On the accuracy of CFD simulations of cross-ventilation flows for a generic isolated building: Comparison of RANS, LES and experiments

T. van Hooff a, b, *, B. Blocken a, b, Y. Tominaga c

a Department of Civil Engineering, KU Leuven, Kasteelpark Arenberg 40 – bus 2447, 3001 Leuven, Belgium
b Department of the Built Environment, Eindhoven University of Technology, P.O. Box 513, 5600 MB Eindhoven, The Netherlands
c Department of Architecture and Building Engineering, Niigata Institute of Technology, 1719, Fujihashi, Kashiwazaki, Japan

Abstract

Accurate and reliable computational fluid dynamics (CFD) simulations are essential for the assessment of cross-ventilation of buildings. To determine which CFD models are most suitable, validation studies are required. A detailed review of the literature indicates that most CFD validation studies only employed the 3D steady Reynolds-averaged Navier-Stokes (RANS) approach and/or focused on a limited set of flow parameters. Therefore, the objective of this paper is the validation of both 3D steady RANS simulations and large eddy simulation (LES) of cross-ventilation in a generic isolated enclosure with wind-tunnel measurements. The evaluation is based on five parameters: mean velocity, turbulent kinetic energy, ventilation flow rate, incoming jet angle and incoming jet spreading width. The RANS simulations are conducted with the standard k–ε (SKE), RNG k–ε, realizable k–ε (RIZ), SST k–ω and RSM turbulence models, whereas the LES is performed with the dynamic Smagorinsky subgrid-scale model. SST/RNG/RSM reproduce the experimentally observed direction of the incoming jet, but all RANS models fail in reproducing the turbulent kinetic energy, which is too low especially above and below the jet, because steady RANS does not capture the vertical flapping of the jet. This transient feature is reproduced by LES, resulting in a better reproduction of all three measured parameters (velocity, turbulent kinetic energy, volume flow rate). It is concluded that choice of the model (RANS vs. LES) actually depends on which parameter is the target parameter, noting that the use of LES entails an increase in computational demand with a factor of ≈ 80–100.

© 2016 The Authors. Published by Elsevier Ltd. This is an open access article under the CC BY-NC-ND license (http://creativecommons.org/licenses/by-nc-nd/4.0/).

1. Introduction

1.1. Literature review

Cross-ventilation is an important ventilation method since it can provide a fast and effective way to remove large amounts of pollutants and to quickly release internal heat from a building or other enclosure (i.e. ventilative cooling). For detailed analysis of natural ventilation in general, and cross-ventilation in particular, researchers are resorting more and more to the use of computational fluid dynamics (CFD) (e.g. Refs. [1–14]). The increased use of CFD for such studies can be attributed to the strong interaction between the outdoor wind flow and indoor airflow when natural ventilation occurs through large openings, which makes the resulting ventilation flow through an enclosure difficult to predict using analytical or (semi-)empirical models. For example, Straw et al. [4] showed in their cross-ventilation study that the use of pressure coefficients in combination with the orifice equation did not result in an accurate prediction of volume flow rates when compared to the measured volume flow rates (deviations of 28–32%). The observed deviations were attributed to either assumptions needed for the values of the discharge coefficient and/or values for the pressure coefficients obtained from a sealed body that do not correspond with the pressure distribution on the same body when (large) openings are present [4]. The coupled CFD simulation (indoor and outdoor airflow modeled simultaneously in the same computational domain) reported in their publication did provide far more accurate results, with a deviation of only 3–9% from the measurements [4].

Most natural ventilation studies presented in the past used the Reynolds-averaged Navier-Stokes (RANS) approach, in which the
Reynolds decomposition is used to split the flow variables in a mean and a fluctuating component. In RANS CFD simulations, the mean flow is resolved whereas the effects of turbulence (fluctuating components) on the mean flow is modeled using a turbulence model. Since assumptions have to be made to model these effects, a proper validation study is imperative to ensure an accurate solution of the natural ventilation flow using RANS. In large eddy simulation (LES), the mean flow and the large-scale turbulence with a length scale larger than the filter size (often taken as the grid size) are resolved and the small-scale turbulence is modeled using a subgrid-scale model. Although LES is an intrinsically more accurate method for CFD simulations of wind flows (e.g. Refs. [15–18]), its use entails a (very) large increase in computational demand (e.g. factor > 10^5 [1]) and it is therefore often not a viable option in research and consultancy (e.g. Refs. [1,2]).

To assess the performance of RANS and LES for cross-ventilation flows, detailed validation studies are required using high-resolution, well-defined and well-documented reduced-scale or full-scale experiments. A detailed review of the literature has been performed, resulting in Table 1 that provides an overview of CFD cross-ventilation studies with validation based on either reduced-scale wind-tunnel or full-scale on-site measurements, or both. Table 1 shows that the majority of these publications considered validation for RANS turbulence models only and only a relatively limited number of these studies compared multiple turbulence models with experimental data. Table 1 also indicates the type of study (generic or applied) and the flow parameters for which the validation study was performed, i.e. validation based on velocities, turbulence levels, volume flow rates, surface pressures, velocity vector fields, etc. Most studies listed in Table 1 included one to three parameters in the validation. For example, Lee et al. [27] performed particle image velocimetry (PIV) measurements in a generic greenhouse and used these results for a comparison with results from CFD simulations using the standard k-ε, renormalization group (RNG) k-ε, and realizable k-ε models, and a Reynolds stress model (RSM). Mean velocity and turbulence intensity were compared along one vertical line in

