Mean air velocity correction for thermal comfort calculation: assessment of velocity-to-speed conversion procedures using Large Eddy Simulation data

Nikolay G Ivanov and Marina A Zasimova

Department of Fluid Dynamics, Combustion and Heat Transfer, Peter the Great St.Petersburg Polytechnic University, Polytechnicheskaya str. 29, St.Petersburg, 195251 Russia

E-mail: zasimova_ma@spbstu.ru

Abstract. The paper presents Large Eddy Simulation (LES) data for the ventilation airflow in an isothermal room corresponding to the Nielsen et al. (1978) test. Based on a simplified formulation with the periodicity boundary conditions in the transverse direction, detailed grid sensitivity study was performed. Using LES data on velocity components and mean speed extracted from a periodic domain-size-independent solution, the paper assesses velocity-to-speed conversion procedures available in the literature for the Reynolds-Averaged Navier-Stokes ventilation airflow data processing.

1. Introduction
Computational Fluid Dynamics (CFD) technologies based on the Reynolds-Averaged Navier-Stokes (RANS) equations solution are widely used in the indoor heating, ventilation and air conditioning (HVAC) systems design [1]. RANS computations provide information on mean components of velocity, \(<V_x>\), \(<V_y>\) and \(<V_z>\), that define the mean velocity magnitude, or simply “velocity”, \(V_m \equiv (\langle V^2_x \rangle + \langle V^2_y \rangle + \langle V^2_z \rangle)^{0.5}\). However, to evaluate thermal comfort and draught risk indices, mean speed distributions, \(V_a \equiv \langle (V^2_x + V^2_y + V^2_z)^{0.5} \rangle\), should be used instead of mean velocity magnitude. The values of \(V_a\) usually differ significantly from the \(V_m\) values [2].

To use RANS data in a ventilation system design, a proper conversion of the \(V_m\) data into \(V_a\) fields is necessary. Empirical correlations derived from ultrasonic anemometer measurements were suggested for mean speed evaluation in [2]. Correlations for RANS velocity processing based on the LDA measurements were proposed later in [3]. Contrary to RANS, Large Eddy Simulation (LES) approach that solves the filtered Navier-Stokes equations resolving large scales of motion [4] provides both the velocity and the speed data directly, and it is possible to use the LES data for \(V_m\) data processing procedure development and testing. It has been already done in [5] where a theoretically developed procedure of \(V_m\) processing has been examined using both the LES and the cabin ventilation qualification test data for the International Space Station pressurized module. Recent contribution [6] presented LES data on mixing ventilation in a test isothermal room with a sidewall jet and discussed the difference between \(V_m\)- and \(V_a\)-values in the jet zone.

The present paper considers geometry of the most popular isothermal benchmark test published by Nielsen et al. [7]. The test data have been used in more than 50 papers for CFD validation during the last two decades. Mean velocity components and fluctuations were measured with the Laser-Doppler
Anemometry (LDA); they are available in [7] and in full on the website http://www.cfd-benchmarks.com/. LDA measurements provide \( V_m \) data that could be directly compared with RANS solutions. Distributions of mean components of velocity (that form \( V_m \)) extracted from LES solutions [8, 9] were compared with the measured data as well. However, it is interesting to compute and compare \( V_{\text{in}} \) and \( V_s \)-fields for the Nielsen configuration with pronounced division into the high-velocity jet zone and the low-velocity zone of induced secondary flows (occupied zone), and it is the main objective of the current LES study. On the other hand, Nielsen test with simple geometry is a good object for the grid and other numerical parameters sensitivity analysis. Therefore, the objective of the paper is also to illustrate the resolution requirements for wall-modeled LES and to prove that the computations are accurate. However, to operate with reasonable computational resources, a simplified formulation with the periodicity boundary conditions in the transverse direction was used for analysis instead of the sidewalls installed in the experimental rig. This formulation resulted in a special study of the computational domain width on the periodic solution that is presented in the paper as well.

