Available online at www.sciencedirect.com

ScienceDirect

Energy Procedia 112 (2017) 252 – 260

Energy Procedia

Sustainable Solutions for Energy and Environment, EENVIRO 2016, 26-28 October 2016, Bucharest, Romania

Experimental and numerical investigation of mechanically ventilated rooms

Raluca Teodosiu[a]*, Damien David[b,d], Cătălin Teodosiu[a], Gilles Rusaouën[b,d], Viorel Ilie[a], Chi-Kien Nguyen[c,d]

[a]*Technical University of Civil Engineering, 122 – 124 Lacul Tei Bvd, Bucharest 020396, Romania*
[b]*Université de Lyon, 69361 Lyon Cedex 07, France*
[c]*INSA-Lyon, 20 Avenue Albert Einstein, 69100 Villeurbanne, France*
[d]*CETHIL UMR5008, 69621 Villeurbanne Cedex, France*

Abstract

The aim of this study is to make available new data concerning the validation of CFD (Computational Fluid Dynamics) results through experimental measurements campaigns on full scale mechanically ventilated test rooms. The experimental set-up is based on a new configuration of the test cell "Minibat" (CETHIL – INSA Lyon, France), equipped with a realistic mixing ventilation system. In this paper, experimental – numerical comparisons are presented in terms of mean air velocity for an isothermal situation. The investigations have been focused in the region near the supply air diffuser as the overall air flow in ventilated enclosures is mainly due to the inlet air jet. The results show a decent agreement between measurements and numerical values in terms of velocity profile at the inlet. In addition, the overall flow development in the air jet is well represented numerically. However, there are some discrepancies concerning the point of maximum velocity in the air jet which must be addressed by further investigations, both experimentally and numerically.

Peer-review under responsibility of the organizing committee of the international conference on Sustainable Solutions for Energy and Environment 2016

Keywords: full-scale test room; CFD modeling; mechanically ventilated rooms; experimental validation

* Corresponding author. Tel.: +40-21-252-4620; fax: +40-21-252-6880
E-mail address: ralucahohota@yahoo.com

Peer-review under responsibility of the organizing committee of the international conference on Sustainable Solutions for Energy and Environment 2016
doi:10.1016/j.egypro.2017.03.1094

1. Introduction

As most people spend over 90% of their time indoors, room occupants' comfort and indoor air quality are becoming extremely important issues nowadays. On the other hand, it is well known that the airflow in indoor spaces is one of the main factor that influences the environmental control of buildings. Consequently, the indoor environment studies have lately known an accelerated expansion. In this context, the CFD (Computational Fluid Dynamics) approach is more and more used for analyses concerning the prediction of air movement in ventilated enclosures [1]. In fact, we can affirm today that the CFD technique is a useful tool, largely adopted by the indoor environment research community [2] for investigations dealing with ventilation efficiency [3-5], indoor air quality [6-8] and thermal comfort [9-11].

Despite this, there still is a major challenge: validation of CFD results by pertinent experimental data. As a result, the objective of this study is to assess the accuracy of numerical results based on full-scale measurements for mechanically ventilated rooms. It is worthwhile to mention that this study is part of a research project on experimental and numerical analyses for thermo-solutal-convection phenomena for humid air in ventilated rooms. We analyse here the results achieved during this project for tests carried out in isothermal conditions.

We first present the experimental set-up based on a full-scale test room facility, followed by the description of the CFD model. We conclude with experimental – numerical comparisons in terms of velocity fields within the ventilated enclosure.

2. Experimental set-up

This work is completely based on the experimental investigations accomplished in the experimental full-scale test room "MINIBAT" (CETHIL – INSA Lyon, France). "MINIBAT" contains three distinct zones: a test cell, a thermal buffer zone and a climatic chamber (see Fig. 1). The experiments are carried out in the test cell. A major change was made compared to the configuration taken into account by Hohotă [12]: the cell is now composed of one single piece whose dimensions are 6.27x3.10x2.50 m (length x width x height). Five of its facades are surrounded by the thermal buffer zone that allows maintaining a stable temperature. The South facade is a glazing wall that is in contact with the climatic chamber. The purpose of this climatic chamber is to simulate outdoor conditions.