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Kurabuchi et al. (2000)</td>
<td>[21]</td>
<td>Generic</td>
<td>RANS</td>
<td>SKE, LK, MMK</td>
<td>WT</td>
<td>V,TP,Q,P</td>
</tr>
<tr>
<td>Bartzanas et al. (2004)</td>
<td>[23]</td>
<td>Applied</td>
<td>RANS</td>
<td>SKE</td>
<td>FS</td>
<td>V,Q,T</td>
</tr>
<tr>
<td>Hu et al. (2005)</td>
<td>[26]</td>
<td>Generic</td>
<td>RANS</td>
<td>SKE, RNG, SKO, SST</td>
<td>WT</td>
<td>V,TP,Q</td>
</tr>
<tr>
<td>Lee et al. (2005)</td>
<td>[27]</td>
<td>Generic</td>
<td>RANS</td>
<td>SKE, RNG, RIZ, RSM</td>
<td>WT</td>
<td>V,VF,TP</td>
</tr>
<tr>
<td>Mochida et al. (2005)</td>
<td>[28]</td>
<td>Applied</td>
<td>RANS</td>
<td>DKE</td>
<td>FS</td>
<td>V</td>
</tr>
<tr>
<td>Mochida et al. (2006)</td>
<td>[29]</td>
<td>Applied</td>
<td>RANS</td>
<td>DKE</td>
<td>FS</td>
<td>V,Q</td>
</tr>
<tr>
<td>Fatnassi et al. (2006)</td>
<td>[31]</td>
<td>Applied</td>
<td>RANS</td>
<td>SKE</td>
<td>FS</td>
<td>Q</td>
</tr>
<tr>
<td>Bartzanas et al. (2007)</td>
<td>[33]</td>
<td>Applied</td>
<td>RANS</td>
<td>SKE, RNG, RIZ, RSM</td>
<td>FS</td>
<td>Q</td>
</tr>
<tr>
<td>Hu et al. (2008)</td>
<td>[34]</td>
<td>Generic</td>
<td>LES</td>
<td>Standard Smagorinsky</td>
<td>WT</td>
<td>V,TP,P</td>
</tr>
<tr>
<td>Kobayashi et al. (2009)</td>
<td>[37]</td>
<td>Generic</td>
<td>RANS</td>
<td>RSM</td>
<td>WT</td>
<td>Q,P</td>
</tr>
<tr>
<td>Kurabuchi et al. (2009)</td>
<td>[38]</td>
<td>Applied</td>
<td>RANS</td>
<td>DKE</td>
<td>WT</td>
<td>V,VF,Q,P</td>
</tr>
<tr>
<td>Norton et al. (2009)</td>
<td>[40]</td>
<td>Applied</td>
<td>RANS</td>
<td>SKE</td>
<td>RS</td>
<td>V,VF</td>
</tr>
<tr>
<td>Kobayashi et al. (2010)</td>
<td>[41]</td>
<td>Generic</td>
<td>RANS</td>
<td>RSM</td>
<td>WT</td>
<td>V,VF,P</td>
</tr>
<tr>
<td>Nikas et al. (2010)</td>
<td>[43]</td>
<td>Generic</td>
<td>RANS</td>
<td>SKO</td>
<td>WT/FS</td>
<td>V,Q</td>
</tr>
<tr>
<td>van Hooff &amp; Blocken (2010a,b)</td>
<td>[44]</td>
<td>Generic</td>
<td>RANS</td>
<td>RIZ</td>
<td>FS</td>
<td>V</td>
</tr>
<tr>
<td>Cheung and Liu (2011)</td>
<td>[45]</td>
<td>Generic</td>
<td>RANS</td>
<td>SKE</td>
<td>WT</td>
<td>V,VF,Q,P</td>
</tr>
<tr>
<td>Larsen et al. (2011)</td>
<td>[46]</td>
<td>Generic</td>
<td>RANS</td>
<td>SKO</td>
<td>WT/FS</td>
<td>V</td>
</tr>
<tr>
<td>Lo &amp; Novoselac (2011)</td>
<td>[47]</td>
<td>Generic</td>
<td>RANS</td>
<td>RNG</td>
<td>FS</td>
<td>TP,Q</td>
</tr>
<tr>
<td>Ramponi &amp; Blocken (2012a,2012b)</td>
<td>[49]</td>
<td>Generic</td>
<td>RANS</td>
<td>SKE, RNG, RIZ, SKO, SST, RSM</td>
<td>WT</td>
<td>V,VF</td>
</tr>
<tr>
<td>Mohamed et al. (2013)</td>
<td>[51]</td>
<td>Generic</td>
<td>RANS</td>
<td>RNG</td>
<td>WT</td>
<td>Q</td>
</tr>
<tr>
<td>Peren et al. (2014a-c, 2016)</td>
<td>[53]</td>
<td>Generic</td>
<td>RANS</td>
<td>SKE, RNG, RIZ, SKO, SST, RSM</td>
<td>WT</td>
<td>V,VF</td>
</tr>
<tr>
<td>Tong et al. (2016)</td>
<td>[55]</td>
<td>Generic</td>
<td>LES</td>
<td>Dynamic Smagorinsky</td>
<td>WT</td>
<td>V</td>
</tr>
<tr>
<td>Tong et al. (2016)</td>
<td>[56]</td>
<td>Generic</td>
<td>LES</td>
<td>Dynamic Smagorinsky</td>
<td>WT</td>
<td>V</td>
</tr>
</tbody>
</table>

the middle of the greenhouse and they concluded that the best agreement was obtained with the RNG k-ε model and the RSM model, with average errors of 7.9% and 7.6%, respectively, for mean velocity. However, Lee et al. [27] also stated that the average errors of turbulence intensity for the RNG k-ε model and the RSM model were much larger: 37.5% and 43.2%, respectively. A comparison of the measured velocity vector fields with those obtained from CFD with the RNG k-ε model showed very similar internal flow patterns [27].

Evoli and Popov [30] compared the mean velocity along three vertical lines inside a cubical building as measured by Jiang et al. [6] in a wind tunnel with the results from steady RANS CFD simulations with both the standard k-ε model and the RNG k-ε model. The agreement was better with the RNG k-ε model than with the standard k-ε model. In terms of volume flow rate, the deviation was 8.7% for the RNG k-ε model and 13.7% for the standard k-ε model. In addition, they compared the mean surface-averaged external pressure coefficients for the roof and the four facades of the building. Hu et al. [26] used the experimental results presented by Kurabuchi et al. [24] for a validation study of four RANS turbulence models (standard k-ε, RNG k-ε, standard k-ω, shear stress transport (SST) k-ω), and focused on mean velocity vector fields and contours of turbulent kinetic energy, which were qualitatively compared in the vertical center plane. They concluded that both k-ε models showed a better agreement with the measurements than the k-ε models, mainly due to a too horizontal jet trajectory predicted by the k-ε models [26]. In addition, Hu et al. [26] showed that the predicted values of turbulent kinetic energy inside the enclosure were lower than the measured ones, while they were higher than the measured ones in the stagnation region outside the enclosure. Overall, they concluded that the best agreement between the experimental and numerical data with respect to the mean velocity vector fields and turbulent kinetic energy contours was obtained using the SST k-ω model [26]. Finally, Hu et al. [26] compared the experimentally obtained volume flow rates with those obtained using the SST k-ω model for different wind directions; the results were within = 12% for each wind direction with an average deviation of 5%. Ramponi and Blocken [48,49] and Peren et al. [53–56] used PIV measurements by Karava et al. [64] to validate steady RANS CFD models of cross-ventilation flow. Ramponi and Blocken [48,49] made a comparison of the performance of the standard k-ε model, realizable k-ε model, RNG k-ε model, standard k-ω model, SST k-ω model, and a RSM model with respect to the mean streamline velocity along a horizontal line through the center of the two symmetrically placed window openings (vertical location in middle of facades). Also mean velocity vector fields were compared with the measured ones. The best agreement was obtained with the SST k-ω model followed by the RNG k-ε model [48]. These two models were the only ones that accurately predicted the direction of the incoming jet through the windward opening. Peren et al. [53–56] also used the experimental data by Karava et al. [64], but this time for asymmetrical window openings; i.e. windward opening in bottom part and leeward opening in upper part of facade. A comparison was made along a horizontal line through the windward opening and along a diagonal line through both the windward and leeward opening. Their validation study resulted in the same conclusions as by Ramponi and Blocken [48]; with the best agreement obtained by the SST k-ω model and the RNG k-ε model [53–56]. Turbulent kinetic energy levels were not assessed in these studies.

