3rd International Conference
″Advanced Composite Materials Engineering ″
COMAT 2010
27- 29 October 2010, Brasov, Romania

# OPTIMISING A HVAC SYSTEM THROUGH NUMERICAL MODELING

**ing. Ştefan Bordei[1], prof.dr.ing. Florin Popescu[2]**
[1] University „Dunărea de Jos” of Galaţi, Galaţi, Romania, stefan.bordei@ugal.ro
[2] University „Dunărea de Jos” of Galaţi, Galaţi, Romania, florin.popescu@ugal.ro

***Abstract.*** *We modeled a complex problem of air flow and heat transfer inside and outside an enclosure taking into account outdoor climatic conditions. Based on the initial data we obtained an extremely complex problem which involves solving several simultaneous phenomena: (1) internally, a problem of heat transfer free convection and the study of air currents that are formed; (2) on the outside, a problem of flow around the building, combined with forced convection heat transfer between the airflow (the wind) and building. These phenomena are linked by simultaneously calculating (3) the heat flow through building walls from the inside out.*
***Keywords:*** *HVAC systems, numerical simulations, computational fluid dynamics laboratory.*

## 1. INTRODUCTION

Nowadays, the heating, ventilation and air conditioning systems design for buildings and civil engineering, in order to ensure the requirements of comfort and optimal working conditions, and not least, to optimize energy consumption, has become increasingly difficult.

Until the 90s, 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 appeared: the computational fluid dynamics, known as CFD, that synergetically completes the theoretical approach and experiment studies, providing to architects and engineers a useful tool in building design.

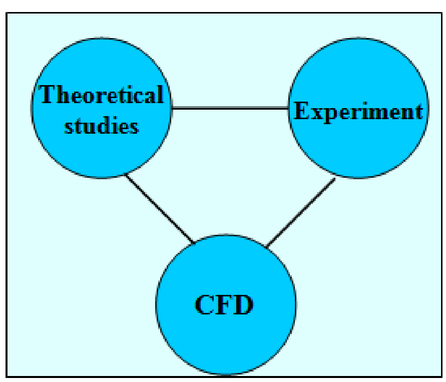


**Figure 1:** Synergy between theory, experiment and numerical modeling

There are several strong motivations regarding the integration of numerical simulations of fluid dynamics studies: better computing performances, lower design prices compared with experimental studies in laboratories, wind tunnels and test wells, easy access to information on physical quantities for a given model in any region of the field.

Based on these requirements, in order to increase the economic competitiveness and innovation, the University “Dunărea de Jos" of Galați, created a computational fluid dynamics laboratory which primarily aims to develop partnerships with industry and design institutes.

The hardware infrastructure of the laboratory is based on a powerful blade center with 60 processors. Meanwhile, the simulation tools consist in a commercial license of a complete Ansys platform which is one the best programs in computational fluid dynamics, structural mechanics, fluid-structure interaction etc.

In order to reflect the quality and performances of numerical simulations in designing HVAC systems, we conducted a study case.

## 2. STUDY CASE

We modeled 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 Lx*l*xH = 6m x 4m x 3m, with thick brick walls of 0.35 m, a glass window and a wooden door. The room is illuminated by six light sources each having an output of 34W. Inside the building there is a 80kg person working at a computer with a power of 173W. It was considered the heat from the body of the person who is wearing a cotton clothes line. Room heating is ensured by free convection by a heat source, with power of 3000W. The building is in an open space where the wind blows from N to S with a speed of 5m/s, the airflow temperature being of -15 degrees Celsius. Based on these initial data we obtain an extremely complex problem which involves solving several simultaneous phenomena:

1. internally, a problem of heat transfer free convection and the study of air currents that are formed;
2. on the outside, a problem of flow around the building, combined with forced convection heat transfer between the airflow (the wind) and building.

These phenomena are linked by simultaneously calculating

3. the heat flow through building walls from the inside out.
4.

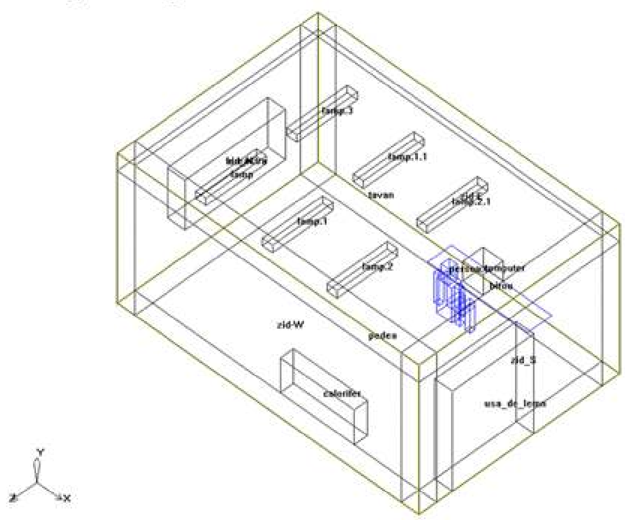


**Figure 2:** Interior boundaries conditions

Due to the low wind speed and the small size of the building, we neglected the effects of interaction between the air flow and the building structure. These phenomena are important for very tall buildings and can be also numerically modeled using the ANSYS program.

