CFD modeling of a self-oscillatory airflow regime in the test ventilated room with plane supply opening

Nikolay Ivanov, Marina Zasimova, Ekaterina Stepasheva, and Vasilisa Kudryavtseva
Peter the Great St. Petersburg Polytechnic University, 195251, 29 Politechnicheskaya Str., St. Petersburg, Russian Federation

Abstract. The current contribution presents the results of numerical simulation of the airflow and heat transfer in a model room of rectangular shape ventilated with a single plane jet under the conditions of the experiment by Mataoui et al. (2001). The 2D and 3D unsteady Reynolds-averaged Navier-Stokes approaches were used. Numerical solutions were obtained with the CFD package ANSYS Fluent based on the finite volume method with the cell-centered variable arrangement. The position of the supply nozzle corresponds to the self-oscillatory experimental conditions. The results of the computational data validation are reported. Effects of the low-frequency velocity oscillations on the heat transfer parameters are discussed.

1 Introduction

The indoor air exchange parameters and the level of comfort depend on the airflow structure in a ventilated room. Supply air jets form the airflow pattern determined by many factors, e.g. supply diffuser location, air jet direction, intake air velocity and turbulence intensity distributions, etc. It is possible to study the effects of these factors by means of Computational Fluid Dynamics (CFD) modeling of airflow in simplified configurations. Moreover, numerical data obtained for model problems allow getting understanding of possible control mechanisms over the flow regime.

Under some conditions, low-frequency self-oscillating flow regimes develop in a ventilated indoor space. Low-frequency velocity fluctuations emerging due to airflow instability may lead to strong time variation of the draught rate that corresponds to possible uncertainty in the percentage of dissatisfied persons [1]. However, until now, designers do not take into account possible effects of these oscillations on ventilation airflows and thermal comfort parameters, solving most of the applied problems in a stationary formulation.

The current contribution considers one of the model problems with self-oscillating flow regime, namely the airflow in a room of rectangular shape ventilated with a single plane jet. The problem formulation of the present CFD study corresponds to the conditions of the experiment by Mataoui et al. [2], where the velocity measurements with constant temperature hot wire anemometry were performed for a wide range of problem parameters. The experimental studies [2], reported also in [3], showed that periodic oscillations might occur in the model configuration considered, depending mostly on the location of the jet exit nozzle inside the cavity. The experimental study identified the steady and unsteady regimes and presented a map of the flow regimes [2, 3].

The test configuration by Mataoui et al. has been already used for numerical data validation, both by the same research group (see [3-7]) and by other research groups [8, 9]. Note that most papers presented results obtained with 2D unsteady Reynolds-averaged Navier-Stokes (URANS) approach. In [8] the 3D results of large eddy simulation (LES) computations were presented and discussed; relatively coarse mesh allowable at that moment was used.

The focus of the current study is the analysis of the low-frequency high-amplitude self-oscillatory airflow regime with respect to the flow pattern and heat transfer characteristics formed in the test ventilated room under such conditions. The results are obtained with URANS approach and form a background for the further vortex-resolving LES computations for the same regime.

2 Problem formulation

The model ventilated room (or the cavity) considered is a rectangular-shaped domain with the dimensions: length of \( X_0 = 0.5 \) m, width of \( W_0 = 0.2 \) m, height of \( H_0 = 0.2 \) m. Air is supplied through a plane duct with the contracted nozzle, the duct has the same width as the room. The nozzle exhaust section has the width-to-height aspect ratio equal to 20, its height is \( h_0 = 0.01 \) m. The computations were performed in 2D and 3D formulations.

Fig. 1 illustrates the geometry of the computational domain (for 3D formulation). The airflow through the duct is not considered, its interior is not included into the computational domain, and the inlet boundary condition is set at the nozzle exhaust section surface. The uniform velocity and temperature approximation at the inlet is assumed, \( V_{in} = 6 \) m/s, \( T_{in} = 300 \) K.