A few validation studies were conducted in which only LES was assessed. The early and pioneering study by Kato et al. [3] compared LES simulations with the standard Smagorinsky subgrid-scale model with their own wind-tunnel experiments finding a good agreement with respect to the mean velocity along four vertical lines, and also with respect to the volume flow rates (within 13%). Kato et al. [3] also compared mean external surface pressure coefficients on the windward and leeward facade and mean surface pressure coefficients on the floor obtained from the experiments with those from LES. LES was capable of reproducing the pressures on the windward facade and ground surface, but deviations were observed for the leeward facade [3]. Also Jiang et al. [6], Kurabuchi et al. [24] and Hu et al. [33] compared LES simulations with the standard Smagorinsky subgrid-scale model with wind-tunnel measurements and found good qualitative agreements as well. Note that no specific quantification of the agreement between CFD and LES was provided at that time.

Only a very limited number of publications reported validation studies including both steady RANS and LES simulations. Kurabuchi et al. [21] compared split-film anemometer measurements with both steady RANS and LES simulations. In the steady RANS simulations, the standard k-ε model and two modified k-ε models were used, whereas the LES simulation used the standard Smagorinsky subgrid-scale model. Based on velocity vector plots and contours of turbulent kinetic energy they concluded that LES provided an excellent qualitative agreement with the measurements, and was more accurate than the RANS simulations. Kurabuchi et al. [21] also compared the predicted flow rates through the building with flow rates based on tracer gas (ethylene) measurements obtaining a very good agreement with LES (2.3% deviation) and a good agreement with the RANS models (4.6–7% deviation). In addition, it was shown that the modified k-ε models showed a better qualitative agreement than the standard k-ε model based on velocity vector plots and contours of turbulent kinetic energy. Finally, they compared measured mean external surface pressure coefficients on the windward and leeward facade and on the roof with results obtained from steady RANS and LES and found a much better agreement with LES than with RANS, both at the windward and leeward facade.

1.2. Introduction of current study

The overview of cross-ventilation validation studies in Section 1.1 clearly demonstrates that there is currently no consensus in the literature on which CFD method provides the best results and the best balance between accuracy and computational demand, which is at least partly due to the lack of validation studies based on detailed experimental data of mean velocity and turbulence intensity and including both RANS simulation and LES simulations; previous studies generally only assessed one of both methods. For example, several studies have reported the inability of steady RANS to accurately predict turbulent kinetic energy inside the enclosure (e.g. Refs. [26,27]), while the predictions of velocities are often stated to be sufficiently accurate. In addition, most validation studies focused on the comparison of mean velocity vector fields, mean velocity and turbulence kinetic energy or volume flow rates, or a combination of these. Although several studies pointed to difficulties in correctly predicting the incoming jet, to the best knowledge of the authors, no validation study has yet analyzed the behavior of the incoming jet and its reproduction by CFD simulations (either RANS or LES) in more detail.

Therefore, the objective of this paper is the validation of both 3D steady RANS simulations and LES of cross-ventilation in a generic isolated enclosure with wind-tunnel measurements from literature [65,66]. The RANS simulations are conducted with the standard k-ε (SKE), RNG k-ε, realizable k-ε (R2), SST k-ω and RSM turbulence models, whereas the LES is performed with the dynamic Smagorinsky subgrid-scale model. The evaluation is based on five parameters: mean velocity, turbulent kinetic energy, ventilation flow rate, incoming jet angle and incoming jet spreading width. The nearly cubical enclosure is equipped with two windows, one on both the windward and on the leeward side of the building. To the best knowledge of the authors, such a detailed validation study for both
RANS and LES, including an analysis of the jet dynamics, has not been reported before and can thus be regarded as novel.

The wind-tunnel experiments and the building geometry are presented in Section 2. In Section 3 the computational settings are described and the results of the grid-sensitivity analysis are presented. The comparison of the steady RANS simulations with the experiments is provided in Section 4. The results of the LES simulation are presented in Section 5 and a more detailed comparison between the RANS and LES results is provided in Section 6. Section 7 (Discussion) and Section 8 (Conclusions) conclude this paper.

2. Description of experiments

Tominaga and Blocken [65] reported reduced-scale measurements of wind-induced cross-ventilation in the atmospheric boundary layer (ABL) wind tunnel at Niigata Institute of Technology in Japan. The wind tunnel has a test section of $13 \times 1.8 \times 1.8 \, m^3$ ($L \times H \times W$). An atmospheric boundary layer (ABL) velocity profile was created using a combination of spires and surface roughness elements. The velocity was measured using a split fiber probe (SFP) (Dantec Dynamics; 55R55) and a constant temperature anemometry (CTA) module (Dantec Dynamics; 90C10) to identify the three-dimensional components of the velocity vector [65]. The measured vertical approach flow profiles of mean streamwise velocity and turbulent kinetic energy are depicted in Fig. 1. The mean streamwise velocity can be fitted to a power law with an exponent of 0.25 [64]:

$$\frac{U(z)}{U_H} = \left(\frac{z}{H}\right)^{0.25}$$

with $U_H$ the velocity at building height $H$. The turbulent kinetic energy values were obtained by three-component measurement of the variances in the velocity fluctuations. This distribution can be approximated by the following equation [65]:

$$\frac{k(z)}{U_H^2} = 0.033 \exp^{-0.32\left(\frac{z}{H}\right)}$$

The velocity at building height $H$ ($U_H$) was equal to 4.3 m/s. The dimensions of the building model under study were $0.2 \times 0.2 \times 0.16 \, m^3$ ($D \times W \times H$) (Fig. 2). An opening area of $3.3 \times 10^{-3} \, m^2$ was present in both the windward and the leeward facade. The thickness of the walls and ceiling was 3 mm ($0.003 \, m$).

3. CFD simulations: computational settings and parameters

3.1. Computation geometry, domain and grid

The computational model represents the reduced-scale model used in the wind-tunnel measurements. The thickness of the walls and ceiling of 3 mm is explicitly included in the computational model.

The computational domain is constructed based on the best practice guidelines by Franke et al. [67], Tominaga et al. [68] and Blocken [69]; i.e. a distance of $5H$ from the building to the top and sides of the computational domain and a distance of $15H$ between the building and the outlet boundary downstream of the building. The upstream length of the domain is reduced to $3$ times the height of the building to limit the occurrence of unintended streamwise gradients [69,70]. The resulting dimensions of the domain are $5.32 \times 1.8 \times 0.96 \, m^3$ ($L \times W \times H$) (Fig. 3a).

The computational grid is created using the surface-grid extrusion technique by van Hooff and Blocken [11] and is shown in Fig. 3b–d. The computational grid consists of hexahedral cells only, with a high spatial resolution near the building and its openings. The grid resolution results from a grid-sensitivity analysis using three different grids, created by refining and coarsening the basic grid with a factor of $\sqrt{2}$ in each direction. The results of the grid-sensitivity analysis are presented in Section 3.4.