2. Problem formulation and computational settings

The computational domain shown in figure 1a is a rectangular room with the inlet slot under the ceiling and the outlet slot on the opposite wall placed just above the floor; an outlet channel was included into the computational domain. The dimensions of the room and ventilation openings are \( H = 3 \text{ m}, L/\text{H} = 3, h_{\text{in}}/\text{H} = 0.056, h_{\text{out}}/\text{H} = 0.16, L_{\text{out}}/\text{H} = 0.5 \). The origin of the coordinate system is in the corner of the room, and the surface \( z = 0.0 \) is one of the periodic boundaries. The opposite periodic boundary is located at \( z = W \). Three cases were considered: \( W/\text{H} = 1/6, 1/3 \) and 1; note that [9] presents the computational data for \( W/\text{H} = 1 \) with sidewalls corresponding to experiment [7].

Dashed lines in figure 1a illustrate positions of profiles used for postprocessing: vertical lines A-A located at \( x/\text{H} = 1.0 \) and B-B at \( x/\text{H} = 2.0 \); horizontal lines C-C located at \( y/\text{H} = 0.972 \) (starting at the mid-height of the inlet slot) and D-D located at \( y/\text{H} = 0.028 \) (that is equal to \( h_{\text{in}}/2 \) from the floor).

![Figure 1](image-url)  

**Figure 1.** (a) Computational domain (\( W/\text{H} = 1/6 \)); mean velocity magnitude distribution at mid-section \( z/\text{H} = 0.25 \) is shown; (b-d) SGS to molecular viscosity ratio (b) at mid-section \( z/\text{H} = 0.25 \), and at cross-sections (c) \( y/\text{H} = 1.0 \) and (d) \( y/\text{H} = 2.0 \).

Air was assumed as an incompressible fluid with constant physical properties \( (\rho = 1.225 \text{ kg/m}^3, \mu = 1.8 \times 10^{-5} \text{ kg/m/s}) \). The bulk velocity at the inlet was \( V_{\text{in}} = 0.455 \text{ m/s} \) (the Reynolds number \( Re = \rho h_{\text{in}} V_{\text{in}}/\mu = 5233 \)). Velocity profiles extracted from time-averaged solutions of auxiliary problems for long straight ducts with suitable \( W/\text{H} \) were set as the boundary condition at the inlet slot section.

Calculations were carried out with the ANSYS Fluent 16.2 software. To model small scales of turbulent motion, the algebraic Wall Modeled LES (WMLES) S-Omega subgrid-scale (SGS) model based on [10] was used. To compute the fluctuating velocity at the inlet slot, the vortex method available in ANSYS Fluent was used.

The non-iterative time advancement scheme (NITA) based on the fractional step method was used. The spatial discretization was performed with the central-differencing scheme for convective terms and the second-order central scheme for viscous terms, the second-order pressure interpolation was used. The uniform meshes created with the ANSYS ICEM CFD 16.2 mesh generator consisted of up
to 58 million cubic cells. For the coarsest mesh the maximum $y^+$-values (dimensionless wall distance) did not exceed 0.5 in the occupied zone and were up to 20 in the jet zone near the inlet slot.

The second-order implicit time integration was used. The value of a time step, $\Delta t$, is equal to 0.006 s, and it was chosen to provide the Courant number in the computational domain less than 1 for all the meshes considered. To accumulate representative statistics, it was required to calculate samples of about 1500 s (250,000 time steps). The calculations were performed using resources of the SPbPU supercomputer (see.spbstu.ru), and up to 512 cores were used in total.

3. Mesh-sensitivity analysis

The mesh-sensitivity analysis was performed for the case with the width of the computational domain $W/H = 1/6$. The initial (coarse) mesh consisted of $536 \times 179 \times 30$ uniform cubic cells (total mesh size of about three million cells). The cell length in this case is equal to $\Delta x = 16.8$ mm. The initial mesh was refined three times with the total cell number for each successively refined mesh increased by a factor of $8^{0.5}$ (i.e., $2^{0.5}$ times in each direction), so that mesh #2 had 8, #3 – 23, and #4 – 58 million cells.