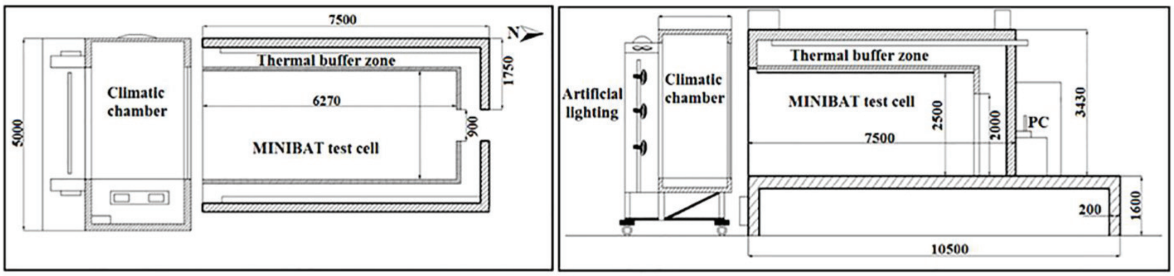


Fig. 1. Test room "MINIBAT": cell plan view and section view

The air temperature inside the thermal buffer zone is entirely regulated using controlled mechanical ventilation, operating in closed loop. Air lateral diffusers are installed in the upper part of the thermal buffer zone and air extraction in the lower part throughout the East and West walls. This disposition promotes regular and homogeneous air mixing.

The climatic chamber is also controlled in temperature using an air handling unit which can vary air temperature between -10°C and +40°C.

The outer envelope, which forms the thermal buffer zone, was made of 20 cm thick cellular concrete. In order to increase the thermal inertia, we added a layer of thick Styrofoam panels (thickness 10 cm) outside of the envelope. This reinforcement of the structure (on the roof and on the East and West walls) leads to lower fluctuations of the air temperature inside the thermal buffer zone (±0.6°C compared to ±1.5°C in the case of the previous structure, according to [12-14]).

As for the test cell, the opaque vertical walls consist of agglomerated wood panels (5 cm thick), covered on their internal faces by plasterboard (1 cm thick). The floor is made of cellular concrete having the same characteristics as those of the thermal buffer zone envelope. The ceiling comprises a 2.5 cm thick plywood plate, covered with 4.5 cm glass wool. In addition, 1 cm thick plasterboard is fixed on the lower part of the ceiling. The glazing wall (1.2 cm thick) installed on the South façade is semi-transparent.

The air in the test cell is supplied by an air handling unit (AHU) installed on the roof of an adjacent local. The test room is provided with a mixing ventilation system: the air inlet is located on the ceiling and the two air exhausts are located in the North wall, near the floor (Fig. 2).

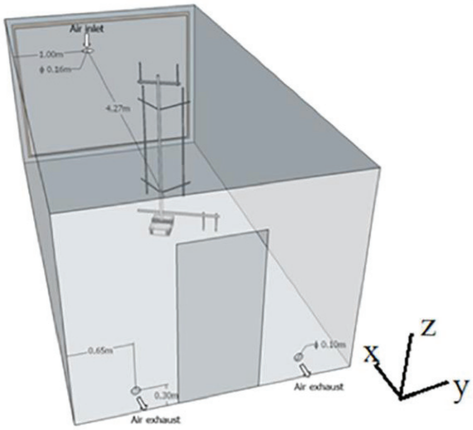


Fig. 2. Ventilation system of the test room “MINIBAT”

By integrating a proportional-integral-derivative (PID) controller, the AHU is able to regulate the airflow and the air temperature entering the test cell. The AHU is operating in closed loop so that the fluctuations due to external conditions can be reduced. All the ductwork is also thermally insulated in order to reduce heat loss throughout the network.

The test cell metrology is based on measurements regarding the boundary conditions (cell wall temperature, thermal buffer zone and climatic chamber temperature, air supply parameters) and experimental data on velocity, temperature and moisture fields within the test cell.

Interior and exterior cell wall temperature measurements can be recorded continuously thanks to a network of 180 Pt100 probes, with an accuracy of ±0.2°C. All the walls surface have 36 measurement points (18 probes on the wall internal surface and 18 probes on the wall external surface), except the North and South sides, with 18 measurement points (in this case, 9 probes on the wall internal surface and 9 probes on the wall external surface).