3.2. Boundary conditions

The boundary conditions reproduce the conditions during the wind-tunnel experiments as much as possible. A logarithmic inlet velocity profile is constructed based on a fit with the power law profile described in Eq. (1). The resulting log-law equation is:

![Fig. 1. Approach flow profiles of dimensionless mean velocity and turbulent kinetic energy [65].](image1)

![Fig. 2. Building geometry. (a) Front view. (b) Vertical cross-section. Dimensions in m.](image2)
The corresponding value of $u^{*}_{ABL}$ and $z$ the height coordinate. The log law is preferred over the power law because it yields the value of $u^{*}_{ABL}$ that is needed for the profile of the turbulence dissipation rate $\varepsilon$. The aerodynamic roughness length $z_0$ is determined based on a fitting procedure using the measured velocity profile yielding $z_0 = 0.0009 \text{ m}$ at reduced scale. The corresponding value of $u^{*}_{ABL}$ is 0.348 m/s for a reference velocity at building height of $U_H = 4.3 \text{ m/s} (H = 0.16 \text{ m})$. The turbulent kinetic energy $k$ is given by Eq. (2). The turbulence dissipation rate $\varepsilon$ is calculated using Eq. (4) [71]:

$$\varepsilon(z) = \frac{(u_{ABL}^*)^3}{k(z + z_0)}$$  

The specific dissipation rate $\omega$ for the SST $k-\omega$ model is calculated using Eq. (5):

$$\omega(z) = \frac{\varepsilon(z)}{C_{\omega}(z)}$$  

where $C_{\omega}$ is an empirical constant taken equal to 0.09. For the LES computations, a time-dependent inlet profile is generated by using the vortex method [72] with a number of vortices $N_v = 190$. As shown by Sergent [72], the influence of small changes in the number of vortices on the generated velocity fluctuations is negligibly small. In addition, this setting was successfully used in previous LES validation studies by the authors for wind flows around buildings [73–76] and indoor airflows [77].

At the ground and building surfaces, either an automated wall treatment is used [78] (for the $k-\omega$ model) or the standard wall functions by Launder and Spalding [79] (for the other models) are used in conjunction with the sand-grain based roughness ($k_S$) modification defined by Cebeci and Bradshaw [80]. The sand-grain roughness height is set to zero ($k_S = 0 \text{ m}$) for the bottom of the domain (to represent a very smooth turntable) and the building surfaces (smooth walls). Fig. 4 shows a check of the horizontal homogeneity for two RANS simulations (RNG $k-\varepsilon$ and SST $k-\omega$ model) and the LES simulation [70]. It compares the dimensionless mean velocity magnitude ($|V/U_H|$) profile at the inlet ($x/D = -3$) with the incident profile at the building location ($x/D = 0$). There is a very small acceleration near the ground due to the absence of a surface roughness. Note however that this acceleration will also be present in the wind tunnel when the air flows over the smooth turntable of the wind tunnel (no roughness elements present at turntable [65]) as also shown in previous experimental studies [81]. The incident profiles obtained with the three other RANS turbulence models are identical to those of the RNG $k-\varepsilon$ and SST $k-\omega$ models and therefore not shown for the sake of brevity. At the outlet plane, zero static gauge pressure is applied and at the top and lateral sides of the domain, zero normal velocities and zero normal gradients of all variables are imposed.

### 3.3. Solver settings

The finite volume method in the commercial CFD code ANSYS Fluent 15 [78] is used to solve the approximate forms of the governing equations. The 3D steady RANS equations are solved in combination with five turbulence models; 4 two-equation eddy-viscosity models and one second-order closure model: (1) standard $k-\varepsilon$ model (SKE) [79]; (2) realizable $k-\varepsilon$ model (RLZ) [82]; (3) RNG $k-\varepsilon$ model [83,84]; (4) SST $k-\omega$ model [85]; (5) low-Re stress-omega RSM model [86]. The SIMPLE algorithm is used for pressure-velocity coupling, pressure interpolation is second order and second-order discretization schemes are used for both the convection terms and the viscous terms of the governing equations. Convergence is assumed to be obtained when all the scaled residuals level off and reach a minimum value. The minimum values of the residuals vary for the different turbulence models, for example the minimum values for the SST $k-\omega$ model are $10^{-7}$ for $x,$
y, z momentum, $10^{-6}$ for k, and $10^{-5}$ for continuity and $\omega$. Note that oscillatory convergence is observed in the simulation with the RSM model, which required averaging over a sufficient number of iterations (e.g. Refs. [48,69]). The LES simulation in this study is performed using the dynamic Smagorinsky subgrid-scale model [87–89]. The filtered momentum equation is discretized with a bounded central-differencing scheme and pressure interpolation is second order. Time integration is second-order implicit. Pressure-velocity coupling is taken care of by the fractional step algorithm. The time step $\Delta t$ is based on a maximum CFL number of 1 and is equal to $\Delta t = 0.0002$ s. The results of the LES computation presented here are averaged over 375,000 time steps, which correspond to a flow time of 75 s, i.e. 60 times the flow-through time (through the computational domain). It is verified that the averaging time is sufficient to obtain statistically-steady results by monitoring the evolution of the mean velocity at several locations inside the enclosure with time (moving average). The simulation is terminated when the moving average of the mean velocity at all location remains within 0.5%.

All simulations are conducted on the HPC cluster (x86_64 architecture) at Eindhoven University of Technology in the Netherlands. For each simulation 14 cores on a node are used (Intel(R) Xeon(R) CPU - X5650 @ 2.67 GHz), and 64 or 96 GB of system memory is available depending on the node used.

3.4. Grid-sensitivity analysis

A grid-sensitivity analysis is conducted for RANS with the SST k-ω model. The coarse, basic and fine grid contain 1,897,416 cells, 5,321,588 cells, and 14,335,040 cells, respectively. 2, 3, and 4 cells are used across the wall thickness for the coarse, basic and fine grid, respectively. Fig. 5 shows the results of the grid-sensitivity analysis by means of the dimensionless mean streamwise velocity ($U_x/U_H$) along three vertical lines in the vertical center plane ($y/W = 0$). The basic and the fine grid provide almost the same results, while the results obtained with the coarse grid show clear deviations with the results of the basic grid at all three lines. Differences between the basic and the fine grid at $x/D = 0.125$ below $z/H = 0.3$ go up to $\Delta(U_x/U_H) = 0.03$.

The grid-convergence index (GCI) as proposed by Roache [90] is used to provide an estimate of the error in $U_x/U_H$ obtained on the basic grid:

$$
GCI_{\text{basic}} = F_s \frac{r^p \left( \left( U_{\text{basic}} - U_{\text{fine}} \right) / U_H \right)}{1 - r^p}
$$

with $r$ the linear grid refinement factor ($r = \sqrt{2}$), $p$ the formal order of accuracy which is assigned a value of 2 due to the second-order discretization schemes used for the simulations, and $F_s$ the safety factor, taken to be 1.25, which is the recommended value when three or more grids are considered in the grid-sensitivity analysis [90]. The GCI provides an estimate of the error in the solution of the basic grid by comparing it to that of the fine grid. The calculated GCI along the three vertical lines is shown in Fig. 5d–f. The average values of GCI along the three lines are: 2.4%, 1.7% and 1.0%, respectively. It is concluded that the basic grid provides nearly grid-independent results and it is therefore used in the remainder of the study for the RANS simulations. The LES simulation is conducted on the coarse grid to limit the computational cost, as explained later. Note that the definition "coarse" with respect to the mesh resolution is a bit misleading, since the minimum cell size is still as small as 0.001 m. 54 x 78 x 56 cells are present inside the enclosure over the building depth, width and height, and the cell count for the building including its direct vicinity (within H) is $\approx 1$ million, resulting in a high-resolution grid in the area of interest.

