ARCHITECTURE CIVIL ENGINEERING ENVIRONMENT
The Silesian University of Technology
No. 3/2014

# THE IMPROVEMENT OF NUMERICAL MODELLING OF AIRFLOW IN VENTILATED ROOM

Piotr CIUMAN [a], Barbara LIPSKA [b*]

[a] MSc, PhD Student; Department of Heating, Ventilation and Dust Removal Technology; Silesian University of Technology, Gliwice, Poland

[b] Prof.; Department of Heating, Ventilation and Dust Removal Technology Silesian University of Technology, Gliwice, Poland

* E-mail address: *barbara.m.lipska@polsl.pl*

Received: 28.04.2014; Revised: 9.05.2014; Accepted: 10.07.2014

Abstract
**In the paper possibility of improvement of airflow modeling in ventilated room was checked. Standard perpendicular room was examined. Into it, through the inlet, isothermal jet was supplied. Numerical simulations were carried out with the use of ANSYS CFX software, based on CFD method. Calculations were executed for steady conditions, with the use of k-ε turbulence model. Results of experiments, conducted in room's physical model, in a region of supplying jet, with the benefit of laser doppler anemometer, were used for the validation of numerical simulations. Simulations were carried out for various variants of discretization grid. Cells size and their refinement were checked in terms of conformity to experiments results. For chosen best variant, effect of iteration process on jet flow's pattern was observed.**

Streszczenie
**W artykule sprawdzono możliwość poprawy dokładności modelowania przepływu powietrza w pomieszczeniu wentylowanym. Badano typowe prostopadłościenne pomieszczenie, do którego przez prostokątną kratkę nawiewano izotermiczną strugę powietrza. Obliczenia numeryczne wykonano za pomocą programu ANSYS CFX bazującego na metodzie CFD. Obliczenia przeprowadzono dla warunków ustalonych przy wykorzystaniu modelu turbulencji k-ε. Do walidacji prognoz numerycznych wykorzystano wyniki pomiarów przeprowadzonych w modelu fizykalnym obiektu, w rejonie strugi nawiewanej, za pomocą dopplerowskiego anemometru laserowego. Symulacje przeprowadzono dla różnych wariantów siatki dyskretyzacji, badając wpływ wymiarów i zagęszczenia oczek na zgodność wyników z pomiarami. Dla wybranego najlepszego wariantu obserwowano wpływ przebiegu procesu iteracji na obraz przepływu strugi.**

Keywords: **Ventilation; Air distribution; Supply jet; Numerical calculation CFD; Validation; Discretization grid; Iteration process.**

## 1. INTRODUCTION

The choice of air distribution concept in a room is one of the most important steps in designing ventilation and air conditioning. Assessment of ventilation and air condition by occupants depends in large part on distribution of air parameters in a room. In modern designing and research the choice of a proper air distribution concept is supported by numerical simulations with the use of Computational Fluid Dynamics (CFD) technic. To be sure that obtained results of numerical modelling can be used with conviction and trust in designing of rooms' ventilation, credibility of CFD software must be checked, which means validation of numerical calculations. According to its principles, which were formulated by [2], it assesses the ability of CFD code, coupled with its user's ability to simulate representative airflow variant in a ventilated room, for which experimental data is available. Thanks to validation it is possible to improve numerical calculations results, making use of various code options, including discretization grid, turbulence models, numerical solution schemes, convergence controls, etc.

Such studies were performed in detail for basic and complex objects in [6] with the use of engineering CFD codes (Vortex, Flovent). Effect of various computational parameters on numerical calculations precision for room with cross-ventilation was tested in [7]. Validation of numerical predictions for an auditorium with the use of mixing ventilation was carried out in [5]. Such a research for a simple room with displacement ventilation was presented in [4].

The aim of presented researches was to improve numerical predictions of airflow in a room with mixing ventilation with tangential flow, carried out using CFD code – Ansys CFX. It was obtained by validation of calculations results with the use of available experimental data for a region of a supply jet.

