CFD based design of a ventilated space

S G Mijorski¹, D G Markov², G T Pichurov², P Stankov², N G Ivanov³, I S Simova²

¹SoftSim Consult Ltd., Mladost-4, bl.438, 1715, Sofia, Bulgaria
²Centre for Research and Design in Human Comfort, Energy and Environment, Technical University of Sofia, 8 Kliment Ohridski Blvd., 1000, Sofia, Bulgaria
³Department of Fluid Dynamics, Combustion and Heat Transfer, Peter the Great St.Petersburg Polytechnic University, 29 Polytechnicheskaya str., St.Petersburg, 195251, Russia

peter.stankov@abv.bg

Abstract. The paper presents Computational Fluid Dynamics (CFD) based design of a densely populated classroom with mechanical ventilation. The aim is by means of numerical simulations to define the proper locations of four four-way ceiling fancoils in order to provide controlled conditions of the thermal environment in the classroom under summer conditions. CFD simulations are completed by Fluent 6.3 software package. Cooling loads through the enclosing surfaces of the room and the material properties of the modeled elements are evaluated by established and approved methods in the field of HVAC engineering.

1. Introduction
Thermal comfort of occupants is increasingly important economic factor and a very complex phenomenon [1]. The basic standards, which define the thermal comfort, are EN ISO 7730-2007 and ANSI/ASHRAE 55-2017 [2, 3]. Both standards apply the concept suggested by Fanger, [4]: a person is at thermal comfort when its whole body is at thermal balance (general thermal comfort) and there are no local discomforts, i.e. unwonted cooling/heating of some body parts. General thermal comfort is evaluated by the Predicted Mean Vote (PMV) index, which afterwards is used for evaluation of the percentage of people who will be dissatisfied with the actual thermal environment - Predicted Percentage of Dissatisfied (PPD) index. Local thermal discomforts, arising due to asymmetry in the thermal environment are radiant asymmetry, risk of draught (Draught Rating, DR), vertical temperature difference, and uncomfortable floor temperature.

Since a human body exchange heat with the surroundings at the skin by conduction, convection, radiation and evaporation, as well as within the lungs by convection and evaporation several physical parameters must be taken into account during the design of the indoor environment. With so many aspects to consider it is challenging to design a comfortable thermal environment in a densely occupied space with total volume ventilation. For this reason it is common to resort to the methods of Computational Fluid Dynamics (CFD).

The goal of this study is to design numerically the thermal environment in a classroom at the Technical University of Sofia following the ISO 7730-2007 standard, so that 24 students and a teacher are at thermal neutrality. The task is to select the positions of four four-way ceiling fancoils and the
angle of the louvers at their outlets, which will ensure the occupants’ thermal comfort during summer season.

CFD simulation predicted the air movement and heat exchange in the room. Fluent 6.3, [5], software package was used for the simulations. This package provides numerical algorithms for calculation and visualization of all pertinent flow parameters - velocity components, temperature and pressure. Comfort indices PMV, PPD as well as local discomfort due to draught (DR), were further programmed via a User Defined Function (UDF) following the ISO 7730-2007 standard [3]. Cooling loads through the enclosing surfaces of the room and the material properties of the modeled elements were evaluated by established and approved HVAC procedures.

At this stage of the design of the thermal environment, simplification was made assuming equally distributed flow rate through the four outlets at each fancoil and uniform velocity distribution throughout. The supply air direction was assumed identical to the inclination of the louvers.

2. Physical model

Figure 1 presents the geometry of the classroom. Although it’s a box shaped room, the presence of columns, beams, window sills, and radiators makes its geometry fairly complex. A door is located on the west wall and all windows are on the east wall. Air volume of the classroom is 193.84 m³.

Based on the preliminary HVAC calculations, it is decided to use four four-way ceiling fancoils TOSHIBA MMU-AP0092H. On the west wall of the room four air transfer grills are located. Two cases with respect to fancoil positions as illustrated on Figure 2 are identified as possible. Case 1 is with distance between the geometrical centers of fancoils along X direction \( \delta_{xc}=100 \text{ cm} \), and Case 2 is with \( \delta_{xc}=80 \text{ cm} \). For both cases Y coordinates of the geometrical centers of the fancoils are the same.

Cooling loads through all enclosing surfaces of the room are calculated based on the data about their geometry, construction materials, prescribed indoor air temperature and outdoor conditions – temperature and solar insolation. Thermal loads for all boundary elements, internal heat sources (occupants) and lighting units are defined based on Methodological guidelines for calculating the annual energy consumption, heat, humidity and energy load of buildings and individual hazardous substances [7].

