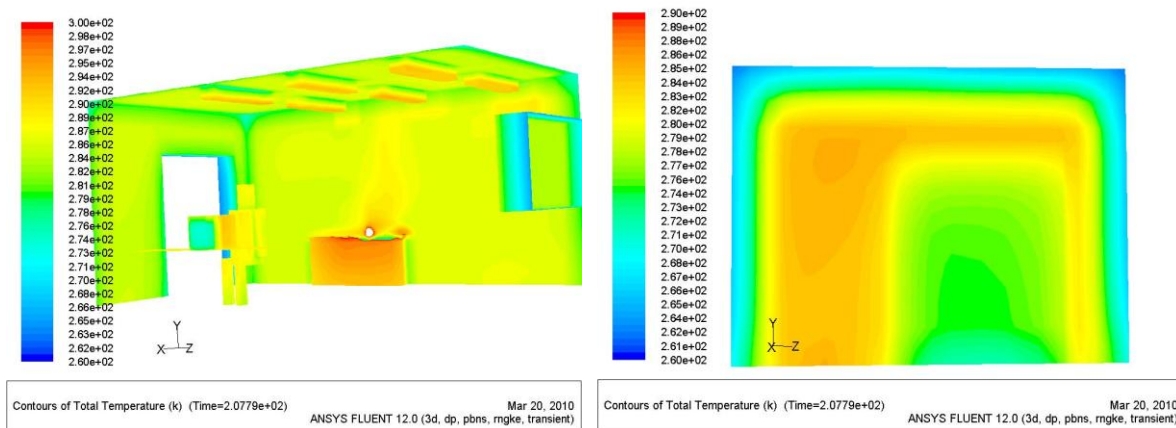Figure 5. Streamlines inside the room, coloured by local temperature

Figure 6. Temperature contours on the internal surfaces (left) and door wall (right)

5. Conclusions

The paper presents the modelling and simulation methodology for a simplified inhabited building, with internal heat sources, subjected to an external low temperature wind cooling. The results demonstrate that useful and comprehensive data can be easily obtained using the presented approach.

REFERENCES

1. Gao C.F. and Lee W.L., *Building and Environment*, 44, **2009**, 2080–2088.
2. Chao C.Y.H., and Wan M.P., *Building and Environment*, 39(7), **2004**, 749–62.
3. Airpak 3 – User manual and tutorials.
4. Ansys Fluent 12 – User manual and tutorials.
5. www.ansys.com