## 2. NUMERICAL METHOD

In presented study ANSYS CFX code [8] was applied, based on Computational Fluid Mechanic CFD. These are numerical methods used for solving models describing fluids flow combined with heat exchange. These models are based on equations which are mathematical notation of mass, momentum and energy conservation. Due to the fact that airflows are turbulent flows most of the times, solution is carried out for equations averaged in time. An equation, which is a result of applied turbulence model, is being added to the set. In presented researches it was the standard k-ε model. Equations set are being discretized with a Finite Volume Method (FVM), and then are solved for appropriately chosen discretization grid. In Ansys CFX unstructured grid is available, which consists of tetrahedral elements with possible appliance of hexahedral cells on boundary level. It is also possible to apply local refinement in chosen regions of modeled object, which was used in presented tests. An equation set is being completed with boundary conditions on walls and on boundary level, where standard wall functions are used and in inlet and outlet openings. The calculations were performed in steady conditions, isothermal. The solution of the model is carried by means of iteration method, whose convergence highly affects final results of calculations and therefore must be under control.

## 3. DESCRIPTION OF THE TEST ROOM AND MEASUREMENTS

Object used for tests was a rectangular prism room of height $H = 3$ m, width $A = 6$ m and length $L = 6$ m

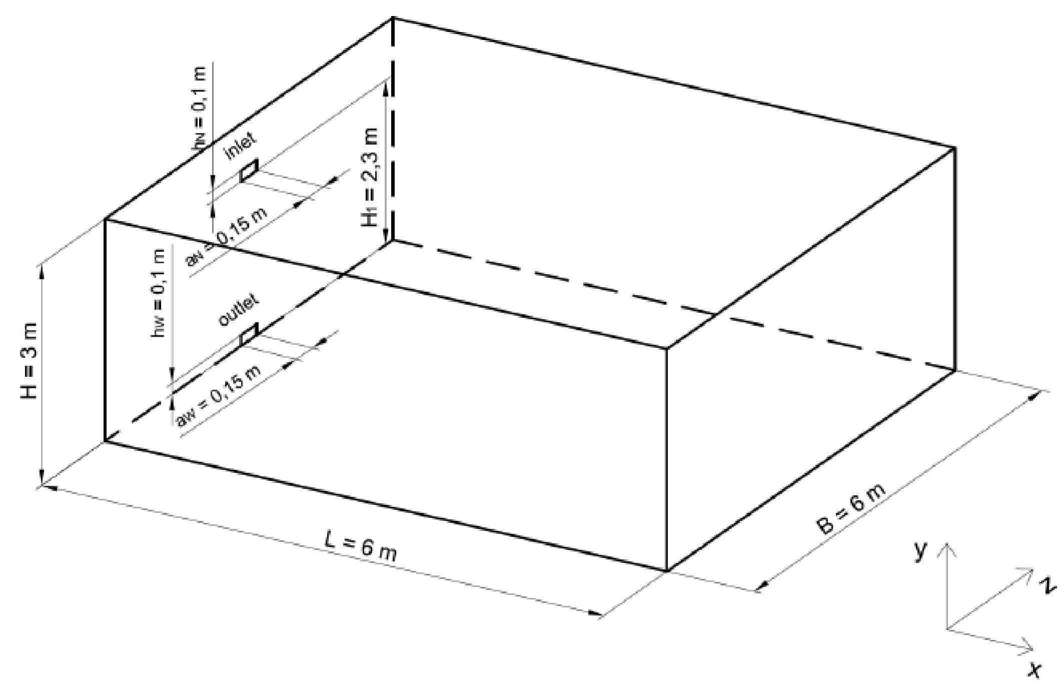


**Figure 1.**
**Scheme of the test room**

(Fig. 1). Inlet in shape of grill of height $h_N = 0.1$ m and width $a_N = 0.15$ m was placed in one of the side walls on height $H_1 = 2.3$ m above the floor. Through it an isothermal jet was supplied into the room with velocity $v_N = 5.2$ m/s. Outlet, of the same sizes, was placed below the diffuser, directly above the floor.

Choice of an object was dictated by the fact that it is a typical, simple example of mixing ventilation with tangential flow, which has application in objects of different use, e.g. auditoriums and other educational rooms, office rooms, ice rings, etc. Furthermore, at disposal were results of measurements carried out with the use of laser Doppler anemometer in the region of supply jet in room's physical model on a scale of 1:5 [1], which was used for experimental validation of numerical predictions. Similar researches for occupied zone, carried out with the use of spherical thermo-anemometer, were presented in [3].