3. Computational details

CFD simulations are based on steady-state Reynolds-Averaged Navier Stokes (RANS) equations, including the energy equation and Discrete Ordinate (DO) radiation model, Boussinesq approximation of buoyancy forces, the standard k-e turbulence model and the respective boundary conditions. Numerical simulations use the pressure-based steady solver for RANS equations of Fluent 6.3, [5].

3.1. Computational grid

The computational grid was prepared in Gambit 2.4.6 mesh generator, comprising 10 cm step structured rectilinear grid, which can reduce numerical errors and guarantee faster convergence. For the purpose of the CFD simulations classroom geometry is slightly modified in order to use 10 cm structured rectangular grid. Heated dummies simulate occupants. Figure 3 shows numerical representation of 24 seated occupants and one standing upright. Figure 4 shows all elements in the computational domain for Case 1 (\( \delta_{xc}=100 \text{ cm} \)).

As shown on Figure 5a, the geometry of the fancoils is too complex. For the purpose of the study it is simplified as shown on Figure 5b.

3.2. Boundary conditions

Boundary conditions for the energy equation are presented in Table 1. The internal objects - desk, columns, and radiators participate only as obstacles, but don’t exchange any heat. For simplification purposes the cooling load of lighting and ceiling is combined. This increases the total load of the ceiling from 1051 to 1251[W]. All elements are split in five different types of materials with corresponding thermal properties. The model features four pressure outlets on the west wall (the air
transfer grills), four velocity outlets (the fancoils inlets), and 16 mass flow inlets (the four outlets of each fancoil). Boundary conditions on the fancoil surface are given in Table 2 and Table 3.

![Classroom geometry: (a) plan; (b) section A-A; (c) section B-B.](image)

**Figure 1.** Classroom geometry: (a) plan; (b) section A-A; (c) section B-B..

![Simulated fancoil locations: a) δ_{ccc}=100cm; b) δ_{ccc}=80cm.](image)

**Figure 2.** Simulated fancoil locations: a) δ_{ccc}=100cm; b) δ_{ccc}=80cm.
Mass flow inlets are divided in four different groups depending on which side of the fancoil are: East, West, North, and South. This gives possibility for group setting of boundary conditions about louvers inclination, Figure 6.
Table 1. Numerical model components’ boundary conditions.

| Component               | Type of material | Numerical area [m²] | Total cooling load [W] | Heat Flux [W/m²] |
|-------------------------|------------------|---------------------|------------------------|-----------------|
| Outer wall (East)       | Outer walls      | 17.01               | 39.38                  | 2.32            |
| Inner wall (West)       | Inner walls      | 31.32               | 365.40                 | 11.67           |
| North wall              | Inner walls      | 16.55               | 168.00                 | 10.15           |
| South wall              | Inner walls      | 16.55               | 133.14                 | 8.04            |
| Windows                 | Glass            | 21.24               | 995.10                 | 46.85           |
| Floor                   | Floor and Ceiling| 59.99               | 1051.00                | 17.52           |
| Ceiling + lamps         | Floor and Ceiling| 68.61               | 1251.00                | 18.23           |
| Sited occupants         | Skin and clothing| 48.00               | 2088.00                | 51.41           |
| Standing occupant       | Skin and clothing| 2.12                | 108.98                 | 43.50           |

Table 2. Fancoils – Velocity Outlets.

| Recirculation elements | Velocity Inlet [m/s] | Turbulence Intensity [%] | Hydraulic Diameter [m] | Temperatures [C] |
|------------------------|----------------------|--------------------------|-------------------------|------------------|
| 1, 2, 3 and 4          | -0.692344            | 10                       | 0.2                     | 24.5             |

Table 3. Fancoils – Mass Flow Inlet.

| Mass Flow Inlet elements | Mass Flow Rate [kg/s] | Turbulence Intensity [%] | Hydraulic Diameter [m] | Temperatures [C] |
|--------------------------|-----------------------|--------------------------|-------------------------|------------------|
| East, North, South and West | 0.226776             | 10                       | 0.5                     | 17.7             |

3.3. Simulated cases

In the analysis 10 simulation cases are considered. For each of the two cases with respect to the distance between the fancoil centres along X-direction are simulated five different inclinations (α – the angle between the plane of the louver and the ceiling) of the louvers at the mass flow inlets. Cases of the inclination for both models are shown on Figure 6. The louver inclination is modelled by changing velocity profile on mass flow inlets.
3.4. Operating conditions and initial guess

Operating conditions are set to: Atmospheric pressure: 94500, [Pa]; Gravity acceleration: -9.81 in Z directions, [m/s²]; Operating temperature: 24.5, [°C].

Initial guess for simulated variables are as follows: Turbulent Kinetic Energy: 0.03747055, [m²/s²]; Turbulent Dissipation Rate: 0.0048341, [m²/s³]; Temperature: 23.89264, [°C].