4. CFD simulations: RANS results

Fig. 6 shows a comparison of x-velocity ($U_x/U_H$) by all five RANS turbulence models with the corresponding measurements along seven vertical lines in the vertical center plane ($y/W = 0$) of the enclosure. The main observations are:

- The different RANS models provide very different results. The RNG, SST and RSM models provide about the same vertical location of maximum $U_x/U_H$ in the jet region along each vertical line. It is remarkable that the RNG and the SST models provide almost identical velocity profiles. On the contrary, the SKE and RLZ models provide a too high location of maximum velocity.
- The RNG, SST and RSM models also provide a larger width of the incoming jet and the related smaller velocity gradients in the shear layer of the incoming jet.
- The RSM model provides substantially higher values for the maximum velocity in the jet region compared to the RNG and...
SST models; e.g. at $x/D = 0.375 UX/U_H = 0.50$ for the RNG and SST models, while $UX/U_H = 0.58$ for the RSM model (Fig. 6c).

- The general flow pattern in the lower part of the enclosure is better reproduced by RNG and SST (better prediction of vertical location of maximum velocity), and to a lesser extent by the RSM model, while the flow in the upper part ($z/H > 0.6$) is more accurately reproduced by SKE, RLZ and RSM.

Fig. 7 displays the contours of the dimensionless mean velocity magnitude ($|V|/U_H$) in the vertical center plane including the jet half-widths $d_{0.5}$ for each model (visualized with white dots) on each side of the jet centerline ($d_{0.5,upper}$ and $d_{0.5,lower}$). The value of $(d_{0.5,upper} + d_{0.5,lower})/H_0$ at $x/D = 0.625$, with $H_0$ the inlet opening height, is also reported. The jet half-width $d_{0.5}$ is defined as the vertical linear distance between the local maximum jet velocity magnitude ($V_{MAX}$) and the location where the velocity magnitude at is equal to half the local jet centerline velocity $V_{MAX}$; i.e. $0.5V_{MAX}$ (see Fig. 7f). Note that due to asymmetry of the incoming jet the jet half-width below the jet centerline ($d_{0.5,lower}$) is not necessarily equal to the one above the jet centerline ($d_{0.5,upper}$). The main observations are:

- The direction of the incoming jet is very different for SKE and RLZ on the one hand, versus RNG, SST and RSM on the other hand. The SKE and RLZ models provide an incoming jet that is more horizontal, in line with the observations from Fig. 6.
- Fig. 7 also clearly shows the larger jet spreading with SKE and RLZ compared to RNG, SST and RSM. The jet spreading parameter $(d_{0.5,upper} + d_{0.5,lower})/H_0$ is largest for RLZ and SKE, with values of 1.7 and 1.6, respectively, followed by SST (1.2), RNG (1.1) and RSM (1.0).
- Fig. 7e shows the higher velocity in the incoming jet predicted by the RSM model, also visible in Fig. 6, which is directly related to the lesser spreading rate.

Fig. 8 provides a comparison of the dimensionless turbulent kinetic energy ($k/U_H^2$) along the seven vertical lines in the vertical center plane. The most important observations are:

- Turbulent kinetic energy is underpredicted in large areas above and below the incoming jet by all RANS turbulence models. This can be explained by the fact that steady RANS does not correctly model the effect of transient flow features on the resolved flow pattern, while the experiments [65] clearly indicate the presence of pronounced transient effects, such as the jet flapping.
Fig. 6. Comparison of dimensionless mean x-velocity ($U_x/U_H$) obtained with steady RANS CFD simulations with different turbulence models (solid and dashed lines), and with wind-tunnel measurements ($\bullet$), along seven vertical lines inside the building in the vertical center plane ($y/W = 0$). (a) $x/D = 0.125$; (b) $x/D = 0.25$; (c) $x/D = 0.375$; (d) $x/D = 0.5$; (e) $x/D = 0.625$; (f) $x/D = 0.75$; (g) $x/D = 0.875$.

Fig. 7. Contours of dimensionless mean velocity magnitude ($|V|/U_H$) obtained with steady RANS CFD simulations with different turbulence models in the vertical center plane ($y/W = 0$). (a) SKE. (b) RNG. (c) RLZ. (d) SST. (e) RSM. (f) Schematic indication of location of maximum jet velocity ($|V|_{\text{MAX}}$, $0.5|V|_{\text{MAX}}$) and jet half-widths $\delta_{0.5,\text{upper}}$, and $\delta_{0.5,\text{lower}}$. The white dots in (a–e) indicate the location of jet half-widths $\delta_{0.5,\text{upper}}$ and $\delta_{0.5,\text{lower}}$. 

T. van Hooff et al. / Building and Environment 114 (2017) 148–165
(vertical motion of the jet) and the Kelvin-Helmholtz type instability in the shear layers of the incoming jet.

- Along the first two vertical lines ($x/D = 0.125$ and $x/D = 0.25$), the RANS models predict values of $k/U_H^2$ around $0.0025$, while the measured values are around $0.015$ and $0.02$, depending on the measurement location.

- Turbulent kinetic energy in the jet region ($0.35 < z/H < 0.6$) is strongly overpredicted by SKE and RLZ at the lines $x/D = 0.125$ and $x/D = 0.25$, which is related to the larger jet spreading observed in Fig. 7a,c.

- RSM predicts the lowest values of turbulent kinetic energy in the jet region at $x/D = 0.125$ and $x/D = 0.25$ and provides a very close agreement with the measurements.

Fig. 9 depicts contours of $k/U_H^2$. The main observations are:

- Turbulent kinetic energy in the incoming jet region is higher with SKE and RLZ (Fig. 9a,c). The higher values seem to originate from outside the enclosure, i.e. from the stagnation region, where a large area with high values of turbulent kinetic energy ($k/U_H^2 > 0.08$) is present. This area with high values of $k/U_H^2$ is much larger for SKE and RLZ than for RNG, SST and RSM. Turbulent kinetic energy is transported to the interior volume and leads to higher indoor values of $k/U_H^2$ for the SKE and RLZ models, especially near the windward opening (see Figs. 8 and 9).

- The higher values obtained with the three $k$-$\varepsilon$ eddy-viscosity RANS models, and especially with SKE and RLZ, are due to the excessive production of turbulent kinetic energy in stagnation zones, i.e. the stagnation point anomaly (e.g. Refs. [17,62,91,92]).