## 4. EFFECT OF DISCRETIZATION GRID ON RESULTS OF CALCULATIONS

Important stage in improving results of numerical calculations was choosing a proper discretization grid. Six variants were tested, differing from each other by parameters shown in Fig. 2, which are: constant edge length, length scale of refinement cell, radius of refinement influence, constant edge length for the inlet face and appliance of inflated boundary layer. Grid refinement in diffuser region was carried along the axis going through the middle of supply opening along the length of the room.

These parameters are presented in dimensionless form, as well. It enables to expand applications of results to other, geometrically similar objects. Constant edge length and length scale of refinement cell were normalized by length of the room $L$. Radius of refinement influence and constant edge length for the inlet face were normalized by inlet's equivalent diameter $d_N = 0.033$ m, calculated according to the formula:

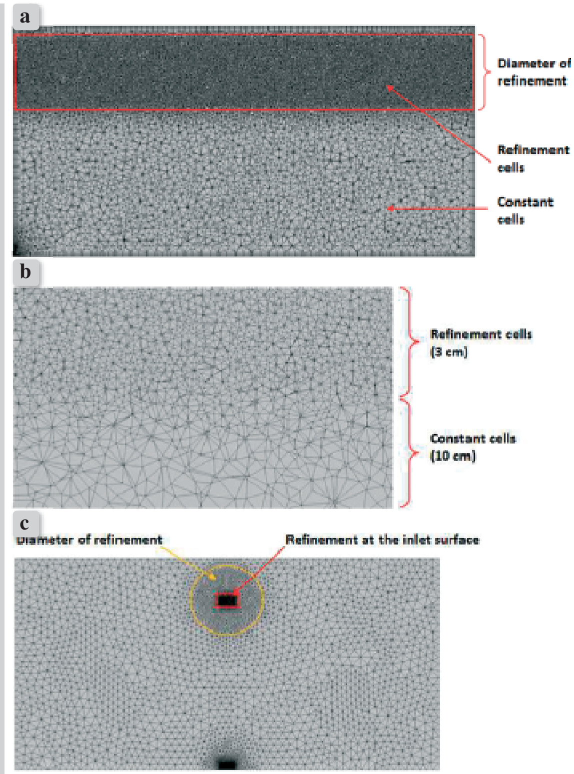


**Figure 2.**
**Discretization grid used in variant 3 a) section in plane Z = 3 m, b) fragment in longitudinal section, c) fragment in cross-section near the diffuser**

$$d_N = \frac{4F}{U}$$

where: $F$ – the surface area of the inlet,
$U$ – circuit of the inlet.

Values of length scale of refinement cell and constant edge length for the inlet face were not changed in tests. They were respectively 3 cm ($0.005L$) and 0.5 cm ($0.15\ d_N$).

Table 1 lists values of remaining parameters, as well as number of nodes and grid cells for simulation variants.

**Table 1.**
**Parameters of tested discretization grids**

| Variant | Constant edge length | | Radius of refinement influence | | Inflated boundary layer | Total number of nodes | Total number of cells |
|---|---|---|---|---|---|---|---|
| | m | Normalized by $L$ | m | Normalized by $d_N$ | | | |
| 1 | 0.3 | 0.05 | - | - | - | 7037 | 35786 |
| 2 | 0.1 | 0.017 | 0.1 | 3 | + | 266268 | 1303002 |
| 3 | 0.1 | 0.017 | 0.4 | 12 | + | 511070 | 2666620 |
| 4 | 0.12 | 0.02 | 0.1 | 3 | + | 305849 | 1503643 |
| 5 | 0.1 | 0.017 | 0.2-0.4 (beginning-end of axis) | 6-12 (beginning-end of axis) | + | 656486 | 3373953 |
| 6 | 0.1 | 0.017 | 0.2 | 6 | + | 576585 | 2923246 |

