Computers & Fluids 79 (2013) 120–133

Contents lists available at SciVerse ScienceDirect

Computers & Fluids

journal homepage: www.elsevier.com/locate/compfluid

# Quality assessment of Large-Eddy Simulation of wind flow around a high-rise building: Validation and solution verification

P. Gousseau [a], B. Blocken [a,*], G.J.F. van Heijst [b]

[a] Building Physics and Services, Department of the Built Environment, Eindhoven University of Technology, P.O. Box 513, 5600 MB Eindhoven, The Netherlands
[b] Fluid Dynamics Laboratory, Department of Applied Physics, Eindhoven University of Technology, P.O. Box 513, 5600 MB Eindhoven, The Netherlands

ARTICLE INFO

*Article history:*
Received 25 June 2012
Received in revised form 16 January 2013
Available online 19 March 2013

*Keywords:*
Computational Fluid Dynamics (CFD)
Validation and Verification (V&V)
Urban wind flow
Large-Eddy Simulation (LES)
Turbulence modeling
Bluff body aerodynamics

ABSTRACT

When undertaking wind engineering problems such as urban pollutant dispersion or pedestrian wind comfort with Computational Fluid Dynamics, an accurate simulation of the flow-field around buildings is required. In this respect, the good performance of Large-Eddy Simulation has already been established but because the formulation and the use of this turbulence modeling approach are complex, the uncertainty on the results is relatively high. This implies the need for Validation and Verification (V&V) studies like the one performed in the present paper for the wind flow around an isolated high-rise building with aspect ratio 1:1:2. In the first part of the study, the numerical results are compared with measurements from a reference wind-tunnel experiment and the agreement is quantified by validation metrics. The vortex method to generate inflow turbulence is shown to provide accurate results. Unexpectedly, the best agreement with the experiments is obtained on the coarsest computational grid, with 20 cells per building side, while a finer grid with 30 cells per building side over-estimates the turbulent kinetic energy measurements. A similar result was also found by earlier studies for different flow configurations. In the second part of the study, solution verification is performed. The Systematic Grid and Model Variation technique is used to provide estimates of the modeling and numerical error contributions. The *LES_IQ* indicator shows that a grid with 20 (resp. 30) cells per building side allows resolving 80% (resp. 91%) of the total turbulent kinetic energy in the region around the building.

© 2013 Elsevier Ltd. All rights reserved.

## 1. Introduction

Computational Fluid Dynamics (CFD) is increasingly used to solve wind engineering problems such as pollutant dispersion in the built environment, pedestrian wind comfort, wind loads on buildings or natural ventilation of buildings [22,63,36,58,7,40]. In all these cases, an accurate simulation of the wind flow around buildings by the CFD model is needed. This is the reason why – supported by the increase of computational power – the use of the Large-Eddy Simulation (LES) turbulence modeling approach is nowadays becoming more widespread in Computational Wind Engineering (CWE). Several earlier studies [35,42,46,51] have indeed demonstrated that LES can provide an accurate description of the mean and instantaneous flow-field around bluff bodies like buildings. In general, it performs better than the Reynolds-Averaged Navier–Stokes (RANS) turbulence modeling approach, at the expense of much larger requirements in terms of computational resources.

Most of the aforementioned studies have established the good performance of LES based on comparison of the numerical results with measurements, often provided by wind-tunnel experiments. However, despite the increasing attention given to the quantification of error and uncertainty in CFD, the techniques that have been developed for general fluid engineering problems to assess the quality of CFD simulations are still marginally used in CWE [13]. This is particularly true for LES.

The aim of the present study is to provide a Validation and Verification (V&V) study of the LES computation of wind flow around an isolated building. To the best of the authors' knowledge, this V&V strategy that has been developed for general fluid engineering problems has not yet been applied to such a flow.

