Large Eddy Simulation of airflow in a test ventilated room

N G Ivanov and M A Zasimova

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

E-mail: zasimova_ma@mail.ru

Abstract. The goal of the study is the validation of Wall-Modeled Large Eddy Simulation (WMLES) of ventilation in an isothermal room at conditions of experiments by Nielsen et al. Two values of the inlet slot width were considered. Calculations were carried out using the ANSYS Fluent 16.2 software with an algebraic WMLES subgrid-scale model. The uniform computational meshes up to 58 million cells were used. The focus in the results discussion is on the accuracy of mean and rms velocity fields prediction both in the jet zone and the low-velocity occupied zone. It is concluded that to predict the jet spread the meshes used are sufficient. However, there is visible difference between the computed and measured data in the occupied zone especially in the essentially 3D case with narrow inlet slot.

Design and optimization of indoor heating, ventilation and air conditioning (HVAC) systems with respect to space air diffusion aspects are often carried out using simplified analytical and empirical models (see, e.g., [1]). Along with the methods assuming complete air mixing, during last decades Computational Fluid Dynamics (CFD) simulation became popular in HVAC industry, as CFD models are able to produce detailed field information concerning air quality, e.g. location of stagnation and high-velocity zones, temperature distributions and contaminant transport information.

In engineering practice, the CFD models based on the Reynolds-Averaged Navier-Stokes (RANS) equations solution are widely used due to relatively moderate computational resources required. RANS approach has been applied to solve complex ventilation problems such as design of HVAC systems in an airplane cabin [2] and in an ice rink arena [3], as well as development of life support systems on board of the International Space Station (ISS) [4, 5]. However, RANS approach solves a set of transport equations obtained from the Navier-Stokes equations by means of the averaging procedure that results in the unknown Reynolds stress tensor emergence. To close the equations, the Reynolds stresses must be modeled, and the commonly used method is to apply the Boussinesq turbulent viscosity approximation [6]. To define the turbulent viscosity, a semi-empirical turbulence model should be involved, and the RANS results depend strongly on the particular turbulence model, especially when a fully developed turbulent flow is coupled with a moderate Reynolds number flow that is typical for ventilation problem [7].

Opposite to RANS, the Large Eddy Simulation (LES) approach solves the filtered Navier-Stokes equations that provides resolving large scales of motion, while small scales must be modeled with an appropriate subgrid-scale (SGS) model. The overview and outlook of LES could be found in [8]. LES, being an eddy-resolving technique, requires very significant computational resources. However, as accurate prediction of turbulent flows becomes more and more important, LES starts to be used in
wide range of industrial application. With respect to ventilation performance prediction, LES has been applied for the first time in [9] to simulate the flow in a 3D test ventilated room for which accurate velocity measurements was performed by Nielsen et al. [10]. Later LES method was used several times to solve model indoor airflows, see [11] and references therein. LES approach was applied also in [12] to the ISS Columbus module ventilation modeling and the simulation results were compared with the measurement data available. Though satisfactory agreement with experimental data was reported in the papers cited, relatively coarse computational meshes of 0.5-2 million cells were used, and a revision of early LES data is anticipated. An example of recent LES simulation is a cross-ventilation flow study [13], but even in this paper it was concluded that due to computational limitations it was possible to run LES using a relatively coarse mesh.

The growing need for CFD verification and validation in the indoor air applications is emphasized by many authors, see, e.g., [6, 13]. The key factor required for CFD results validation is an appropriate experimental data. Experiment by Nielsen et al. [10] is one of the well-documented isothermal tests where the buoyancy effects were excluded. Accurate mean velocity data and corresponding rms values were obtained with the Laser-Doppler Anemometry (LDA) technique, with the uncertainty less than 0.5% for mean velocities [10] (see also [14] and references therein for detailed description of the test). The goal of the present study is the validation of the LES of ventilation airflow. Numerical solutions were obtained with the CFD package ANSYS Fluent 16.2.

1. Problem formulation and computational settings

Figure 1a shows the geometry model adopted for the computations of the indoor airflow that corresponds to the test facility [10]. The rectangular room with the height of \( H = 3 \) m was considered, the longitudinal and transverse dimensions of the room were \( L/H = 3.0 \) and \( W/H = 1.0 \). The rectangular inlet slot of height \( h_{in}/H = 0.056 \) was placed under the ceiling (in the center of the wall with respect to \( z \)-coordinate). The origin of the coordinate system is in the corner of the room (see figure 1a). Two cases with various values of the slot width, \( w_{in} \), were considered: Case 1 with \( w_{in}/H = 1.0 \) and Case 2 with \( w_{in}/H = 0.5 \). The dimensions of the outlet slot located on the opposite wall near the floor were \( w_{out}/H = 1.0 \) and \( h_{out}/H = 0.16 \). An outlet channel with the length of \( L_{out}/H = 0.5H \) was included into the computational domain.