The calculations were performed for the isothermal case corresponding to the experimental test conditions and for the non-isothermal case with the hot constant-temperature model room (cavity) walls, \( T_e = 310 \) K.
Based on the preliminary auxiliary computations, an additional volume was attached to the cavity domain. According to the description of the experimental setup [2, 3], the cavity is open to the ambient air from one side that allows air to exit freely. To simulate this, the boundaries of the additional volume were set at the distance of $X_0$ from the cavity in the upward and downward directions, and at the distance of $1.5X_0$ from the cavity in the longitudinal $x$-direction. The outlet boundary condition was set at the vertical boundary only; other boundaries of the additional volume were treated as solid walls with the slip boundary condition (to avoid requirements on the near-wall resolution in case of the no-slip boundary condition).

The air is considered as an incompressible fluid. The Reynolds number computed with the nozzle exhaust section height as the length scale and the mean supply velocity is equal to $Re = 4 \times 10^3$.

For the non-isothermal case, the buoyancy effect is taken into account using the Boussinesq approximation. The Grashof number computed with the cavity height $H_0$ as the length scale is equal to $Gr = 1.2 \times 10^7$. Accordingly, the Richardson number is equal to $Ri = Gr/Re^2 = 0.75$ that corresponds to the mixed convection conditions.

The unsteady Reynolds-averaged Navier-Stokes equations were solved (URANS approach). Two turbulence models were used: standard $k-\varepsilon$ and $k-\omega$ SST. The turbulence intensity, $I = 14\%$, and the turbulent viscosity ratio, $\mu_t/\mu = 6.2$, were set at the inlet boundary. Numerical solutions were obtained with the CFD package ANSYS Fluent based on the finite volume method with the cell-centered variable arrangement. The time step value of 0.01 s was used in all the cases considered. The second order temporal and spatial discretization schemes available in the solver were activated. 3D computations were performed with the usage of the resources of Super Computer Center (SCC) «Polytechnic» (www.scc.spbstu.ru).

The computational meshes were generated with the ANSYS ICEM CFD software. Meshes consisted of hexahedral cells and were refined near the solid wall surfaces. For 2D cases, the total amount of mesh elements is in the range of 10...60 thousand cells. Fig. 2 presents a view of the 2D mesh with 60 thousand cells and the corresponding instantaneous dimensionless distance from the first near-wall cell to the wall $y^+$ distributions over the cavity walls. The $y^+$ values are less than 1 in the case of the finest mesh. For 3D cases, two meshes were used, the initial one of 2.3 million cells and the refined one of 5 million cells. The latter also provides the $y^+$ values less than 1. This 3D mesh has the same topology in a vertical section as shown in Fig. 2; it was obtained by means of translation of a correspondent 2D mesh in the transversal direction.

3 Results and discussion

3.1 Airflow pattern

The choice of the position of the exhaust nozzle section in the rectangular-shaped ventilated room made in the current study resulted in stable quasi-periodic jet oscillations. Thus, the computations reproduce an unsteady self-oscillatory airflow regime that is in full accordance with experimental data [2]. Fig. 1b shows an instantaneous velocity magnitude isosurface with the jet directed towards the bottom wall of the model room. During one period of oscillations the room is ventilated effectively. Large-scale oscillations result
in the periodic jet propagation towards the upper and lower cavity walls, from time to time the jet penetrates also to the side wall opposite to the inlet nozzle section, and this periodic process does not allow to form any pronounced stagnant zone (supposed to be formed in case of a steady-state airflow). The instantaneous airflow structures obtained in the current CFD results are in agreement with the experimental observations reported in [2, 3].

Even an instantaneous flow pattern presented in Fig. 1b demonstrates that 3D effects are not strong: the jet remains 2D shape at a long distance from the nozzle. To evaluate 3D effects, for the time-averaged solution obtained with 3D formulation an additional spatial averaging over the transversal direction (z-direction) was performed. The resulting averaged field was compared with the 2D time-averaged field. The comparison showed that mean fields are almost the same. These estimations justify the usage of 2D computational and measured data at monitoring points considered, at least on the stage of the flow field evaluation.

