August 27-31, 2023
FLORENCE - ITALY
ICWE 16
16th International
Conference on
Wind Engineering

Inflow condition effects on large-eddy simulation of flow around an isolated building

Usman Shaukat[1] and Knut Erik Teigen Giljarhus[1]

[1] *University of Stavanger, Department of Mechanical and Structural Engineering and Materials Science, Stavanger, Norway, usman.shaukat@uis.no*

SUMMARY:
In wind engineering simulations, large eddy simulations can generate more reliable results than simulations based on Reynolds-averaged Navier-Stokes models. However, the application of inflow boundary conditions at the computational domain inlet is one of the challenges in large eddy simulations in urban wind engineering applications. This study applies large eddy simulation to a 1:1:2 single isolated building, with the purpose of investigating various inflow conditions and their effects on the turbulent statistics of the wind flow field around the building. Three methods are used to produce inflows at the boundary inlet: precursor simulation, digital filter method to generate artificial turbulence, and mean velocity profile without turbulence. Overall, precursor simulation results are closest to experiments compared to other methods. However, inflow conditions have minimal impact on mean flow and turbulent kinetic energy downstream of the building at pedestrian height. The results for the digital filter method and mean velocity profile without turbulence cases become less accurate away from the building.

Keywords: Large-eddy simulations, inflow conditions, wind engineering

1. INTRODUCTION

The increase in computational power in recent years has shifted the researchers' focus towards predicting the flow field using large-eddy simulations (LES) as it has higher accuracy than Reynolds-averaged Navier-Stokes (RANS) equations in urban environments (Blocken, 2018). LES simulations have been used in numerous studies to investigate the flow field in the urban environment for problems such as pollutant dispersion and pedestrian wind comfort (Gousseau et al., 2011; Liu et al., 2019).

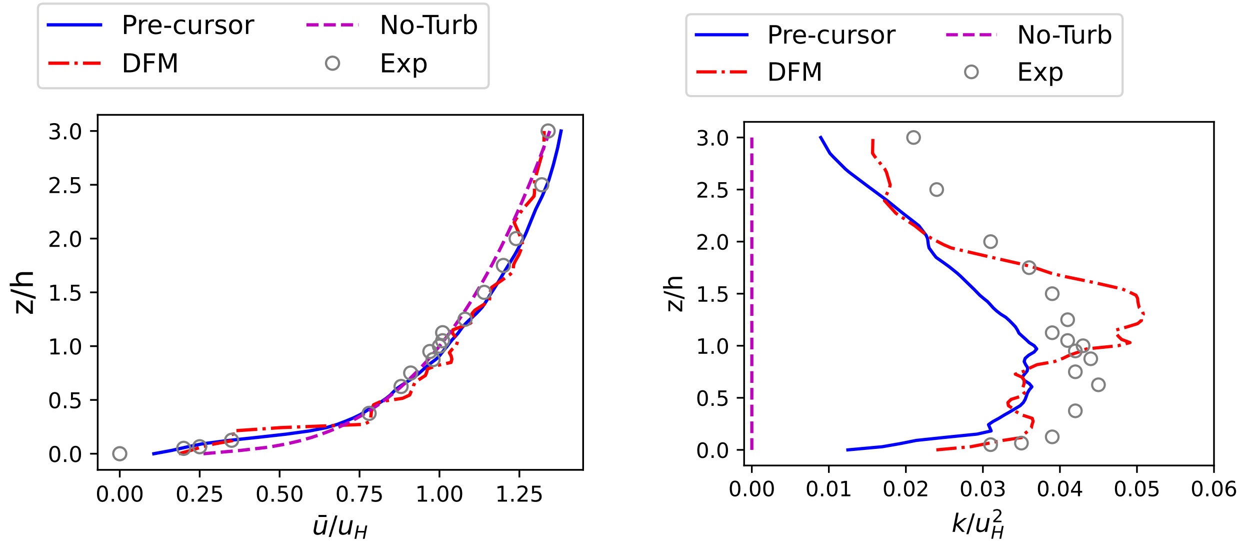
One of the main challenges for LES simulation is inflow boundary conditions (Blocken, 2018).In previous studies, results of LES performed with artificially generated turbulent inflows have shown an advantage over inflow without perturbation for isolated buildings (Gousseau et al., 2013; Tominaga et al., 2008a). Various methods have been used to generate inflows with perturbation: vortex generation, digital filter method (DFM), and precursor simulations to analyze flow fields in urban applications (Gousseau et al., 2013; Jia and Kikumoto, 2022; Okaze et al., 2021). According to the author's knowledge, the impact of different inflow generation methods on to flow field around the building has not been conducted in detail.