Four validation metrics are used to obtain an overall and quantitative evaluation of the performance of all five RANS turbulence models; the factor of 2 of the observations (FAC2), the factor of 1.3 of the observations (FAC1.3), the fractional bias (FB), and the normalized mean square errors (NMSE). For the streamwise velocity only FAC2 and FAC1.3 are used, while for turbulent kinetic energy FAC2, FB, and NMSE are calculated. The metrics are calculated using Eq. (7) until Eq. (10):

$$\text{FAC}_2 = \frac{1}{N} \sum_{i=1}^{N} n_i \quad \text{with} \quad n_i = \begin{cases} \frac{P_i}{O_i} & \text{for} \quad 0.5 \leq \frac{P_i}{O_i} \leq 2 \\ 0 & \text{else} \end{cases}$$ (7)

$$\text{FAC}_{1.3} = \frac{1}{N} \sum_{i=1}^{N} n_i \quad \text{with} \quad n_i = \begin{cases} \frac{P_i}{O_i} & \text{for} \quad 0.77 \leq \frac{P_i}{O_i} \leq 1.3 \\ 0 & \text{else} \end{cases}$$ (8)
FB = \frac{[O] - [P]}{0.5([O] + [P])} \tag{9}

NMSE = \frac{\left([O] - [P]\right)^2}{\left([O]\right)^2} \tag{10}

with \([P]\) the predicted (CFD) value, \([O]\) the observed (measured) value and \(n\) the number of measurement locations, which is equal to 63 (7 x 9). The square brackets indicate averaging over all data points. The results are shown in Table 2, with an indication of the ideal values at the bottom row; i.e. 1 for FAC2 and FAC1.3, 0 for FB and 0 for NMSE.

FAC2 for \(U_X/U_H\) for all RANS models lies between 0.52 and 0.63, with the highest value for SKE (0.63), followed by RSM (0.59), and the lowest value for RLZ (0.52). However, the calculation of FAC1.3 results in a different ranking with the best performance with SKE (0.33), closely followed by SST (0.32), and then by RNG (0.27), RSM (0.25), RLZ (0.24). For turbulent kinetic energy, the FAC2 values are highest for SKE (0.52) and lowest for RSM (0.32). The same holds for FB and NMSE; for both metrics SKE shows the best performance and RSM the worst performance. The other three models score very differently in the ranking for each of the three validation metrics used; i.e. RLZ scores better than the RNG and SST for FAC2 and FB, while it scores worse than RNG and SST for NMSE. The results from Table 2 might seem to differ from the findings extracted from Fig. 6, however, it must be noted that in the calculation of the validation metrics all locations are given the same weight, where the locations in the jet region—often regarded as the most important region—are considered equally important as the locations above and below the jet region. The overall best performance of SKE compared to the more sophisticated models is noteworthy. It is attributed to the observed overprediction of turbulent kinetic energy in the stagnation region outside the building (Fig. 9), which counteracts the lack of turbulence above and below the incoming jet region due to the use of the steady RANS approach. Since the excessive generation of turbulent kinetic energy was most pronounced for SKE (see Fig. 9), the results obtained with this model show a good agreement with experimental results, especially above and below the incoming jet region.

Finally, the volume flow rates through the building are compared with the experimentally obtained value [65]. The volume flow rate is made dimensionless using the velocity at building height (\(U_H\)) and the window area (\(A_{\text{inlet}}\)); i.e. \(Q/U_H A_{\text{inlet}}\) [65]. Table 3 shows the values obtained from the steady RANS simulations and

Table 2

<table>
<thead>
<tr>
<th>Method</th>
<th>Turbulence model</th>
<th>(U_X/U_H)</th>
<th>(k/U_H^2)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>FAC2</td>
<td>FAC1.3</td>
<td>FAC2</td>
</tr>
<tr>
<td>RANS</td>
<td>SKE</td>
<td>0.63</td>
<td>0.33</td>
</tr>
<tr>
<td></td>
<td>RNG</td>
<td>0.56</td>
<td>0.27</td>
</tr>
<tr>
<td></td>
<td>RLZ</td>
<td>0.52</td>
<td>0.24</td>
</tr>
<tr>
<td></td>
<td>SST</td>
<td>0.56</td>
<td>0.32</td>
</tr>
<tr>
<td></td>
<td>RSM</td>
<td>0.59</td>
<td>0.25</td>
</tr>
<tr>
<td>LES</td>
<td>Dynamic Smagorinsky</td>
<td>0.83</td>
<td>0.48</td>
</tr>
<tr>
<td>Ideal values</td>
<td></td>
<td>1</td>
<td>1</td>
</tr>
</tbody>
</table>

Table 3

<table>
<thead>
<tr>
<th>Method</th>
<th>Turbulence model</th>
<th>(Q/U_H A_{\text{inlet}}) [-]</th>
<th>Deviation [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Experiments</td>
<td>0.50</td>
<td></td>
<td></td>
</tr>
<tr>
<td>RANS</td>
<td>SKE</td>
<td>0.483</td>
<td>3.5</td>
</tr>
<tr>
<td></td>
<td>RNG</td>
<td>0.506</td>
<td>1.2</td>
</tr>
<tr>
<td></td>
<td>RLZ</td>
<td>0.503</td>
<td>0.6</td>
</tr>
<tr>
<td></td>
<td>SST</td>
<td>0.516</td>
<td>3.1</td>
</tr>
<tr>
<td></td>
<td>RSM</td>
<td>0.543</td>
<td>8.6</td>
</tr>
<tr>
<td>LES Dynamic Smagorinsky</td>
<td>0.538</td>
<td>7.6</td>
<td></td>
</tr>
</tbody>
</table>
the experimental one \((Q/U_{inlet} = 0.5)\). The largest deviation is present for RSM (8.6%) followed by SKE and SST, with deviations of 3.5% and 3.1%, respectively. The best agreement is provided by RLZ (0.6% deviation) followed by RNG (1.2% deviation).

5. CFD simulations: LES results

Figs. 10–13 show the dimensionless mean velocity and turbulent kinetic energy obtained by LES using the dynamic Smagorinsky subgrid-scale model on the coarse grid. The results of SKE and SST
on the basic grid are added to the figures for comparison. The following main observations are made for Fig. 10 (Ux/UH in the vertical center plane):

- A very good agreement is present between LES and experiments, clearly better than the best performing RANS models (SKE and SST).

- Especially in the areas below and above the incoming jet, LES outperforms RANS. For example, at x/D = 0.5 (Fig. 10d) the LES results closely follow the measured values below z/H = 0.3 and above z/H = 0.5, while clear discrepancies are present at these locations between RANS and the experimental results. Also further downstream, at x/D = 0.625, x/D = 0.75 and x/D = 0.875, the results obtained with LES are much better, with the
strongest improvements above the incoming jet for RANS SST and below the jet for RANS SKE.

- In general, the velocity gradients are smaller in the profiles obtained with LES compared to RANS SST, which can be attributed to the transient flow features that are captured with LES and which lead to a stronger redistribution of momentum and turbulence levels. The smaller velocity gradients and the broader jet (more spread) are for example clearly visible in Fig. 10d. The velocity gradients with RANS SKE are similar to those of LES, however, in LES they are the result of an overestimation of turbulent kinetic energy in the stagnation region outside the enclosure.