The comparison between the CFD and experimental data could be performed using the time-evolution of velocity magnitude; these data at monitoring points are available in [2]. Fig. 3 illustrates the results of 2D URANS computations obtained with two semi-empirical turbulence models – standard k-ε and k-ω SST (these models are widely used in the engineering applications). The velocity magnitude time-evolution curves are given for two monitoring points. Positions of these points are presented in Fig. 2: point P1 is placed at the jet axis, while point P2 is shifted toward the bottom wall of the cavity.

![Fig. 3. Velocity evolution obtained in 2D URANS calculations (with standard k-\(\varepsilon\) and k-\(\omega\) SST models) and in experiment [2] at two monitoring points a) P2 and b) P1.](image)

The data computed at monitoring point P1 demonstrate a good agreement with the experimental data: both the period and the amplitude of pulsations almost coincide in the measurements and computations. It could be concluded that the effect of the turbulence model choice is negligible. The agreement between the computational and measured data at monitoring point P2 is worse. In general, the computations reproduce the period of pulsations (Fig. 3a), but the shape of the curve and the amplitude of pulsations are different. High amplitude of pulsations predicted in calculations, with the maximum velocity values of about 4 m/s, corresponds to the instants when the jet core achieves the location of the point P1 (the maximum is compared with the maximum at point P1, see Fig 3b). It corresponds to the animation of the unsteady velocity field evolution in time. The same results were obtained in 3D computations. However, the measurement data do not detect these peak values.

Table 1 shows the period of oscillations, \(t_p\), predicted with 2D simulation. The data are given for two isothermal cases with different turbulence models and for the non-isothermal case with relatively cold supply jet air \((T_w = 300 \text{ K})\) and relatively hot cavity walls \((T_u = 310 \text{ K})\). The measured oscillations period is \(t_p = 1.56 \text{ s}\) [2]. The relative difference between the computed and experimental period, \(\delta\), are given in the last column of the table. The difference between the computed and measured period of oscillations is less than 8% for all the cases considered. Remarkably, that the use of a non-isothermal problem formulation with relatively small temperature difference between the supplied air and the cavity walls resulted in the reduced difference that is less than 1%. Note that in [2] there is no information about possible uncertainty due to non-isothermal effects in the airflow measurements.

![Table 1. Period of oscillations for 2D URANS cases.](image)

In general, the computational results demonstrate a satisfactory agreement with the measured data on the local airflow characteristics in the room (global airflow structure, period of oscillations, instantaneous velocity field evolution in time).

3.2 Heat transfer parameters

The time-dependent airflow with strong low-frequency self-oscillations result in better mixing in the ventilated space as compared with the steady-state regime. Global unsteadiness could increase in the efficiency of cooling or pollutant removal. In particular, unsteady effects could enhance heat transfer from the walls. As an initial study of this phenomenon, the case with low temperature difference between the room walls and the supplied jet, \(\Delta T = 10K\), was considered.

Fig. 4a illustrates time-evolution of the heat flux values area-averaged over each wall of the ventilated room for the non-isothermal case considered. An instantaneous temperature distribution is presented in Fig. 4b. At the particular time instant considered, the jet
turns towards the upper wall that is visible in the temperature field (white colour). This instant corresponds to a maximum value of the black curve in Fig. 4a. The instantaneous distributions of the local values of the specific heat flux \( q_w \) along the room walls are also illustrated in Fig. 4b.

As the cool jet is directed towards the upper wall, the maximum of heat flux is observed there: locally \( q_w \) values achieve 550 W/m\(^2\). The correspondent instantaneous mean value area-averaged over the upper wall is about 250 W/m\(^2\). It corresponds to the Nusselt number value, \( Nu = q_w h_0/\lambda \Delta T \), of about 10.5.