Pt100 probes are also employed to determine the temperature (accuracy ±0.2°C) in the following locations: AHU supply, exhaust air, cooling coil and supply air in the test cell. In addition, three humidity probes SHT75 are installed at the first three locations previously mentioned, in order to obtain the measurements of relative humidity (accuracy ±1.8%) and specific humidity thanks to integrated temperature measurement (accuracy ±0.3°C).

On the other hand, a propeller flow meter (uncertainty ±0.5 m^3/h) is used to measure the air supply flow rate.

In order to determine the air velocity, air temperature, and air humidity fields within the test cell, a mobile robot equipped with various sensors has been designed and programmed by ourselves (Fig. 3). The robot chassis is made of aluminum alloy whose dimensions are 40x30x12 cm, which is robust enough to support a load of 15 kg. The main mast is formed of carbon fiber to have a good compromise between lightness and rigidity. Its displacement is made possible thanks to 4 WD mecanum wheels (10 cm diameter), connected to four DC brushless motors. These motors are supplied and controlled by an Arduino microcontroller board, which is connected to a mini PC, installed on the horizontal mast (Fig. 3). The robot steering is operated by remote connection method, using another PC located outside the test cell. Finally, a light detection and ranging system (LiDAR) is added in order to know the exact location of the robot in the cell.

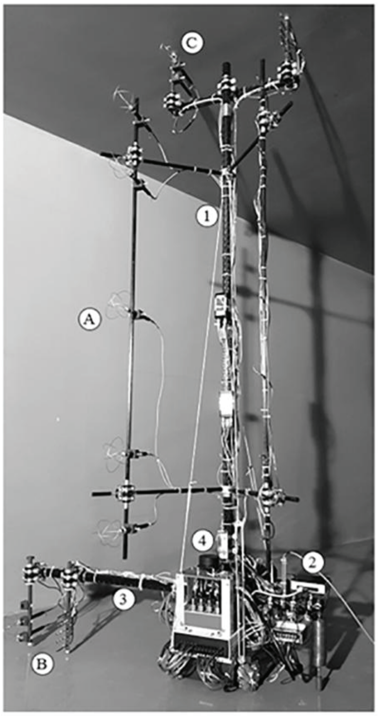


Fig. 3. Mobile robot scheme of the test room “MINIBAT”
1 – main mast; 2 - mini PC Intel® NUC; 3 – horizontal mast; 4 – LiDAR system

Concerning the sensors installation, the following three profiles have been implemented, in order to track regions characterized by high gradients (Fig. 3): center (A), near the floor (B) and near the ceiling (C), with a higher number of sensors for the last two profiles. Consequently, air velocity, air temperature and air humidity fields measurements can be performed as it follows:

- air velocity: 10 thermoelectric anemometers, with 5 probes installed at center profile (measurement range between 0.15-5 m/s); 2 probes near the ceiling and 3 probes near the floor (measurement range between 0.015-1 m/s). At room temperature, the uncertainty of these anemometers is ±1.5% of measured value; the acquisition frequency is 10 Hz (the recorded values are averaged using 2000 values, meaning 200 seconds);
- air temperature: 27 fast-response thermocouples type K (diameter 25µm, ±0.1°C); the acquisition frequency is 10 Hz (the recorded values are averaged using 10 values, meaning 1 second);
- air relative humidity: 17 SHT75 probes (accuracy ±1.8%); the acquisition frequency is 0.5 Hz.

In general, compared to a mobile arm device used in other studies based on “MINIBAT” test room [12,13,15,16], the mobile robot system is able to reduce air flow disturbances thanks to less materials installed inside the test cell. In addition, it also allows us to acquire experimental data in any area of interest, avoiding regular measurement meshes as it was happened before [12,13,15,16].

3. CFD model

All the numerical investigations presented in this work are based on a general-purpose, finite-volume, Navier-Stokes solver (Fluent 15.0.0). The principal characteristics of the numerical model are shown in Table 1.