The solution was calculated using the programs ANSYS 12.0 CFD and Airpak. The preprocessing was done in Airpak while the solution and the preprocessing were calculated in ANSYS 12.0 CFD We illustrate in Fig. 3 to 8 are some suggestive pictures for velocity fields, pressures, temperatures, pathlines and PMV (predicted mean vote), both inside and outside the building.

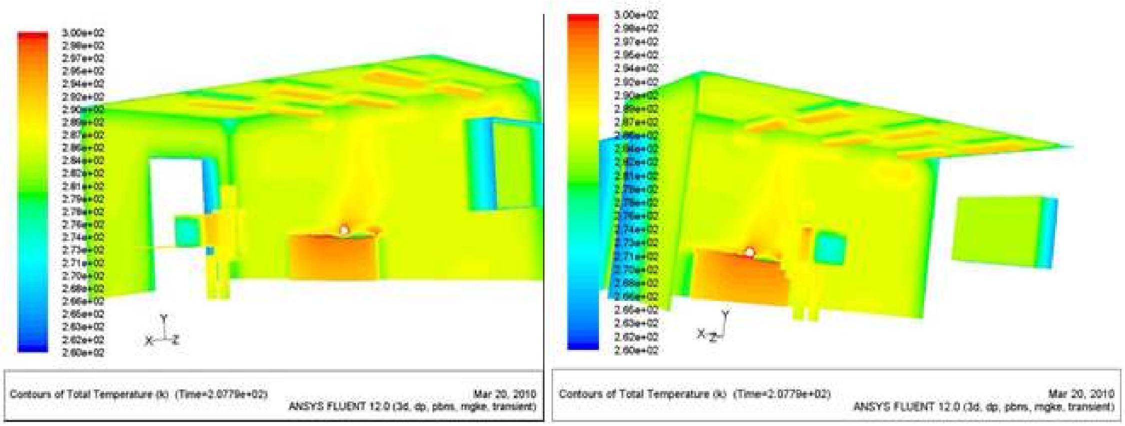


**Figure 3:** Temperature fields inside the building

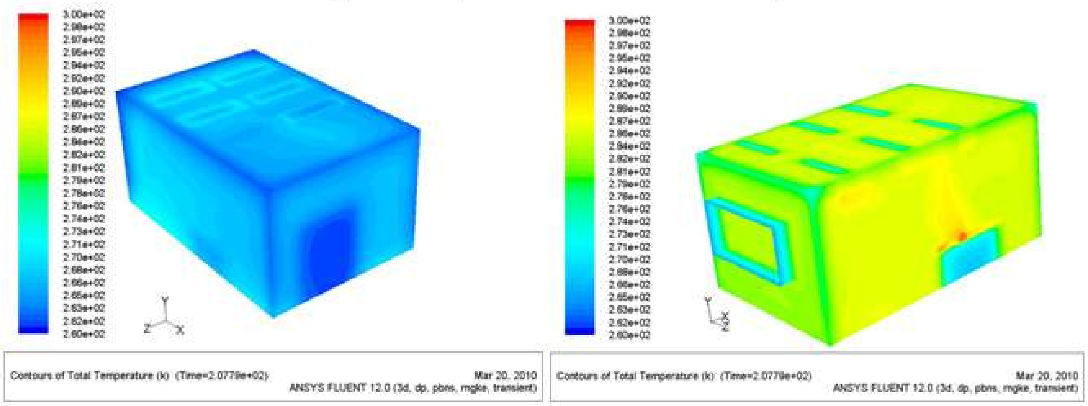