Figure 2a,b shows the mesh refinement influence on skin friction coefficient over the top and bottom walls (for the bottom wall $C_f$ distribution over the exit duct at $x > 9$ m is also shown, figure 2b). The strongest mesh dependence of $C_f$ was detected at the top wall near the inlet, at $x < 3$ m, where $y^+$-values were the highest, and the wall modeling effects were pronounced. Downstream, at larger $x$, mesh dependence becomes weaker, and the solutions obtained with meshes #3 and #4 almost coincide at $x > 6$ m. For the bottom wall, skin friction does not change with the transition from mesh #3 to mesh #4 over the entire surface. The same conclusion about negligible difference between two solutions everywhere except the initial region of the jet zone (top part of section A-A) could be drawn from figure 2c where the longitudinal velocity profiles are compared.

The summary is that the solution obtained with mesh #3 ($\Delta = 8.4$ mm) could be treated as mesh independent solution. The deviation from the mesh independent solution seems to be critical for mesh #1, for both $C_f$ and velocity. On the contrary, accuracy of the solution obtained with mesh #2 ($\Delta = 12$ mm; $751 \times 252 \times 42$ cubic cells) is acceptable, and the meshes with this cell size were used for computations at other $W/H$ values, both for periodic problems and for the problem with sidewalls [9].

Note that if mesh #2 is used as the basic one, for $W/H = 1$ the total mesh size is 48 million cells, while for $\Delta = 8.4$ mm (mesh #3) it is about 140 million cells; if one uses mesh #2 the reduction of the computational cost is significant. For the basic mesh #2 the Kolmogorov scale for the mixing layer region is about 0.7 mm, and in the occupied zone it is about 2.5 mm. The ratio of the SGS to molecular viscosity is about 3.0 in the jet zone and less than 1.0 in the occupied zone (see figure 1b-d).

Figure 2. Effect of mesh refinement on the solution: skin friction coefficient over lines (a) $y = 3$ m, $z = 0.25$ m and (b) $y = 0$ m, $z = 0.25$ m; c) mean longitudinal velocity profiles at sections A-A and B-B; dashed lines indicate zoom zones I, II and III; arrows indicate inlet and outlet regions
4. Effect of the computational domain width on the time-averaged flow pattern
As discussed in Section 3, relatively narrow computational domain with \( \frac{W}{H} = 1/6 \) was used for mesh sensitivity analysis to reduce computational cost. However, the flow field for this narrow periodic problem differs much from the airflow in the 3D room with sidewalls (\( \frac{W}{H} = 1 \)) even at the mid-plane, as it was reported already in [9]. To evaluate the effect of the computational domain width on the time-averaged solution computed with the periodic conditions, two additional cases were computed, with \( \frac{W}{H} = 1/3 \) (the mesh of 16 million cells) and \( \frac{W}{H} = 1 \) (the mesh of 48 million cells).

The effect of \( \frac{W}{H} \) change is illustrated in figures 3,4, and it is evident that the computational domain width influences much the time-averaged flow structure. This is due to small values of \( \frac{W}{H} \) restrict the vortex size in transversal direction that results in some kind of artificial anisotropy. Formation of vortices with large scale not only in longitudinal, but also in transversal direction is visible in figure 1c,d where instantaneous SGS to molecular viscosity ratio distributions are given. At \( \frac{W}{H} = 1 \) vortex structures with large-scale in z-direction are detected in the occupied zone (see the plot at \( y/H = 1.0 \), figure 1c). They are prohibited if \( \frac{W}{H} \) is small, and it is the main reason for weakening of mean flow in the left part of the room visible in figure 3a for \( \frac{W}{H} = 1/3 \). The main recirculation zone is even more localized in the right part of the room if \( \frac{W}{H} = 1/6 \) [9], see also figure 1a.