4. Results about air velocity and temperature field

Here are presented only the results for Case 1 ($\delta_{pp}=100\text{cm}$) with blade angle $\alpha=30^\circ$. The variation of both air temperature and velocity magnitude are visualized in the Y-Z plane at X=2.6 m (between the fancoils – Figure 2) and in the X-Y planes at Z=0.6 m (the body centre of gravity for a seated person) and Z=1.1 m (head level of a seated person).

Air temperature variation in the classroom is visualized on Figure 7 and Figure 8 and air velocity variation on Figure 9 and Figure 10.
5. Results about PMV, PPD, and DR
The purpose of the simulations is to determine the basic parameters for the thermal comfort of occupants in the investigated room. The calculation of these parameters is based on the EN ISO 7730:2007 standard [3] and is described with full details in [6].

Risk of draught, which characterizes the local heating/cooling of the body by convection, depends on the local features of both the air velocity field (mean velocity and turbulence intensity) and air temperature field. In EN ISO 7730-2007 it is defined as

\[
DR = (34 - t_a)(\bar{V} - 0.05)^{0.62}(0.37\bar{V}T_u + 3.14)
\]

where DR is the percentage of people dissatisfied due to draught, %; \(t_a\) is the local air temperature, °C; \(\bar{V}\) is the local mean velocity (magnitude), m/s; \(T_u\) is the local turbulence intensity, %.

Results about DR, PMV, and PPD variation in the horizontal planes Z=0.6 m and Z=1.1 m are presented on Figure 11, Figure 12 and Figure 13, respectively.

![Figure 11. DR, Case 1, α=30°: a) Z=0.6 m; b) Z=1.1 m.](image)

![Figure 12. PMV, Case 1, α=30°: a) Z=0.6 m; b) Z=1.1 m.](image)

![Figure 13. PPD, Case 1, α=30°: a) Z=0.6 m; b) Z=1.1 m.](image)
6. Analysis and discussion
Based on equation 1 in order to avoid draught discomfort at head level it was decided to use four four-way ceiling fancoils, for air treatment, though only 2 are enough to remove the excess heat from the room. With four fancoils the flow rate of the recirculated air is increased but the temperature difference between the air in occupied zone and the cooled air is decreased to less than 2°C. Figure 11b shows that it is a good decision. There are a few zones only, far from the occupants, were it is possible to have 10% ≤ DR ≤ 20%, which meets the requirements for category II of the EN ISO 7730-2006 standard.

EN ISO 7730-2006 standard puts requirements for PMV index at the point where the occupant’s body center of gravity will be located. At all workplaces PMV ≤ 0.7, (Figure 12a). This meets the requirements for category III of the standard. The PPD distribution shows that by rearrangement of the work places it is possible to ensure that no more than 15 % of the occupants would be dissatisfied with the general thermal comfort.

The CFD simulation results show that the optimal location of the air distribution units and louvers inclination at their outlets is successfully identified.

This design was later physically implemented in the simulated room and used for further experimental studies.

7. Conclusions
The study employed the methods of CFD in the design stage of a total volume ventilation system. There is clear evidence that CFD based simulations of room air movement is the only effective tool in design optimization of HVAC systems. The simulated design was later installed and commissioned for experimental studies. It pays off to put extra coding effort and calculate thermal comfort indices that are otherwise not part of the classical CFD. The thermal comfort distribution allows not only to select a proper ventilation design, but also to allocate workplaces under optimal thermal conditions that will reduce human complaints and work absence due to illness.

Acknowledgments
The present study is supported by Bulgarian Science Fund of the Ministry of Education and Science (grant № ДНТС/Русия 02/11 from 15.06.2018) and by the Russian Foundation for Basic Research (grant № 18-58-18011).

References
[1] Angelova, R. A. 2016 Textiles and human thermophysiological comfort in the indoor environment, Boca Raton (US)
[2] ANSI/ASHRAE Standard 55-2017 2007 Thermal environmental conditions for human occupancy
[3] EN ISO 7730 2006 Ergonomics of the thermal environment – Analytical determination and interpretation of thermal comfort using calculation of the PMV and PPD indices and local thermal comfort criteria
[4] Fanger P O 1972 Thermal Comfort: Analysis and Applications in Environmental Engineering, New York: Mcgraw-Hill
[5] Fluent 6.3 2007 User’s guide, Fluent 6.3 documentation, Fluent Inc.
[6] Markov D. 2006 Methodology For Thermal Environment Assessment in Practice, EuroAcademy on Ventilation and Indoor Climate, ISBN – 13: 978-954-91681-4-3
[7] Methodological guidelines for calculating the annual energy consumption, heat, humidity and energy load of buildings and individual hazardous substances 2007 ISBN 978-954-8873-83-3, (in Bulgarian)