Concerning the airflow modeling, the appropriate approach for the description of the turbulence is crucial for accurate prediction of air distribution within the studied test room. As a result, the selection of the turbulence model is extremely important in the CFD simulation process. On the other hand, when using a turbulence model, a good compromise between precision, hardware resources and computational time must be taken into consideration. The choice of the suitable turbulence model is also influenced by the objective of the study. For example, in the case of investigations dealing with airflow in ventilated rooms, the mean air parameters represent the most relevant results compared with turbulent characteristics of the airflow. Considering all this, we use in this study a revised k-ε turbulence model, the "realizable" k-ε model [17], as the replies of this approach, at all the issues mentioned before, can be considered satisfactory. In addition, it is shown that this turbulence model leads to good predictions concerning the airflow in mechanically ventilated rooms [18].

Table 1.Numerical model main elements and hypotheses.

Feature	Description
Air	Newtonian fluid; incompressible; constant viscosity
Flow	Three-dimensional; steady; isothermal; turbulent
Computational domain discretization	Finite volumes; unstructured mesh (tetrahedral elements); 10,191,248 cells (grid independent solutions)
Turbulence model	Reynolds-averaged Navier-Stokes (RANS), eddy viscosity model: "realizable" k-ε (two-equation model) [17]
Near wall treatment	Enhanced wall treatment (two-layer model with improved wall functions) [18]
Numerical resolution	Segregated implicit solver; diffusion terms: second order central-difference scheme; convective terms: second order upwind schemes; velocity-pressure coupling: SIMPLE algorithm; convergence acceleration: algebraic multigrid (gradient method, Green-Gauss cell based)
Boundary conditions	Air supply: velocity – fixed values across the inlet (ratio of the measured air flow rate to the inlet area); turbulence quantities: uniform specification, defining two parameters (turbulence intensity and hydraulic diameter); wall: no slip boundary conditions; air exhaust: longitudinal exit velocity from mass balance and transverse velocity components set to zero.

The near-wall modeling is based on the so-called "enhanced wall treatment" that combines a two-layer approach with enhanced wall functions. Generally, a fine near-wall mesh (characterized by y^+ values around 1, see Table 2) was used, in order to be able to solve the viscous sublayer using the classic two-layer methodology for the boundary layers. On the other hand, it is worthwhile to mention that grid dependency tests have revealed that finer meshes (e.g. 12,101,415 cells) do not substantially improve the y^+ values and coarser grids (e.g. 6,374,973 cells) lead to increased values of y^+ (Table 2).

4. Results

The results are presented for the experimental conditions shown in Table 3. The test taken into account is representative for an isothermal situation, according to air supply temperature and mean air room temperature.

As the air flow within a mechanically ventilated room is mainly governed by the air jet issued from the inlet, we focus our experimental – numerical comparisons in terms of mean air velocity in the region near the supply air diffuser.

Table 2. Boundary layer mesh: Y^+ mean values

Wall	Coarser mesh (6,374,973 cells)	Used mesh (10,191,248 cells)	Finer mesh (12,101,415 cells)
South	3.155	1.027	1.991
North	2.673	2.104	2.156
West	1.874	1.143	1.141
East	1.559	1.983	1.018
Ceiling	1.798	1.212	1.155
Floor	4.316	2.397	2.377