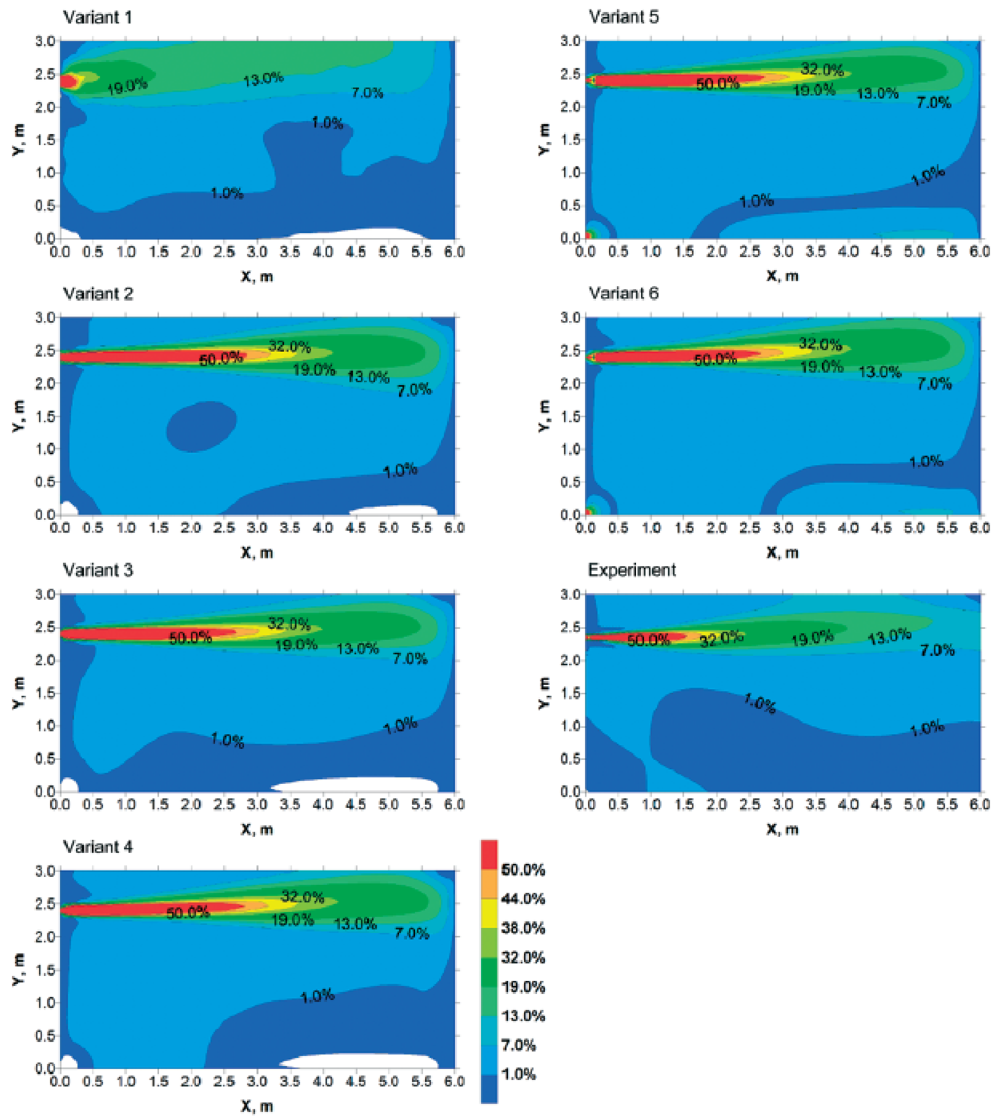


**Figure 3.**
**Maps of the axial averaged velocity component, normalized by supply air velocity, from the CFD calculation for various discretization grids and experiment, in the vertical longitudinal plane Z = 3 m**

Fig. 3 presents results of the simulation for every variant. Maps of the axial averaged velocity component normalized by supply air velocity were compared with the map obtained from the experiments. Following parameters were validated: range of the value of axial velocity component, jet throw, width of jet and its symmetry, deflection and penetration length. On this basis it was stated that greatest compliance of simulation results to experiment was obtained for variant 3. This mesh was the most appropriate for numerical predictions in this case. It was a compromise between computational time and calculations precision. Further tests presented in this article, as well as in [3] were carried out with the use of this grid.