For the bottom wall, the peak heat flux value averaged over the bottom wall of the room is detected when the jet turns in the opposite direction as compared with Fig. 4b. Due to the buoyancy effects, the peak instantaneous mean heat flux value at this wall is slightly less than at the upper wall; it is equal to about 230 W/m\(^2\), (see the light-gray curve in Fig. 4a). The corresponding \( Nu \) number value is 9.7. At both the walls, the minimum heat flux values are detected when the cooled jet is directed towards the opposite wall; the area-averaged minimum heat flux values are the same for both the horizontal walls, equal to 110 W/m\(^2\) (\( Nu = 4.5 \)). Accordingly, the temporal non-uniformity of the area-averaged heat flux could achieve 100% of the time-averaged value: the maximum is two times higher than the minimum.

For the frontal sidewall (opposite to the nozzle) the heat flux values averaged through the wall surface are less than 100 W/m\(^2\), much less than for the horizontal walls (see the dark-gray curve in Fig. 4a). Sure, the heat transfer characteristics at this wall depend on the distance from the nozzle.

![Fig. 4. a) Area-averaged heat flux time evolution for three walls of the ventilated room; b) an instantaneous temperature field, for the same time instant the solid lines show the instantaneous heat flux distributions along three walls.](https://rscf.ru/en/project/22-29-00224/)

In general, the computational results presented in Fig. 4 demonstrate strong influence of the global airflow unsteadiness on the local and mean heat transfer characteristics at the room walls. The detailed study of the unsteady effects on the heat and mass transfer in the ventilated room is considered as the future work.

### 4 Conclusions

The airflow and heat transfer in a model room of rectangular shape ventilated with a single plane jet was computed using the Unsteady Reynolds-Averaged Navier-Stokes approach. The position of the air supply nozzle considered correspond to the self-oscillatory experimental conditions.

For the experimental case considered the computations reproduced well the period and amplitude of the pulsations with the 2D formulation. It was shown that 3D effects were weak, and the use of 2D formulation was justified, at least on the stage of the flow field evaluation. In general, the computational results demonstrated a satisfactory agreement with the measured data on the local airflow characteristics.

For the non-isothermal case with the hot constant-temperature room walls, time evolution of the heat transfer characteristics was evaluated. The computational data demonstrate strong influence of the global airflow unsteadiness on the local and mean heat transfer characteristics. It was found that the temporal non-uniformity of the area-averaged heat flux could achieve 100% of the time-averaged value. The further work will cover a detailed study of the unsteady effects influence on the heat and mass transfer in the test ventilated room using the vortex-resolving LES approach.

The study was supported by the Russian Science Foundation, grant no. 22-29-00224, information about the project is available at [https://rscf.ru/en/project/22-29-00224/](https://rscf.ru/en/project/22-29-00224/).

### References

1. D. Markov, N. Ivanov, G. Pichurov, M. Zasimova, P. Stankov, E. Smirnov, I. Simova, V. Ris, R.A. Angelova, R. Velichkova, Applied Sciences 10, 5036 (2020)
2. A. Mataoui A., R. Schiestel, A. Salem, Flow, Turbulence and Combustion 14, 267 (2001).
3. A. Mataoui, R. Schiestel, A. Salem, Applied Mathematical Modelling 27, 89 (2003).
4. A. Mataoui, R. Schiestel, J. of Fluids and Structures 25, 60 (2009).
5. F. Iachachene, Y. Halouane, A. Mataoui, Thermal Science 20, 1485 (2016).
6. F. Iachachene, Y. Halouane, A. Mataoui, J. Heat Transfer 137, 101702-1 (2015)
7. F. Iachachene, Y. Halouane, A. Mataoui, Progress in Computational Fluid Dynamics 16, 179 (2016)
8. D.M. Denisikhina, I.A. Bassina, D.A. Nikulin, M.Kh. Strelets, High Temperature 43, 568 (2005)
9. M. Aminzadeh, J. Khadem, S.A. Zolfaghari, A. Omidvar, Int. Communications in Heat and Mass Transfer 129, 105680 (2021)