Fig. 11 shows contours of $|V/U_H|$ in the vertical center plane. The most important observations are:

- The LES results provide a slightly higher velocity magnitude in the incoming jet compared to RANS.
- The incoming jet is slightly more horizontal in LES than with RANS SST.
- The spreading of the jet is more pronounced in LES. For example, at $x/D = 0.625$ the jet predicted by LES is wider compared to the jet predicted by SST, as quantified in Fig. 10. The calculated spreading of the incoming jet in the LES simulation based on the jet half-widths $(D_{0.5,\text{upper}} + D_{0.5,\text{lower}})/H_0$ at $x/D = 0.625$ is about equal to those predicted by SKE and RLZ, i.e. $(D_{0.5,\text{upper}} + D_{0.5,\text{lower}})/H_0 = 1.7$ (see Fig. 7a,c).

Fig. 12 shows values of $k/U_H^2$ for the following main observations are made:

- The turbulent kinetic energy levels above and below the jet are generally much higher in the LES simulation compared to RANS and generally show a better agreement with the experimentally obtained values.
- LES overpredicts turbulent kinetic energy in the jet region at $x/D = 0.25$ until $x/D = 0.875$.
- At $x/D = 0.125$ and $x/D = 0.25$, LES predicts a turbulent kinetic profile with two clear peaks in the shear layer of the incoming jet (at $z/H = 0.38$ and $z/H = 0.55$ at $x/D = 0.125$; at $z/H = 0.35$ and $z/H = 0.5$ at $x/D = 0.25$), which are not visible in both RANS results.

Fig. 13 shows contours of $k/U_H^2$ by RANS SKE, RANS SST and LES in the vertical center plane. The following main observations are made:

- As opposed to SKE and SST, LES provides a region of high turbulence in front of the building, which can be attributed to the standing vortex in front of the building which is a highly transient flow feature (moving back and forward along the ground surface).
- The interior distribution of $k/U_H^2$ is very different. As indicated before, RANS SKE predicts very high values near the windward window opening due to the overprediction of turbulent kinetic energy in the stagnation zone outside the enclosure. RANS SST mainly predicts higher indoor values near the upstream window due to transport of turbulent kinetic energy from the stagnation zone outdoors to the indoor environment and in the shear layer of the incoming jet. LES on the other hand provides higher values downstream of the opening and in a far larger region around the jet.

Also for the LES results the agreement with the experimental data is quantified using four validation metrics (Eqs. (7)–(10)), Table 2 shows that the agreement between LES and the measurement data is much better than for the RANS simulations. The FAC2 and FAC1.3 values for the x-velocities are equal to 0.83 and 0.48, and FAC2 is equal to 0.95 for turbulent kinetic energy, while the highest FAC2 value for turbulent kinetic energy for the RANS was 0.52. Also the FB and NMSE scores are clearly better for LES: $-0.2$ and 0.3, respectively.

Table 3 shows the mean volume flow rate through the building predicted by LES, together with the steady RANS results. The volume flow rate deviates with 7.6% from the experimentally obtained value. This deviation is larger than the deviation with the best performing steady RANS model; i.e. RLZ (deviation of 0.6%). Note that the use of tracer gas measurements to obtain volume flow rates has its limitations with respect to accuracy and it is therefore difficult to draw strong conclusions on the performance of each of the tested models.

6. Comparison RANS vs. LES

The results from Section 4 (RANS) and Section 5 (LES) show clear differences in the predicted flow fields with respect to mean velocity, turbulent kinetic energy and general flow pattern. In this section, a more detailed analysis is provided with respect to the observed differences and their physical background.

Comparison of Figs. 9 and 13c shows that the interior distribution of turbulent kinetic energy obtained with LES is clearly different from that of the steady RANS simulations. There is a different origin of turbulence inside the enclosure between RANS and LES. In the case of SKE and RLZ, the indoor turbulent kinetic energy levels can largely be attributed to the excessive generation of turbulent kinetic energy in the stagnation zone outside the enclosure (Fig. 9a,c), which is subsequently transported indoors. In LES, this excessive generation of turbulent kinetic energy in the stagnation region is not present at all (Fig. 13c) and the increased levels of turbulence are the result of velocity gradients and local unsteadiness of the flow inside the enclosure; i.e. the jet flapping (vertical motion of the jet) and the instabilities in the shear layer of the jet flow. To illustrate the presence of one of the unsteady flow features inside the enclosure (i.e. vertical jet flapping), Fig. 14 shows instantaneous images from the flow visualizations and the LES simulation. Vertical jet flapping is present in both the experiments and the LES simulation; the jet direction varies considerably in time, from an upward direction over a nearly horizontal direction with a small downward bend when the jet enters the enclosure to a downward direction. The absence of this distinct vertical flapping motion in the steady RANS simulations results in deviations of the simulation results when compared to the measurement data. Fig. 14a–c also shows Kelvin-Helmholtz type instabilities in the shear layer of the incoming jet, which is not visible in the LES simulation. The accurate prediction of this type of instability in the shear layer requires a much higher grid resolution, which for the present study exceeded the available computational resources. The computational cost for the LES simulation on the coarse grid is already about 80–100 times higher than for steady RANS on the basic grid. LES on the basic grid would add a factor of 2–4, resulting in 160–400 times the computation time of a steady RANS simulation on the same grid.

Fig. 15a shows the predicted jet trajectory for all evaluated models based on the jet centerline velocity (i.e. $V_{\text{max}}$). The symbols represent the vertical location ($z_{\text{max}}$) of the local maximum jet velocity magnitude ($V_{\text{max}}$) along all seven vertical lines inside the enclosure. There is no RANS model that predicts the same trajectory as the LES model; SKE and RLZ predict a more horizontal jet direction (as discussed earlier), while RNG, SST and RSM predict a more downward direction of the incoming jet (Fig. 15a).
and Eq. (11):

\[ \alpha = \tan^{-1} \left( \frac{Z_{\text{MAX},X/D=0} - Z_{\text{MAX},X/D=0.5}}{x} \right) \]  

(11)

The value of \( \alpha \) ranges from 13.6° to 24.6° for the five RANS models and the LES model. The SST, RSM, RNG models predict a value for \( \alpha \) (22.7°–24.6°) close to that of the LES model (\( \alpha = 23.0^\circ \)). SKE and RLZ provide low values of \( \alpha \) due to the more horizontal jet direction. Note that \( \alpha \) not only depends on the location of \( V_{\text{MAX}} \) at \( x/D = 0 \) as depicted in Fig. 15a, but also on the location of \( V_{\text{MAX}} \) at \( x/D = 0.5 \), which differs considerably for all models due to different velocity distributions over the height of the windward window opening.

Fig. 15c compares \( U_{\text{H}/U_{\text{H}}} \) obtained with RANS and LES with the measurement data along a horizontal line through the middle of the two window openings. The RANS models either underpredict or overpredict the mean velocity along this line, which can directly be attributed to the jet being too horizontal (SKE and RLZ) or too inclined (RNG, SST, RSM) compared to the measurements. The results of the LES simulation provide a good agreement with the measurement data, with an overprediction of \( U_{\text{H}/U_{\text{H}}} \) along the line.