![Figure 1](image-url)

**Figure 1.** (a) Computational domain; instantaneous velocity magnitude distributions at cross-sections \( z/H = 0.1, 0.5 \) are shown; (b, c) inlet velocity profiles for Case 1 at (b) \( z/H = 0.5 \) and (c) \( y/H = 0.972 \)

The measurement data [10, 14] are available at eight dashed lines shown in figure 1a: vertical lines A-A located at \( x/H = 1.0 \) and B-B at \( x/H = 2.0 \) and horizontal lines C-C located at \( y/H = 0.972 \) (starting at the mid-height of the inlet slot) and D-D located at \( y/H = 0.028 \) (that is equal to \( h_{out}/2 \)). The subscript «1» in the line notation corresponds to \( z/H = 0.5 \), while subscript «2» – to \( z/H = 0.1 \).
Besides the main 3D problem with side walls, an additional 3D problem with the periodicity boundary condition in the transversal z-direction was set to perform mesh sensitivity study at lower computational cost. The transverse size of the computational domain for the parametric computations was $W_{ped}/H = 1/6$ (note, that an increase of $W_{ped}/H$ by two times did not change the flow pattern).

Air was assumed as an incompressible fluid with constant physical properties ($\rho = 1.225$ kg/m$^3$, $\mu = 1.8\times10^{-4}$ kg/m-s). The boundary conditions approximately reproduced the experimental conditions [10]. To provide this, LES computations of turbulent flow in a straight rectangular duct with dimensions $L_{duct}/H = 2.0$, $W_{duct}/H = 1.0$, $H_{duct}/H = 0.056$ were performed prior to the main room airflow simulation. The time-averaged velocity profile extracted from the duct flow solution was set as the boundary condition at the inlet slot section. The computed inlet profiles are compared with the experimental data in figure 1b,c. The bulk velocity at the inlet slot was $V_{in} = 0.455$ m/s that corresponded to the Reynolds number $Re = \rho h_{in} V_{in}/\mu = 5233$. For the periodic problem, the 1D profile corresponding periodic duct flow was set. To model the fluctuating velocity at the inlet slot, the vortex method available in ANSYS Fluent was used.

For the incompressible fluid, the filtered Navier-Stokes equations are written as follows:

$$\frac{\partial \hat{\mathbf{V}}}{\partial t} + \nabla \cdot (\mathbf{V} \hat{\mathbf{V}}) = -\frac{1}{\rho} \nabla \cdot p + 2\nu(\nabla \cdot \mathbf{S}) - \nabla \cdot \tau_{SGS}$$

(1)

where $\hat{\mathbf{V}}$ is the velocity vector with components $(u, v, w)$, $\mathbf{S}$ is the strain rate tensor for the resolved motion, and $\tau_{SGS}$ is the SGS stress term arising from the spatial filtering procedure. The filtering operation [8] for a variable $f$ determines the filtered (resolved) and small-scale (non-resolved) components $\hat{f}$ and $f^\prime$ as follows:

$$\hat{f}(x, t) = \int_{V_{vol}} G(x - x', \Delta) f(x', t) dx'. f^\prime = f - \hat{f},$$

(2)

where $G(x - x', \Delta)$ is a filter function that determines the size and structure of the small scales, $x$ is a coordinate of the point under consideration, and $\Delta$ is the filter width.

To determine the SGS stress term, the generalized Boussinesq hypothesis is used:

$$\tau_{ij}^{SGS} = \frac{1}{3} \delta_{ij} \tau_{kk}^{SGS} - 2\nu_{SGS} S_{ij}$$

(3)

where $\nu_{SGS}$ is the SGS viscosity, which must be determined by a SGS model.