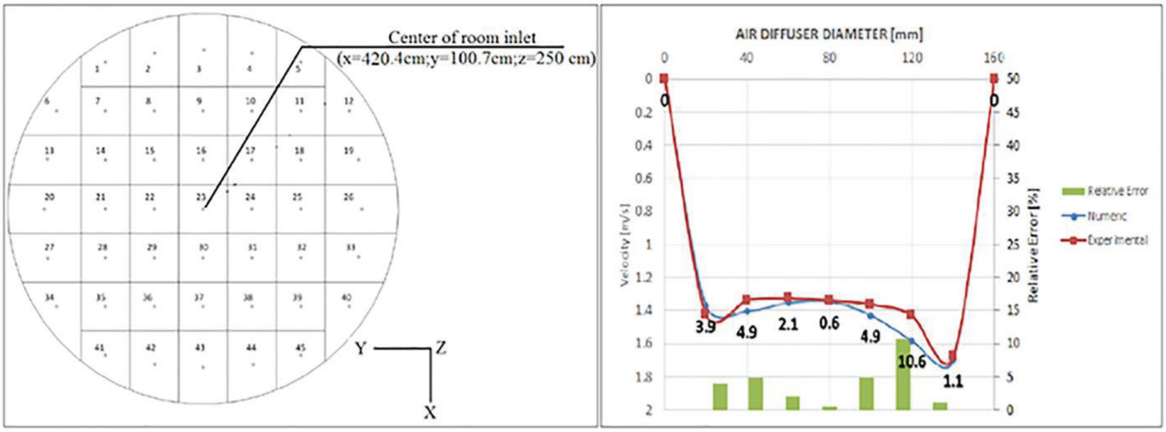
The first data series refers to the inlet supply velocity profile. For this section, 45 points have been taken into account for measuring air velocity. These points are in the center of a square grid with elements of side 2 cm, starting from the center of the inlet (see Fig. 4a). The inlet supply velocity profiles presented in Fig. 4b are achieved in the following manner: each of the 7 points (from 20 to 140 mm on the diameter of the inlet) is obtained as the average experimental and numerical point values located on the same line (points 1-5, 6-12, 13-19, 20-26, 27-33, 34-40, and 41-45, respectively). The results in Fig. 4b show a good agreement between measurements and numerical values. Nevertheless, there is a numerically overestimation of the air velocity, especially at 120 mm on the diameter of the inlet.

Table 3. Experimental conditions.

Parameter	Value
Air flow rate - air supply (m^3/h; h^{-1})	100.7; 2.1
Air supply temperature (ºC)	20.1
Air exhaust temperature (ºC)	22.2
Mean air test room temperature (ºC)	21.5
Internal surface ceiling mean temperature (ºC)	22.3
Internal surface floor mean temperature (ºC)	22.4
Internal surface North wall mean temperature (ºC)	21.7
Internal surface South wall mean temperature (ºC)	22.4
Internal surface West wall mean temperature (ºC)	22.2
Internal surface East wall mean temperature (ºC)	22.2

In addition, the jet region has been investigated by means of 50 points positioned at 5 different heights in the vertical plane passing through the center of the inlet (Fig. 5). The results obtained are shown in Fig. 6. The experimental – numerical relative errors are also inserted in Fig. 6 next to every point. It is worthwhile to mention that errors over 50% are recorded only in points where the air velocity has low values (generally less than 0.02 m/s).

On the other hand, it can be seen that the overall flow development is well represented numerically. In addition, the maximum air velocity in the jet and its decay are rather well predicted by the numerical model. Nevertheless, the maximum velocity determined experimentally lies along the centre-line of the jet for all the 5 heights taken into account. This does not happen in the case of numerical results, the point of maximum velocity being increasingly shifted as the air jet develops. This issue must be elucidated in further research. For instance, the explanation can be an inadequate numerical description of the initial jet development (as can be noticed in Fig. 4b – the inlet velocity profile is asymmetric overrated in the numerical model) or, there is an influence on the air flow in that region, due to experimental devices.



a) b)

Fig. 4. a) Measurement points at the inlet; b) Supply velocity profile at the inlet.

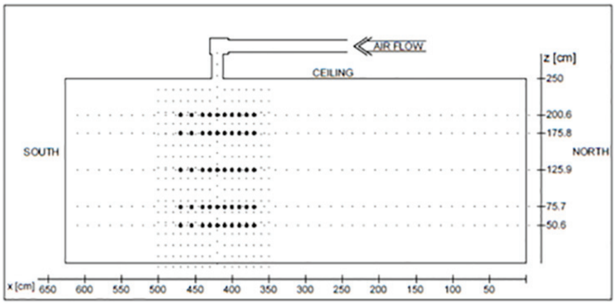


Fig. 5. Position of the measurement points in the jet region

5. Conclusions

The encouraging results achieved allow for further investigations in anisothermal conditions and in the presence of moisture in order to make available useful data concerning dynamic, thermal and humidity fields in mechanically ventilated rooms. On the other hand, deeper analyses are needed to ameliorate the predictions of air velocity in the ventilated test room. This can be fulfilled by special approaches concerning the description of inlet boundary conditions.