The overall effect of inlet conditions may become negligible in the urban environment when the wind is subjected to various local obstacles generating turbulence. Thus, understanding the impact of inflow generation in LES to wind flow around a single isolated building will help to ease the setup of wind simulations in urban environments among CFD users.

In this study, LES simulation of an isolated building with shape 1 (length):1 (width):2 (height) is conducted using three different inflow generation methods. Inflows are generated using precursor simulation, artificial turbulence generation using the DFM (Klein et al., 2003), and mean velocity profile without turbulence. LES results are also compared to RANS and wind tunnel experiments.

2. METHODOLOGY

Simulations are performed in OpenFoam v2206. Data for results validation are obtained from the wind tunnel experiment performed at the Architectural Institute of Japan (AIJ) (Okaze et al., 2021).



(a) *Normalized mean velocity, u/u_H, at $x = -2.5H$.*

(b) *Normalized turbulent kinetic energy, u/u_H, at $x = -2.5H$.*

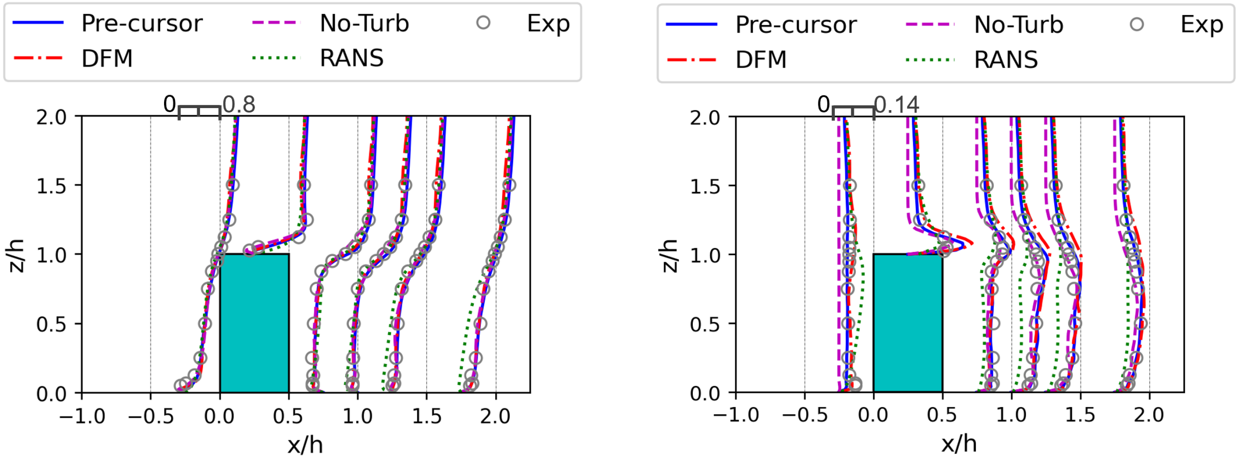
Figure 1. Shows comparison of inflow profile of precursor simulations (blue line), DFM method (red line), mean profile without turbulence (magenta line) and experimental profile (grey circles) at plane $y = 0$.

The computational domain is created according to the guidelines (Franke et al., 2007; Okaze et al., 2021; Tominaga et al., 2008b). It has a dimension of $11H(x) \times 6H(y) \times 5H(z)$, with a distance between the inlet and the upwind facade equal to $2.5H$, where $H = 0.2\,\mathrm{m}$ is the building height. The minimum grid spacing is $H/40$ next to the building, stretching away from the building with an expansion ratio of 1.08 (Okaze et al., 2021).

Three different inflow conditions are applied at the inlet. The mean streamwise velocity at building height is $u_H = 4.7\,\mathrm{m/s}$. For the first case, velocity data with a frequency of $1\,\mathrm{kHz}$ for precursor simulation is obtained from the AIJ (Okaze et al., 2021). For the second case, DFM is used to generate artificial turbulence (Klein et al., 2003). In this method, first- and second-order statistics at each point are specified to generate turbulent velocity data for LES. Information, such as the Reynolds stress tensor, length scale, and mean velocity, are provided as input. These values are approximated from experimental measurements. Finally, the mean velocity profile without turbulence is implemented for the last case.