**Figure 4:** Temperature field on inside and outside building walls

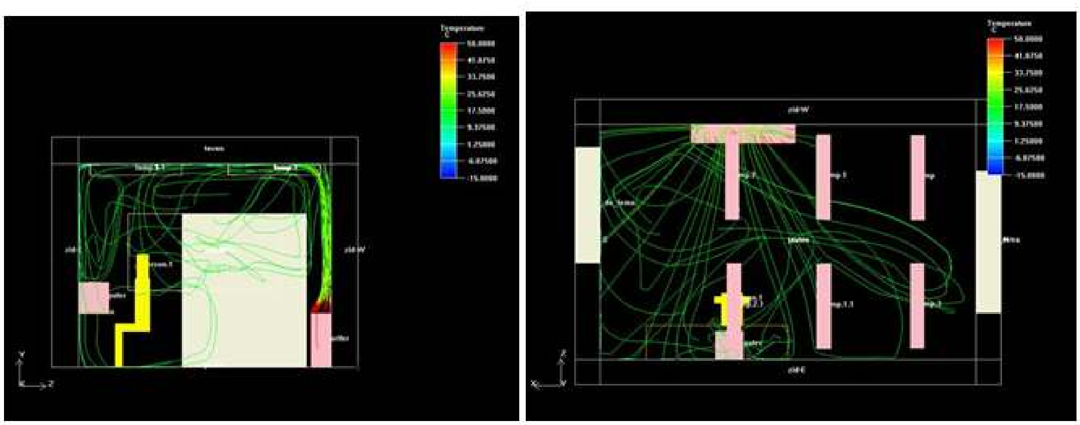

**Figure 5:** Streamlines inside the building

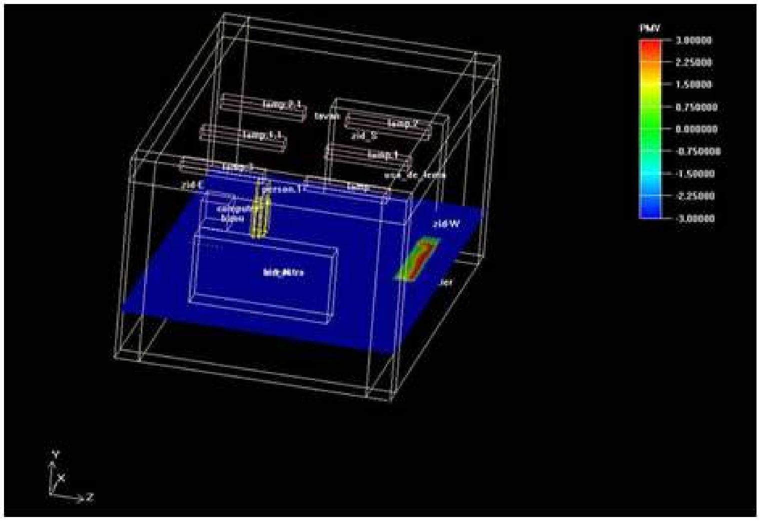

**Figure 6:** PMV – predicted mean vote at a certain level inside the buliding

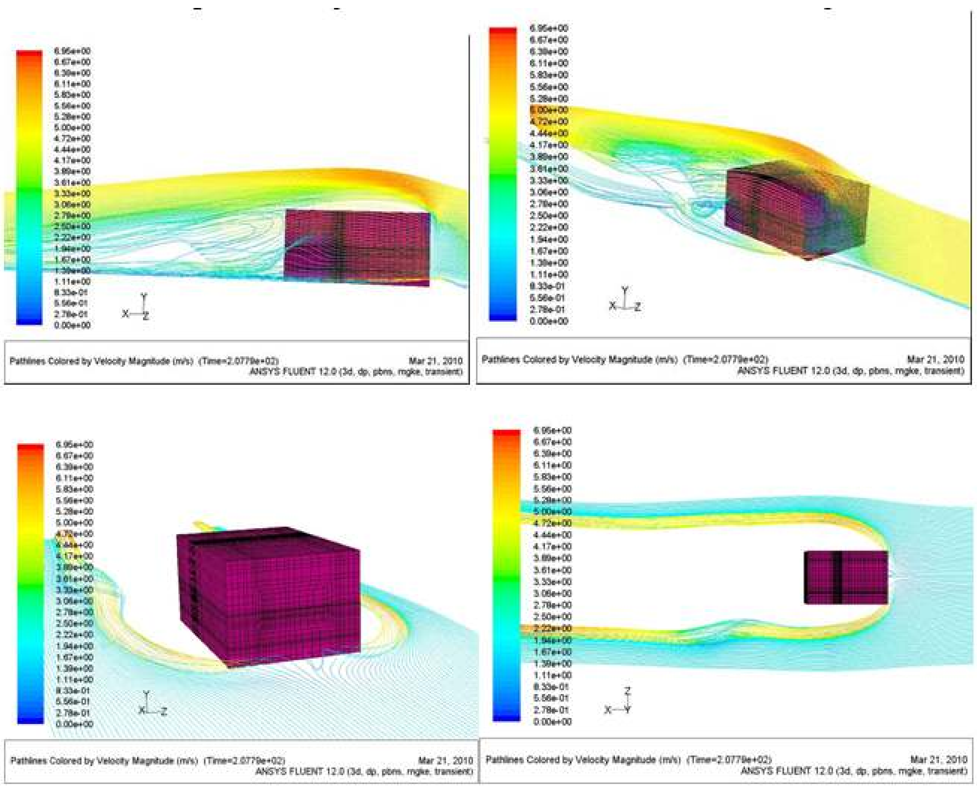


**Figure 7:** Outside air pathlines around the building

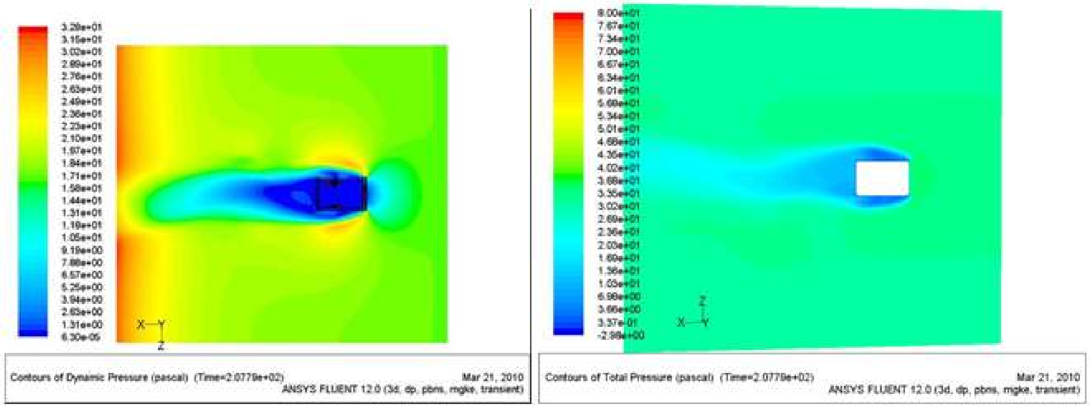