When \( \frac{W}{H} \) becomes sufficient to allow large vortices formation, the global flow structure changes: as shown in figure 3b, the main recirculation zone penetrates to lower \( x \) and becomes larger. It corresponds to the flow pattern obtained at the mid-plane of the 3D room with sidewalls presented previously in [9]. The effect of \( \frac{W}{H} \) is illustrated also in figure 4 by means of longitudinal velocity profiles: the data for periodic problems with various \( \frac{W}{H} \) are plotted together with the experimental [7] and computed profiles for the problem with the sidewalls (\( \frac{W}{H} = 1 \)). Note that in all periodic cases the averaging periods were enough to get the mean flow patterns uniform in z-direction, while complicated 3D flow pattern is obtained for the problem with the sidewalls. The general conclusion is that \( \frac{W}{H} = 1 \) is enough to provide domain-independent periodic solution.

![Figure 3. Time-averaged velocity fields at mid-sections: (a) \( \frac{W}{H} = 1/3 \) and (b) \( \frac{W}{H} = 1 \)](image)

![Figure 4. Mean longitudinal velocity profiles at mid-sections, for problems with \( \frac{W}{H} = 1/6, 1/3 \) and 1](image)

5. Velocity-to-speed conversion procedure assessment
As mentioned in introduction, \( V_m \) distribution presented in figure 3 is not suitable to evaluate the ventilation conditions for occupants. It is necessary to use mean speed distributions, \( V_a \), instead. The difference between \( V_m \) and \( V_a \), both extracted from the periodic LES solution at \( \frac{W}{H} = 1 \), is illustrated in figure 5a. In the jet core with the predominant flow direction the difference between \( V_m \) and \( V_a \) does not exceed 10%. The difference between \( V_a \) and \( V_m \) exceeds 50% in the mixing shear layers and, what is more important, in the occupied zone. Locally the difference is up to 90%.
The difference between the LES-obtained $V_m$ field and the $V_m$ distribution computed using the $V_m$ field with the correction procedure proposed by Smirnov et al. [5], is given in figure 5b:

\[ V_{Smirnov} = (V_m^2 + 5/3<V_k>)^{0.5}, \]

where $<k> = (<V_x'^2>+<V_y'^2>+<V_z'^2>/2$ is resolved kinetic energy. The same distribution for the correction by Popiolek and Melikov [3]

\[ V_a^{Popiolek} = V_m(1 - 0.044 Tu_m + 1.195 Tu_m^2 - 0.329 Tu_m^3), \]

if $Tu_m \leq 1.3$, 

\[ V_a^{Popiolek} = V_m(0.287 + 1.502 Tu_m), \]

if $Tu_m > 1.3$, 

is given in figure 5c; in (2a), (2b) the turbulence intensity $Tu_m = 2(<k>/3)^{0.5}/V_m$ is used instead of $<k>$. Finally, application of the correction procedure proposed by Koskela et al. [2]

\[ V_a^{Koskela} = V_m(1 + Tu_m^2), \]

if $Tu_m \leq 0.45$, 

\[ V_a^{Koskela} = V_m[(1.596Tu_m^2 + 0.266Tu_m + 0.308)/(0.173 + Tu_m)], \]

if $Tu_m > 0.45$, results in the distribution shown in figure 5d. Note that white lines in figures 5c,d correspond to the $Tu_m$ values where transition from one correlation piece to another occurs.

The distributions shown demonstrate the validity of $V_a$ estimation if $V_m$ and $<k>$ data are available (i.e. when RANS modeling is applied). All correction procedures resulted in adequate $V_a$ estimation: on the average, the difference does not exceed 5%. Larger difference between the $V_a$ data and the result of the correction procedure application is visible locally in the occupied zone due to flow pattern peculiarities: locally it achieves 10% and even more; procedure [3] is a little bit more accurate there.

6. Pulsations of mean velocity and mean speed evaluation

Figure 6 presents fluctuations of velocity and speed extracted from the LES solution, $V_m' = \{(<V_x'^2>+<V_y'^2>+<V_z'^2>/3)^{0.5}$ and $V_a'$, the difference between two fields is shown in figure 7a. The values of $V_m'$ and $V_a'$ do not differ much in the jet mixing layer where pulsations are the most intensive. Noticeable difference between $V_m'$ and $V_a'$ is visible in the center of the recirculation zone and near the left sidewall, where pulsations are relatively small. In general, the difference between $V_m'$ and $V_a'$ does not exceed 20% over the most volume of the room.