The range of axial averaged velocity component contour lines, for which maps were prepared on the basis of calculations and measurements, was the same. However, it is important to notice that regardless of applied grid, predicted jet was wider, and particular contour lines were more lengthened than results of experiment. It didn't apply to contour line 7%, which didn't reach opposite wall, as it happened in the real room. However, in this case, it can be supposed that it was an effect of necessity of approximation of measurement results in this region, because test grid ended 0.25 m before walls.

## 5. EFFECT OF CONVERGENCE OF ITERATION PROCESS ON CALCULATIONS RESULTS

A significant factor, which decides about correctness and precision of obtained numerical solution, is convergence of iteration process. It means achieving by a residual expected target value.

The profile of residuals, presented for tested variant in the Fig. 4, didn't stabilize fully for the most of controlled dependent variables, but oscillated at low level of convergence between $10^{-5}$ and $10^{-6}$. To check how this effected results of calculations, in significant places of the tested room monitor points were established (at the beginning and the end of jet axis). The course of changeability of dependent variables was observed by means of: averaged velocity components, kinetic energy of turbulence, dissipation rate of kinetic energy of turbulence during the iteration process (Fig. 5). On this basis, it was found that values of those variables stabilized after all.

To make sure that it concerns the whole room, it was checked how the image of airflow was changing during iteration process. Therefore, Fig. 6 presents maps of normalized axial averaged velocity component at vertical longitudinal section for chosen iteration steps (marked in Fig. 4). These maps are presented in two-color version. Based on the distribution in Fig. 3 contour line 7% was established as a conventional jet border. It divides modelled zone into two parts: of velocities higher than it (red color) and lower (blue color). Maps were recorded first at intervals of 50 iteration steps, then of 100, and after that of 300 steps. In effect, it was possible to observe how predicted airflow was forming in the room along the iteration process and if it stabilized during further iteration steps. Thanks to that, it was possible to present influence of supply jet on air masses in tested object. After 200 iterations a jet which encompassed with its range whole length of the room was formed. Airflow relatively stabilized. Jet was stack to the ceiling and only in this region were observed small changes of its shape, as an effect of a change of penetration length. This change probably occurred because of Coanda effect influence. Beginning from 800 step, continuation of the iteration process didn't improve numerical solution.