Validation is defined as "the process of determining the degree to which a model is an accurate representation of the real world from the perspectives of the intended uses of the model" [2]. It will be performed here by comparing the numerical results with the measurements from a reference wind-tunnel experiment and by quantifying the agreement with validation metrics (Section 4). The influence of the subgrid-scale (SGS) model and grid resolution will be assessed. In particular, the results of simulations without SGS model, with the standard Smagorinsky SGS model and with its dynamic version will be compared. For the standard

* Corresponding author. Tel.: +31 (0)40 247 2138; fax: +31 (0)40 243 8595.
*E-mail addresses:* pierre.gousseau@gmail.com (P. Gousseau), b.j.e.blocken@tue.nl (B. Blocken), g.j.f.v.heijst@tue.nl (G.J.F. van Heijst).

0045-7930/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.compfluid.2013.03.006

Smagorinsky model, an appropriate value for the Smagorinsky coefficient will be determined in what is usually referred to as "calibration" in the V&V process [2].

Verification is defined as "the process of determining that a model implementation accurately represents the developer's conceptual description of the model and the solution to the model" [2]. Note that other definitions for the terms "validation" and "verification" can be found for example in [8] or [48]. The process of verification is twofold: on the one hand the code verification and on the other hand the solution verification [2,41,37]. The former will not be treated here: the CFD code used is a commercial code (Ansys/Fluent 12.1) and is assumed to be verified in the development process. The solution verification will be performed in four steps:

(1) Evaluating the turbulent inflow generation technique. Here, the Vortex Method (VM) [45,31] is used. Besides testing the influence of inflow turbulence on the flow field around the building (validation), a posteriori verification will be performed indicating that the mean inflow is a good representation of the experimental one (Section 4.1).
(2) Assessing the statistical convergence of the numerical solution. The LES results are compared to the measurements in terms of mean values. It will be verified that the first moments of velocity are sufficiently converged (Section 5.1).
(3) Evaluating the modeling and numerical error contributions in the LES solution. For basic flows at low Reynolds numbers, this can be achieved using Direct Numerical Simulation (DNS) results [60,17,33]. However, in the present study the high Reynolds number of the flow prohibits the application of DNS so a multi-grid technique is used: the Systematic Grid and Model Variation (SGMV) [25,14,26,11] (Section 5.2).
(4) Evaluating the proportion of the total turbulent kinetic energy which is resolved by the LES model with the LES Index of Quality (*LES_IQ*) [9,10,11] (Section 5.3).

The reference experiment that will be reproduced with CFD is described in the next section. Next, the computational model is outlined, before presenting and analyzing the results.

## 2. Description of the experiment

The wind-tunnel experiment by Meng and Hibi [32] is used as a validation experiment. A building with dimensions $b \times b \times h$ ($b = h/2 = 0.08$ m) in the streamwise ($x$), lateral ($y$) and vertical ($z$) direction, respectively, is placed in the test section of a wind tunnel where an Atmospheric Boundary Layer (ABL) flow is simulated. The Reynolds number based on $b$ and the mean velocity of the incident flow at building height ($U_h$) is equal to $2.4 \times 10^4$. The origin of the coordinate system is the center of the building's ground face. The streamwise turbulence intensity at $z/b = 0.125$, 2 and 7.5 is equal to 22.8%, 18% and 4.5%, respectively. The undisturbed ABL profiles of mean streamwise velocity ($U = \langle u \rangle$), standard deviation of velocity in the three directions ($\sigma_u$, $\sigma_v$, $\sigma_w$) and shear stress ($-\langle u'w' \rangle$, where $u'_i = u_i - U_i$ denotes the fluctuation of the velocity in the direction $x_i$) are provided in the experimental report.