The algebraic Wall-Modeled LES (WMLES) S-Omega SGS model available in ANSYS Fluent was applied in the current study. The model realization in the code is based on [15], and the SGS viscosity is calculated with the use of a hybrid length scale and the wall-damping function:

$$\nu_{SGS} = \min \left\{ \left( k d_w \right)^2, \left( C_s \Delta \right)^2 \right\} \left( S - \Omega \right) \left( 1 - \exp \left( \left( -y^+ / 25 \right)^4 \right) \right)$$

(4)

where $S$ and $\Omega$ are the strain rate and vorticity magnitude, $C_s = 0.2$ is the Smagorinsky constant, $\kappa = 0.41$ is the von Karman constant, $d_w$ is the distance to the nearest wall, $y^+$ is the normal to the wall inner scaling. The grid scale is defined as follows:

$$\Delta = \min \left\{ \max \left( C_u d_w, C_w \Delta_{max} \Delta_{ewn} \right), \Delta_{max} \right\}$$

(5)

where $\Delta_{max}$ and $\Delta_{ewn}$ are the maximum local grid spacing and the grid step in the wall-normal direction, and $C_u = 0.15$ is the empirical constant.

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. For all the cases computed, the uniform meshes with cubic cells were used. The size of the meshes used is discussed below, in Section 2. For all the meshes the maximum values of the normalized distance from the first computational cell to the wall 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 0.01 s, and it was chosen to provide the Courant number in the computational domain less than 1 for all the
meshes considered. Note that a special time-step sensitivity study was performed with \( \Delta t = 0.006 \) s, and no changes in the averaged characteristics were fixed. To accumulate representative statistics, it was required to calculate samples of about 150,000 time steps (1500 s). It was checked that the averaging time has been sufficient to obtain statistically steady data. It was possible to perform massively parallelized computations (up to 512 cores) due to computational resources of the Peter the Great St. Petersburg Polytechnic University supercomputer center (scc.spbstu.ru).

2. Mesh-sensitivity analysis for periodic problem

The mesh-sensitivity analysis was performed for the periodic problem, as it was not possible to complete a mesh dependence study for the room problem with side walls due to enormous computational cost. The initial coarse mesh consisted of 536\( \times \)179\( \times \)30 uniform cubic cells that corresponds to about 3.0 million cells. The initial mesh was refined three times, and the total cell number for each successively refined mesh was increased by a factor of \( 8^{0.5} \) (i.e., \( 2^{0.5} \) times in each direction), so that mesh #2 had 8 million cells, #3 – 23, and #4 – 58 million cells.

Sure, the flow pattern computed with the periodic assumption differs much from the airflow in the 3D room with side walls (\( W/H = 1.0 \)) even at the mid-plane. Time-averaged velocity magnitude fields computed with two problem formulations are compared in figure 2 (here and after the dimensional velocities are presented). The use of the periodic assumption results in more pronounced recirculation flow as there is no side walls that slow down the flow, and the averaged flow pattern was uniform in z-direction. However, in both the cases the flow patterns could be separated into two zones: the relatively high-velocity zone of the near-wall jet with the mixing layer region and the relatively low-velocity occupied zone with the clockwise recirculation flow. Conclusions from the mesh dependence discussion for the periodic problem could be extended to the problem with side walls.

![Figure 2. Mean velocity fields at mid-sections: (a) the periodic problem; (b) the problem with \( W/H = 1.0 \)](image)

![Figure 3. Longitudinal velocity profiles for the periodic problem](image)

![Figure 4. SGS to molecular viscosity ratio for the periodic problem; meshes of (a) 8 and (b) 58 million cells](image)

Figure 3 shows the mean longitudinal velocity profiles along two lines in the vertical mid-plane, with zoomed near-wall regions. The profiles demonstrate monotonous growth of velocity values with mesh refinement in the jet zone. The deviation from results obtained with the finest mesh seems to be
critical for mesh #1, and could be treated as acceptable for mesh #2. In the occupied zone, the velocity values also depend on mesh size, but not as much as in the jet zone. Thus, mesh #2 provides nearly mesh independent results. Figure 4 gives additional information on the contribution of the non-resolved scales for the meshes used: for the finest mesh the ratio of the SGS to molecular viscosity is less than 3.0 everywhere in the computational domain (figure 4b).

Based on the results obtained for the periodic problem, for the room with side walls the basic mesh of almost 48 million cells was used (751×252×250 cells), and the results obtained with this mesh are discussed below in Section 3. The mesh provides the cell length in each direction of 12 mm. Note that in this case the Kolmogorov scale for the mixing layer region was about 0.7 mm, and in the occupied zone it was about 2.5 mm.

3. Comparison of WMLES results with the experimental data

Figure 5 compares computed and measured [10] profiles of mean longitudinal velocity and its pulsations: the upper row of the plots corresponds to the mid-section of the room, while the lower row demonstrates flow behavior near the side wall. It is visible that the jet spreading is reproduced quite accurate (the only deviation is in the $\langle u \rangle$ values at large $x$, figure 5b that corresponds to slight overprediction of the mixing in the jet, visible also in figure 5d). At the same time there is a pronounced difference between the computational velocity profiles and the experimental data in the recirculation zone (figure 5c,f); the pulsations in this zone are underpredicted.