The validation of CFD models by experimental measurements campaigns on full scale test room, such as those carried out in this study, allows supplementary research in the field of ventilation efficiency, thermal comfort, and indoor air quality in mechanically ventilated rooms. This is extremely important as the configuration taken into consideration was intended to be as close as possible to a real one.

Consequently, this study brings new elements of analysis in the field of CFD model validation through relevant experimental data, obtained for real configurations of mechanically ventilated rooms.

Acknowledgements

This work was supported by a grant of the Romanian Ministry of Education and Research, CNCS-UEFISCDI, project number PN-II-ID-JRP-RO-FR-2012-0071.

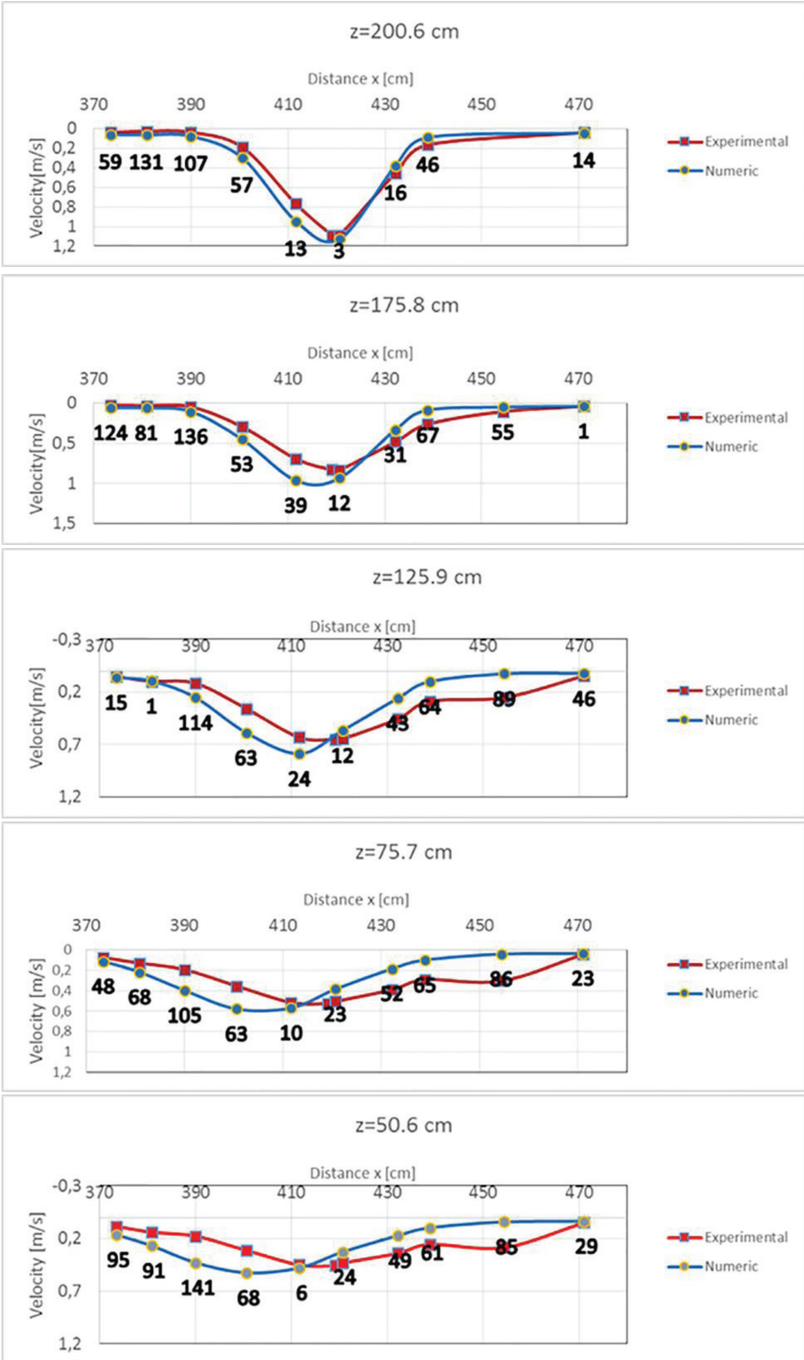


Fig. 6. Air velocity experimental-numerical comparisons in the jet region (the values in bold black represent the relative errors for each point)