The mean ($U$, $V$, $W$) and standard deviation of the three velocity components have been measured with a constant-temperature anemometer with split-fiber probe at 186 points around the building. 66 of these points are in the vertical mid-plane $y/b = 0$, hereafter denoted by V0. Two horizontal planes at $z = 1$ cm (H1; $z/b = 0.125$) and $z = 10$ cm (H10; $z/b = 1.25$) contain 60 additional measurement points each. In each plane, the points are distributed along nine lines at $x/b = -0.75$; $-0.5$; $-0.25$; 0; 0.5; 0.75; 1.25; 2; 3.25. Because of space limitations, the graphical comparison (profiles) of experimental and numerical data will be performed only in the planes V0 and H10 for a limited number of points (5 out of 9 measurement lines per plane) and variables ($U$ and the turbulent kinetic energy $k = 0.5 \times (\sigma_u^2 + \sigma_v^2 + \sigma_w^2)$). The validation metrics, however, take into account all the data points. Note that this experiment has been reproduced with CFD before by Tominaga et al. [51]; their LES results will also be used in our study for comparison purposes.

## 3. Computational model

### 3.1. LES modeling

The commercial CFD code Ansys Fluent 12.1 is used here, with LES as a turbulence modeling approach. The filtered incompressible Navier–Stokes equations are given by:

$$\frac{\partial \overline{u}_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial \overline{u}_i}{\partial t} + \overline{u}_j \frac{\partial \overline{u}_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\nu \frac{\partial \overline{u}_i}{\partial x_j}\right) - \frac{\partial \tau_{ij}}{\partial x_j} \tag{2}$$

where the overbar denotes the filtering operator (with filter width equal to grid size), $\rho$ and $\nu$ are the air density and kinematic viscosity, respectively, $p$ the pressure and $\tau_{ij}$ the components of the SGS stress tensor:

$$\tau_{ij} = \overline{u_i u_j} - \overline{u}_i \overline{u}_j \tag{3}$$

Two simulations will be run without any subgrid-scale model, i.e. by omitting the last term on the right-hand side of Eq. (2). The intention is to observe whether the dissipation due to the numerical scheme used here can mimic the effect of the smallest scales of motion. For the other simulations, the Smagorinsky SGS model [47] is applied to close the system of equations and determine the SGS stresses via the SGS turbulent viscosity $\nu_{SGS}$ and the filtered rate of strain $\overline{S}_{ij} = (\partial \overline{u}_i/\partial x_j + \partial \overline{u}_j/\partial x_i)/2$:

$$\tau_{ij} - \frac{1}{3}\tau_{kk}\delta_{ij} = -2\nu_{SGS}\overline{S}_{ij} \tag{4}$$

with:

$$\nu_{SGS} = L_{SGS}^2 \overline{S} \tag{5}$$

where $\overline{S} = (2\overline{S}_{ij}\overline{S}_{ij})^{1/2}$ is the characteristic filtered rate of strain and $L_{SGS} = \min(\kappa d, C_s V_c^{1/3})$ is the SGS mixing length, with $\kappa$ the von Karman constant, $d$ the distance to the closest wall, $V_c$ the volume of the computational cell and $C_s$ the Smagorinsky coefficient. Note that Eq. (2) corresponds to the momentum equation filtered with a uniform filter width and the commutation error that arises when filtering the equation on a non-uniform grid is neglected [34,18,59].

The distinction is made here between the so-called standard Smagorinsky model, where $C_s$ is a user-prescribed constant, and the dynamic version [16,28], where $C_s$ is computed at each time step with a test-filter (whose width is twice the grid size) and clipped to the range [0; 0.23] to avoid numerical instabilities. The upper bound of this range aims at preventing the appearance of extremely high $C_s$ values which, on the one hand, are not physical and, on the other hand, can lead to high spatial variations of $C_s$ and destabilize the solver. The imposed maximum value for $C_s$ ($C_{s,max}$) should be high enough to allow the description of all types of flow, but the particular value imposed is different in each CFD code, showing that there is no widely-accepted value for $C_{s,max}$. Here, $C_{s,max} = 0.23$ is used, which is the default value in Ansys Fluent 12.1 [3]. The two versions of the Smagorinsky model will be used and compared in the present study. For the standard version,

Download English Version:

https://daneshyari.com/en/article/7157532

Download Persian Version:

https://daneshyari.com/article/7157532

Daneshyari.com