Using the fields of $V_m$, and $<k>$ coupled with $V_a$ distribution obtained with one of the correction procedures, the speed standard deviation could be estimated as $V_a' = (V_m'^2 + 2<k>-V_m'^2)^{0.5}$. Evaluation of $V_a'$ distributions obtained with $V_a$ from three correction procedures (1), (2) and (3) is given in figures 7b-d. In all cases the difference between the estimated value and the $V_a'$ field extracted from
LES data is in the range of 20-30% that is comparable with the difference between $V_m'$ and $V_a'$. The same level of uncertainty of the standard deviation of the speed is reported in [3].

![Figure 7](image-url)

**Figure 7.** (a) The difference between the LES-obtained rms of mean velocity and mean speed; (b-d) rms mean speed evaluation with three various $V_m$ correction procedures; data are given at mid-section $z = 1.5m$ (WIH = 1)

7. Conclusions

Wall-modeled Large Eddy Simulation of airflow in an isothermal quasi-two-dimensional ventilated room have been performed using the ANSYS Fluent 16.2 package. Based on the conditions of the Nielsen et al. (1978) test, a simplified formulation with the periodicity boundary conditions in the transverse direction was used. It was proved that width-to-height ratio WIH = 1 is enough to provide domain-independent periodic solution. The mesh sensitivity study demonstrated that for the $Re = 5 \times 10^3$ considered it is enough to use 250 uniform cells per room height to get accurate solution.

Based on the LES data, evaluation of the mean velocity processing procedures available in the literature was performed. The processed $V_a$-fields are very close to the mean speed distributions extracted directly from the LES data. The results show that the mean speed can be estimated from the mean velocity and turbulent kinetic energy fields with an uncertainty of 5%, while the uncertainty of the standard deviation of the speed is up to 30%.

**Acknowledgment**

The work was supported by the Russian Foundation for Basic Research (grant No. 18-58-18011).

**References**

[1] Nielsen P V 2015 Fifty years of CFD for room air distribution *Build. Environ.* 91 pp 78-90
[2] Koskela H, Heikkinen J, Niemela R and Hautalampi T 2001 Turbulence correction for thermal comfort calculation *Building and Environment* 36 (2) pp 247-255
[3] Popiolek Z and Melikov A 2008 Improvement of CFD predictions of air speed turbulence intensity and draught discomfort *Proc. of 11 Int. Conf. on Indoor Air Quality* pp 1-8
[4] Piomelli U 2014 Large eddy simulations in 2030 and beyond *Phil. Trans. R. Soc. A* 372 pp 1-13
[5] Smirnov E M, Ivanov N G, Telnov D S and Son C H 2006 CFD modelling of cabin air ventilation in the International Space Station: a comparison of RANS and LES data with test measurements for the Columbus module *Int. J. of Ventilation* 5 (2) pp 219-27
[6] Zasimova M and Ivanov N 2018 Numerical simulation of air distribution in a room with a sidewall jet under benchmark test conditions *AIP Conference Proceedings* 1959 050033
[7] Nielsen P V, Restivo A and Whitelaw J H 1978 The velocity characteristics of ventilated room *ASME J. Fluids Engineering* 100 pp 291-98
[8] Davidson L and Nielsen P V 1996 Large eddy simulation of the flow in a three-dimensional ventilation room *Proc of the 5th Int. Conf. on Air Distribution in Rooms, ROOMVENT’96* (Yokohama, Japan, July 17-19, 1996) pp 161-68
[9] Ivanov N and Zasimova M 2018 Large Eddy Simulation of airflow in a test ventilated room *Int. Conf. Physics.A.Spb/2017 J. of Physics: Conf. Series* (Accepted for publication) pp 1-6
[10] Shur M L, Spalart P R, Strelets M K and Travin A K 2008 A hybrid RANS LES approach with Delayed DES and Wall Modelled LES capabilities *Int. J. Heat Fluid Flow* 29 (6) pp 1638-49