![Figure 5](image_url)

**Figure 5.** Mean longitudinal and rms velocity, Case 1, $w_{in}/H = 1.0$: (a – c) $z/H = 0.5$ and (d – f) $z/H = 0.1$

![Figure 6](image_url)

**Figure 6.** Mean longitudinal velocity, $w_{in}/H = 0.5$: (a) $z/H = 0.5$ and (b) $z/H = 0.1$

The difference between the computed and measured data is even larger if Case 2 with narrow inlet slot is considered. Inlet slot width reduction leads to appearance of additional mixing layers and 3D effects are more pronounced. The computed mean longitudinal velocity profiles are compared with the measurements in figure 6 (unfortunately, experimental pulsation data are not available for this case).
The main difference is visible in figure 6b, at line B2-B2 that corresponds to the turn of the stream from the inlet. This difference due to inaccuracy of mixing prediction (see line B1-B1); another reason for it may be in non-symmetrical recirculation flow pattern that will be analyzed in the future.

4. Conclusions

Wall-Modeled Large Eddy Simulation of indoor air flow in an isothermal ventilated room at conditions of benchmark test experiments at Re of 5×10^3 have been performed using the ANSYS Fluent 16.2 CFD package. The mesh-dependence study has been performed for a simplified periodic problem formulation using the uniform meshes up to 58 million cells. The conclusion is that for the mesh with 250 cells per room height provides nearly mesh-independent results.

Comparison of WMLES results with the experimental data on mean and rms velocities results in conclusion that the mesh used is sufficient to predict accurately the jet spread, though there is slight overprediction of the jet mixing. At the same time, there is some difference between the computed and measured data in the low-velocity occupied zone, especially for the essential 3D case with narrow inlet slot. The future work will focus on the detection of possible reasons for this difference, as well as on the detailed analysis of the unsteady 3D structure of the recirculation flow.

References

[1] Wang S K 2001 Handbook of air conditioning and refrigeration (New York: McGraw-Hill)
[2] Liu W, Wen J, Chao J, Yin W, Shen C, Lai D, Lin C H, Liu J, Sun H and Chen Q 2012 Accurate and high-resolution boundary conditions and flow fields in the first-class cabin of an MD-82 commercial airliner Atmospheric Environment 56 pp 33-44
[3] Palmowska A and Lipska B 2016 Experimental study and numerical prediction of thermal and humidity conditions in the ventilated ice rink arena Building and Environment 108 pp 171-82
[4] Son C H, Ivanov N G, Telnov D S and Smirnov E M 2015 Integrated computational fluid dynamics ventilation model for the international space station Proc. of the 35th Int. Conf. on Environmental Systems (Rome, Italy, July 11-14 2015) SAE Paper 2005-01-2794 pp 1-7
[5] Son C H, Ivanov N G, Telnov D S and Smirnov E M 2012 Numerical Study of Ammonia Leak and Dispersion in the International Space Station Proc. of the 42nd Int. Conf. on Environmental Systems (San Diego, California, USA, July 15-16 2012) pp 1-10
[6] Hinze J O 1975 Turbulence (New York: McGraw-Hill)
[7] Li Y and Nielsen P V 2011 CFD and Ventilation Research Indoor Air 21 pp 442-53
[8] Piomelli U 2014 Large eddy simulations in 2030 and beyond Phil. Trans. R. Soc. A 372 pp 1-13
[9] 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
[10] Nielsen P V, Restivo A and Whitelaw J H 1978 The velocity characteristics of ventilated room ASME J. Fluids Engineering 100 pp 291-98
[11] Jiang Y, Su M and Chen Q 2003 Using Large Eddy Simulation to Study Airflows in and around Buildings ASHRAE Transactions 109 (2) pp 517-26
[12] Smirnov E M, Ivanov N G, Telnov D S and Son C H 2006 CFD modeling 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 219-28
[13] Hooff T, Blocken B and Tominaga Y 2017 On the accuracy of CFD simulations of cross-ventilation flows for a generic isolated building: Comparison of RANS, LES and experiments Building and Environment 114 pp 148-65
[14] Nielsen P V 1990 Specification of a two-dimensional test case Aalborg: Institutet for Bygningsteknik, Aalborg Universitet, Gul serie R9040 (8) pp 1-15
[15] 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. of Heat and Fluid Flow 29 (6) pp 1638-49