References

[1] Nielsen PV. Fifty years of CFD for room air distribution. Building and Environment 2015; **91**:78-90, doi: 10.1016/j.buildenv.2015.02.035
[2] Nielsen PV, Liu L, Peng L, Li Y. The Status of Computational Fluid Dynamics. In: 12th REHVA World Congress – Clima 2016, Aalborg, Denmark, 2016.
[3] Holmberg S, Chen Q. Air flow and particle control with different ventilation systems in a classroom. Indoor Air 2003; 13(2):200-204, doi: 10.1034/j.1600-0668.2003.00186
[4] Kaji H, Akabayashi SI, Sakaguchi J. CFD analysis for detached house: Study on the ventilation efficiency on constantly ventilated house part 1. Journal of Environmental Engineering 2009; **74(636)**:161-168, doi: 10.3130/aije.74.161
[5] Mao N, Song MJ, Chan MY, Pan DM, Deng SM. Computational fluid dynamics (CFD) modelling of air flow field, mean age of air and CO_2 distributions inside a bedroom with different heights of conditioned air supply outlet. Applied Energy 2016; **164**:906-915, doi: 10.1016/j.apenergy.2015.10.096
[6] Yang L, Ye M, He BJ. CFD simulation research on residential indoor air quality. Science of the Total Environment 2014; **472**:1137-1144, doi: 10.1016/j.scitotenv.2013.11.118
[7] Zhuang R, Li X, Tu J. CFD study of the effects of furniture layout on indoor air quality under typical office ventilation schemes. Building Simulation 2014; **7(3)**:263-275, doi: 10.1007/s12273-013-0144-5
[8] Gilani S, Montazeri H, Blocken B. CFD simulation of stratified indoor environment in displacement ventilation: Validation and sensitivity analysis. Building and Environment 2016; **95**:299-313, doi: 10.1016/j.buildenv.2015.09.010
[9] Hajdukiewicz M, Geron M, Keane MM. Calibrated CFD simulation to evaluate thermal comfort in a highly-glazed naturally ventilated room. Building and Environment 2013; **70**:73-89, doi: 10.1016/j.buildenv.2013.08.020
[10] Nada SA, El-Batsh HM, Elattar HF, Ali NM. CFD investigation of airflow pattern, temperature distribution and thermal comfort of UFAD system for theater buildings applications. Journal of Building Engineering 2016; **6**:274-300, doi: 10.1016/j.jobe.2016.04.008
[11] Ahmed AQ, Gao SA, Kareem AK. A numerical study on the effects of exhaust locations on energy consumption and thermal environment in an office room served by displacement ventilation. Energy Conversion and Management 2016; **117**:74-85, doi: 10.1016/j.enconman.2016.03.004
[12] Hohotă R. Moisture modeling in a CFD code (low velocity in large cavity) – comparison with the experiment (in French), Ph. D. thesis, INSA Lyon, France; 2003
[13] Kuznik F. Experimental study of horizontal anisothermal axisymmetric jets issuing near a wall: Application to CFD modelling of ventilated enclosures (in French), Ph. D. thesis, INSA Lyon, France; 2005
[14] Găvan V. Full-scale experimental evaluation and modelling of a double-skin facade. Optimal control of thermal and visual comfort, Ph.D. thesis, INSA Lyon, France; 2009
[15] Castanet S. Contribution to the study of ventilation and indoor air quality (in French), Ph. D. thesis, INSA Lyon, France; 1998
[16] Laporthe S. Contribution to the qualification of ventilation system of buildings (in French), Ph. D. thesis, INSA Lyon, France; 2000
[17] Shih TH, Liou A, Shabbir A, Yang Z, Zhu J. A new k-ε Eddy-Viscosity Model for High Reynolds Number Turbulent Flows - Model Development and Validation. Computers Fluids 1995; **24(3)**:227-238, doi: 10.1016/0045-7930(94)00032-T
[18] Teodosiu C, Hohotă R, Rusaouën G, Woloszyn M. Numerical Prediction of Indoor Air Humidity and Its Effect on Indoor Environment. Building and Environment 2003; **38(5)**:655-664, doi: 10.1016/S0360-1323(02)00211-1