Fig. 15d shows \( k/U_{\text{H}}^2 \) along the same horizontal line (Fig. 15e). LES overpredicts the values of \( k/U_{\text{H}}^2 \) in the middle of the enclosure (0.25 < \( x/D < 0.875 \)), while RNG, SST and RSM underpredict \( k/U_{\text{H}}^2 \) along this line. SKE and RLZ strongly overpredict \( k/U_{\text{H}}^2 \) in the upstream part of the enclosure (\( x/D < 0.25 \)), but subsequently predict similar levels of \( k/U_{\text{H}}^2 \) as measured in the wind tunnel between 0.375 < \( x/D < 0.875 \).

7. Discussion

7.1. Comparison with other studies

A comparison of the performance of the five different RANS turbulence models is not straightforward due to the different assessment criteria that can be used; e.g. different validation metrics, emphasis on different parts of the flow (e.g. jet region vs. entire enclosure), emphasis on one parameter over another (e.g. velocity vs. turbulence levels), etc. A comparison with conclusions from previous validation studies from literature also becomes difficult due to the issues mentioned above and it can be complicated even more due to different geometries used, different locations at which velocities and turbulence levels are compared, etc.

Despite all difficulties mentioned above, the performance of different turbulence models in predicting volume flow rates as found in literature is compared with the performance found in the current study. Table 3 showed that all models predict volume flow rates within 8.6% of the measured value, with the RLZ model showing the best performance (0.6% deviation). One must note that the tracer gas method has its limitations with respect to accuracy and that strong conclusions are difficult to make. In literature, Straw et al. [4] for example found higher differences in volume flow rate using RANS with RNG (3–9%), however, they did not test other turbulence models in their study. Evola and Popov [30] found deviations of 8.7% for RNG and 13.7% for SKE when they compared steady RANS CFD with wind-tunnel measurements of the volume flow rate by Jiang et al. [6]. The better performance of RNG compared to SKE in Evola and Popov [30] is found in the current study as well. Hu et al. [26] found deviations within ≈ 12% for each wind direction with an average deviation of 5%, using SST, which is higher than the deviation in the current study (3.1%). As mentioned before, the results from different studies are very difficult to compare, due to the different experimental techniques used, the wide range of computational settings that all affect the outcome of the simulations, etc.

7.2. Limitations and future work

This paper presented the results of a validation study of cross-ventilation flow in a generic enclosure. Both RANS and LES simulations were compared with experimental data from wind-tunnel measurements by Tominaga and Blocken [65]. The following points of discussion and of future work can be raised:

- Due to computational limitations in the present study it was not possible to run a LES simulation on the basic grid. Future research might include this simulation and also include a more detailed parameter study for LES for cross-ventilation flow. For example, a more uniform grid distribution could be tested, as

![Fig. 14. Comparison of instantaneous images from flow visualization [65] (a–c) with images from LES simulation (d–f). The figures indicate the observation and prediction of jet flapping inside the enclosure. (a,d) Upwards directed jet (bended jet). (b,e) Horizontally directed jet with small downward bend after jet entry. (c,f) Downward directed jet.](image-url)
as other subgrid-scale models, inflow turbulence generators, etc. The use of the coarse grid among others resulted in the absence of Kelvin-Helmholtz type instabilities in the LES simulation, while this phenomenon could be observed in the flow visualizations by Tominaga and Blocken [65]. Note that the grid-resolution for the “coarse” grid was still very high with around 1.5 million hexahedral cells for the building and its direct vicinity (within H of the building) and that LES on this grid provided a very good agreement with the experimental results.

- Future work will focus on the use of unsteady RANS (URANS) to capture the flow field in the cross-ventilated building. Although the use of URANS is not straightforward, past research has also shown the potential of URANS with respect to obtaining the unsteady motion of the flow (e.g. Ref. [93]), and it is worthwhile to study it since the computational demand is less than that of LES.

- In this study, five commonly used turbulence models were included. The study can be extended by including for example other second-order closure models, eddy-viscosity models, or hybrid RANS/LES models (e.g. Ref. [94]).

8. Conclusions

The objective of the study presented is the validation of cross-ventilation flow through a generic enclosure for five different RANS turbulence models and for a LES model in combination with
the dynamic Smagorinsky subgrid-scale model. A comparison with a range of RANS models as well as LES is not common, as can be seen in the overview of validation studies earlier in this introduction section, although such comparisons are of primary importance when deciding which numerical method to use and to aid in drafting guidelines on the use of CFD for the analysis and optimization of cross-ventilated buildings. Mean velocity, turbulent kinetic energy, and volume flow rate are compared with experimental data from literature. In addition, an evaluation of the jet half-widths ($d_{50,\text{upper}}$ and $d_{50,\text{lower}}$), incoming jet angle ($\alpha$) and transient flow features (i.e. jet flapping) is presented, which is regarded as an innovative aspect compared to previous studies from literature. From this study, the following main conclusions can be made:

- The five different steady RANS models provide very different results. The RNG, SST and RSM models provide about the same location of maximum velocity (and thus jet direction) along each vertical line. SKE and RILZ provide a too high location of maximum velocity (too horizontal direction compared to experiments). In addition, SKE and RILZ predict a wider range of the incoming jet (larger value for $d_{50,\text{upper}}$ and $d_{50,\text{lower}}$) and smaller velocity gradients in the shear layer of the incoming jet than RANS, SST and RSM.

- The general flow pattern in the lower part of the enclosure is better reproduced by RNG and SST, and to a lesser extent by SKE, while the flow in the upper part is more accurately reproduced by SKE, RILZ and RSM.

- Turbulent kinetic energy is overpredicted in large areas above and below the incoming jet by all five RANS turbulence models. SKE and RILZ predict too high levels of turbulent kinetic energy in the incoming jet region, which originate from outside the enclosure (stagnation region). This stagnation region with high values of $k/UH$ is much larger for SKE and RILZ than for RNG, SST and RSM.

- SKE provides the best agreement with the experiments in terms of the validation metrics (FAC2, FAC1.3, FB, NMSE) for $x$-velocity and turbulent kinetic energy; the worst agreement is obtained with RILZ and RSM.

- The LES simulation provides a very good agreement with the experimental data, both with respect to the mean velocities, flow pattern (jet direction), and turbulent kinetic energy, although the latter is overestimated in the jet region in the LES simulation.

- LES performs better than the steady RANS models, which is supported by the results of the four validation metrics (FAC2, FAC1.3, FB, NMSE). The better performance of LES can largely be attributed to the unsteady flow features (e.g. vertical jet flapping) present in the flow, as also observed in the experiments, which is reproduced by the LES model and not by the steady RANS models.

- The use of LES entails an increase in computational demand with a factor of $\approx 80$–100, which is a strong increase, especially when considering that LES is performed on the coarse grid and RANS on the basic grid. The majority of the RANS simulations took a bit less than 12 h, while the LES simulation lasted about 40 days.

**Acknowledgements**

Twan van Hooff is currently a postdoctoral fellow of the Research Foundation Flanders (FWO) and acknowledges its financial support (project FWO 12B9715N). The authors also gratefully acknowledge the academic partnership with ANSYS CFD.

**References**


T. van Hooff et al. / Building and Environment 114 (2017) 148–165


[70] D.K. Lilly, A proposed modification of the Germano subgrid-scale closure...