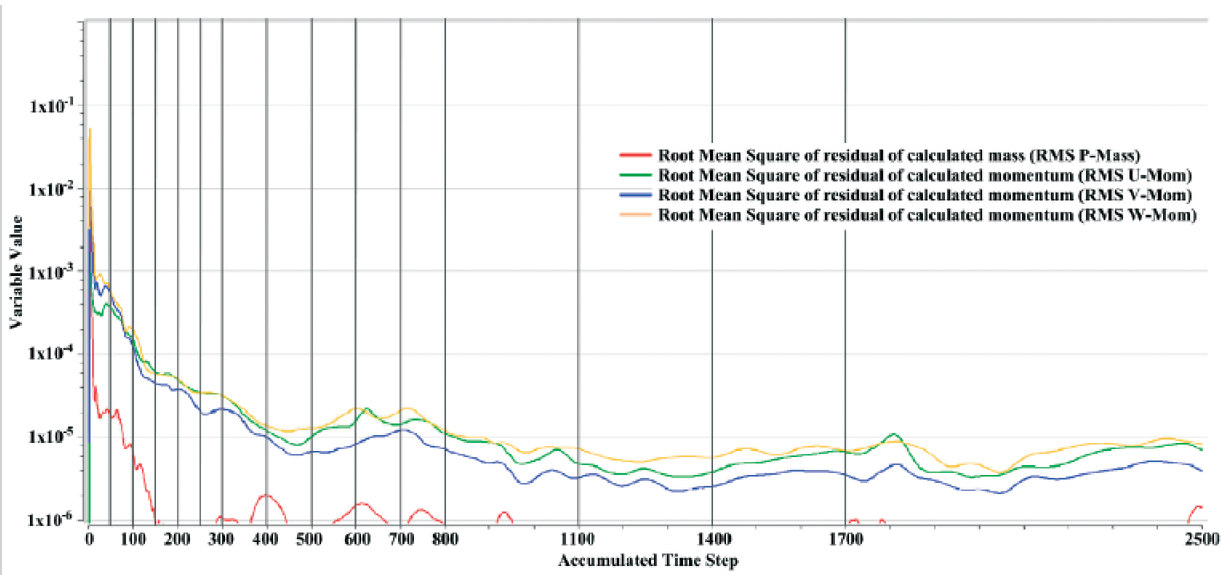


**Figure 4.**
**Profile of residuals iteration process for chosen dependent variables**

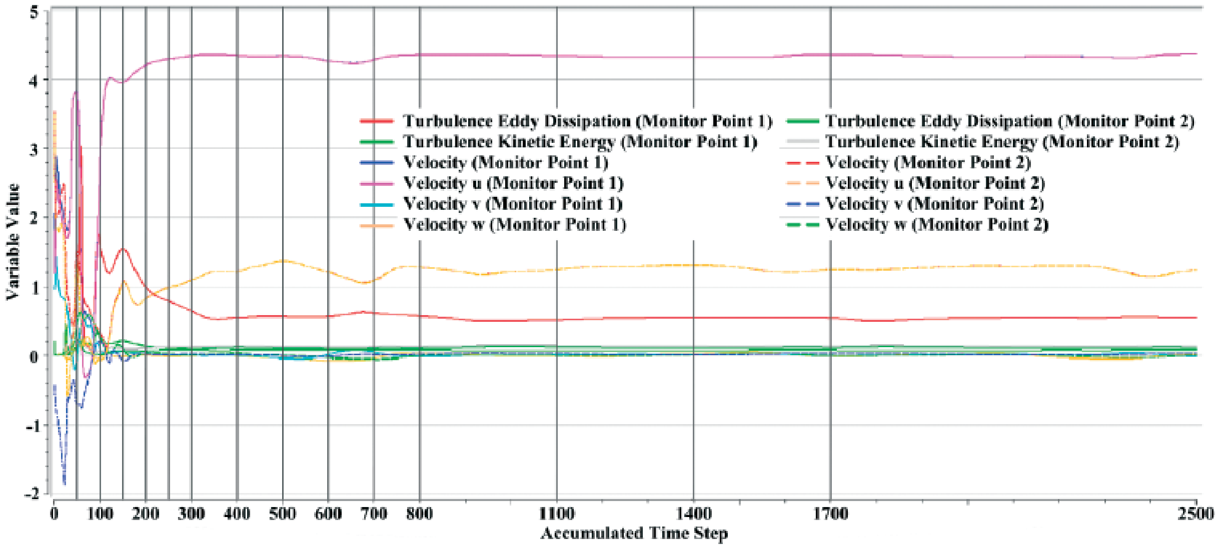


**Figure 5.**
**Profile of dependent variables values in two monitor points during the iteration process**

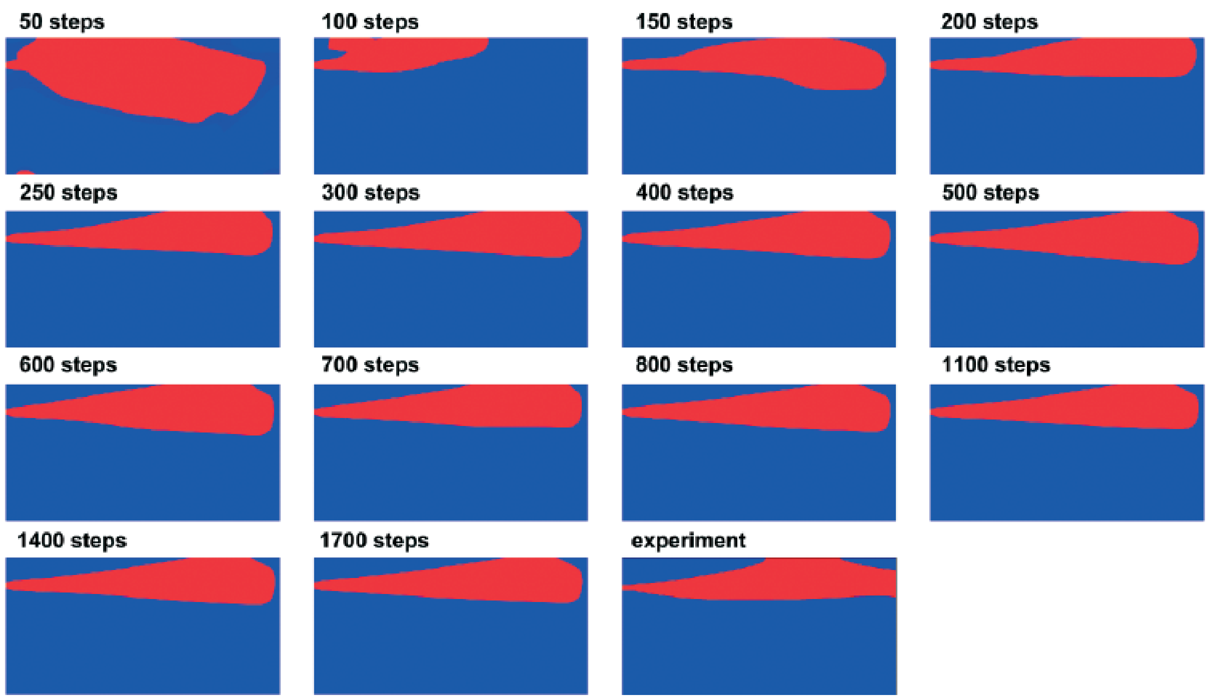


**Figure 6.**
**Two-color maps of normalized axial averaged velocity component from calculations for chosen iteration steps and experiment**

## 6. CONCLUSIONS

1. To carry out numerical calculations for analyzed case of supply jet the best discretization grid was the one of constant cell length 10 cm ($0.017L$) and length scale of refinement cell 3 cm ($0.05L$), radius of refinement influence 40 cm ($3d_N$), constant edge length for the inlet face 0.5 cm ($0.15d_N$), with inflated boundary layer.
2. Predicted and measured ranges of axial averaged velocity component in the jet are very similar.
3. It wasn't possible to obtain similarity of predicted and measured airflow image at the end of jet zone, near the opposite to diffuser wall, which could be an effect of approximation of measurement results in this zone.
4. Lack of stabilization of residuals profile with their oscillation at low level wasn't a sign of lack of proper solution. However, this assessment had to be backed up by controlling stabilization of dependent variables in chosen, characteristic points of modeled room.
5. In tested case, beginning with 800 iteration step stabilization of predicted pattern of supply jet was observed.
6. Small changes of this pattern in a zone where jet was stack to the ceiling could be a result of Coanda effect influence. Although, to make sure, it would be necessary to perform calculations of airflow for the same case in transient state.