**Figure 8:** Contours of dynamic pressure and total pressure at a certain level outside the building

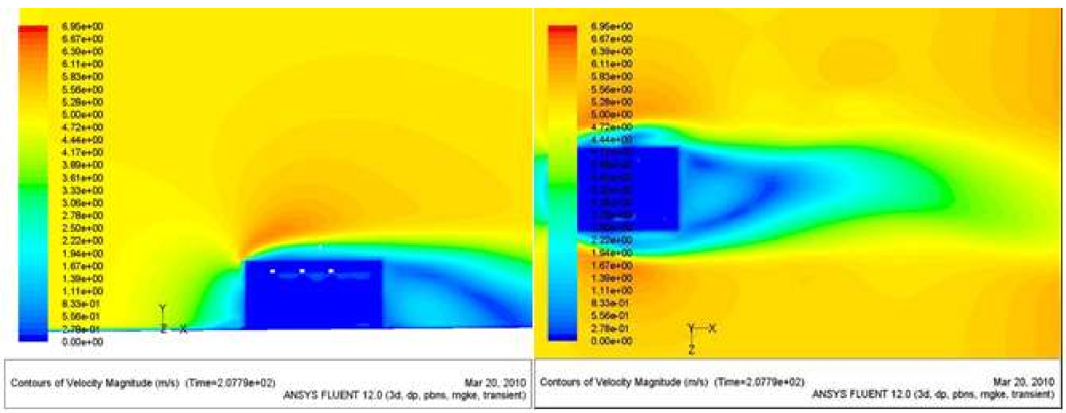


**Figure 9:** Contours of velocity magnitude in different outdoor sections

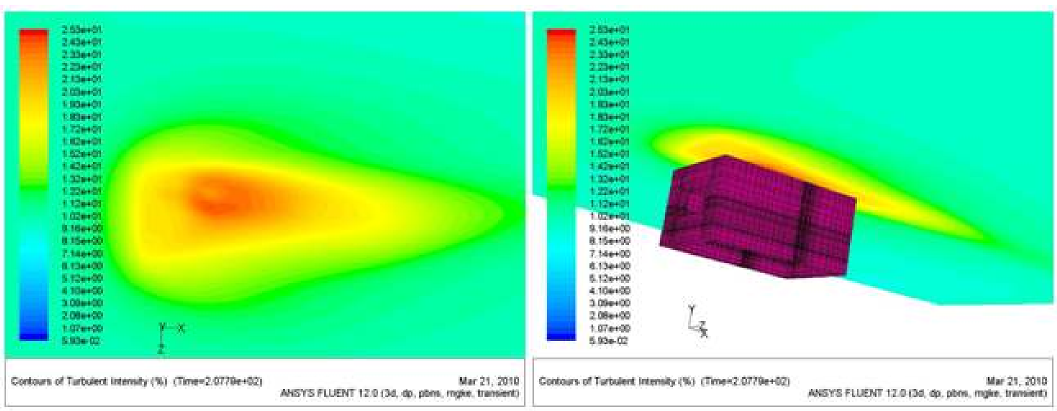


**Figure 10** Contours of Turbulent Intensity at a certain level over the building

## REFERENCES:

[1] Airpak – User manual and tutorial
[2] Ansys CFD 12.0 – User manual and tutorial
[3] www.ansys.com