## REFERENCES

[1] *Blaszczok M.*; Badanie właściwości ruchu powietrza w pomieszczeniach z wentylacją mieszającą [Analysis of air motion characteristic in rooms with mixing ventilation]. Praca doktorska. Politechnika Śląska, Gliwice 2006 (in Polish)

[2] *Chen Q., Srebric J.*; A procedure for verification, validation and reporting of indoor environment CFD analyses. International Journal of HVAC&R Research 8(2)/2002, p.201-216.

[3] *Ciuman P.*; Wpływ sposobu modelowania turbulencji na prognozowanie numeryczne strug nawiewanych [The effect of turbulence modelling on numerical prediction of supply air jets]. Praca dyplomowa magisterska pod kierunkiem Lipskiej B., Politechnika Śląska, Gliwice 2013 (in Polish)

[4] *Deevy M. et al.*; Modelling the effect of an occupant on displacement ventilation with computational fluid dynamics. Energy and Buildings Vol.40/2008, Issue 3, p.255-264

[5] *Koper P.*; Modelowanie rozdziału powietrza wentylacyjnego w salach audytoryjnych [Modelling of airflow in a lecture hall]. Praca doktorska. Politechnika Śląska, Gliwice 2013 (in Polish)

[6] *Lipska B.*; Kontrola jakości numerycznego modelowania przepływu powietrza w pomieszczeniach wentylowanych [Quality control of numerical modelling of airflow in ventilated rooms]. Monografia. Zeszyty Naukowe Politechniki Śląskiej nr 1718, Gliwice 2006 (in Polish)

[7] *Ramponi R., Blocken B.*; CFD simulation of cross-ventilation for a generic isolated building: Impact of computational parameters. Building and Environment 53/2012 p.34-48

[8] www.ansys.com