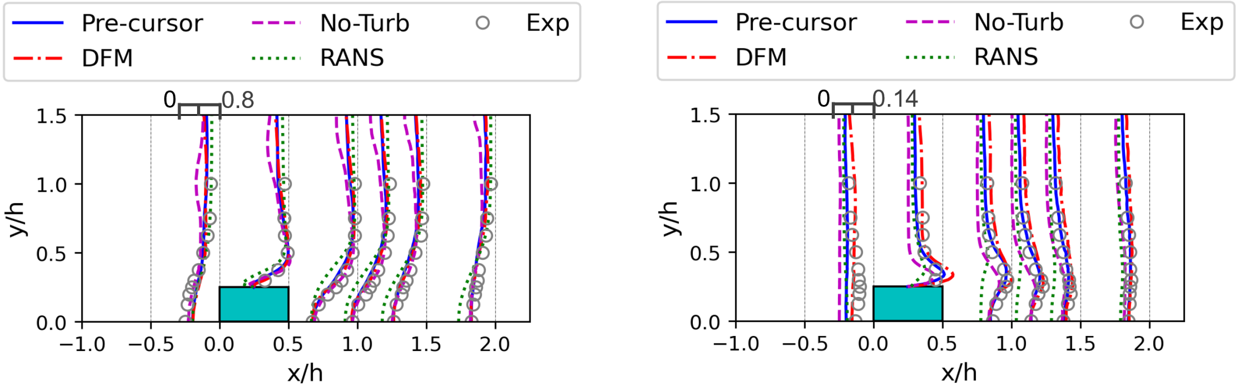
The building block and ground surface are defined as no-slip and modelled with the Spalding wall function. Domain sides are also defined as no-slip to replicate wind tunnel settings, while the top boundary is a free stream. An advective boundary condition is applied at the outlet.

The standard Smagorinsky model with a Smagorinsky constant of 0.12 is applied to solve sub-grid scale kinetic energy. For the convection terms, second-order discretization schemes Filtered Linear 2V are used. This scheme removes high-frequency modes while adding a small amount of upwind scheme for stabilization. Finally, the second-order backward scheme is used for temporal discretizations, with time steps chosen such that the maximum Courant number is less than 0.5 for all the cases.



(a) *Normalized mean velocity, u/u_H, plane $y = 0$.*

(b) *Normalized turbulent kinetic energy, k/u_H^2, plane $y = 0$.*



(c) *Normalized mean velocity, u/u_H, plane $z = H/16$.*

(d) *Normalized turbulent kinetic energy, k/u_H^2, plane $z = H/16$.*

Figure 2. Shows comparison of normalized mean velocity and turbulent kinetic energy profile of LES (precursor simulations (blue line), DFM method (red line), mean profile without turbulence (magenta line) with RANS (green dots) and experiment (grey circles) at plane $y = 0$ and $z = H/16$.

3. RESULTS AND DISCUSSION

Fig. 1 illustrates a comparison of mean velocity and turbulent kinetic energy profile at a boundary inlet for all cases together with the experiment. The mean velocity profile is similar to the experiment for every case, i.e. Fig. 1a. Turbulent kinetic energy is closest to the experiment for AIJ's precursor simulation at the inlet, but coarser grids of precursor simulation lead to under-prediction (Okaze et al., 2021). Contrary, DFM over-predicts turbulent kinetic energy at the inlet due to approximation with experimental data.

Fig. 2 depicts normalized mean streamwise velocity and turbulent kinetic energy profiles for three LES and RANS simulations at different distances from an isolated building at plane $y = 0$ and $z = H/16$ (pedestrian height). The LES results clearly show improved accuracy than RANS. Since AIJ's precursor simulations produced similar atmospheric conditions as the experiment, overall results for both mean flow and turbulent kinetic energy are closest to those found in the experiment compared to other cases. However, as an obstruction to the flow triggers high turbulence, the results turn out to be comparable downwind for all cases, specifically near the building. As shown in Fig. 2c and 2d, results at pedestrian height are not affected significantly downwind since the localized influence of obstruction diminishes the external effect. In addition, it can also be observed that results far from the building have higher errors for inflow without perturbation, similar to the previous study (Tominaga et al., 2008a).

ACKNOWLEDGEMENTS

This research is part of a Future Energy Hub Project funded by The Norwegian Research Council (project no.: 280458), The University of Stavanger, and local industry partners.

REFERENCES

Blocken, B., 2018. LES over RANS in building simulation for outdoor and indoor applications: A foregone conclusion? Building Simulation 11, 821–870.

Franke, J., Hellsten, A., Schlünzen, H., and Carissimo, B., Jan. 2007. Best Practice Guideline for the CFD Simulation of Flows in the Urban Environment. COST action 732, 51.

Gousseau, P., Blocken, B., and Van Heijst, G., 2013. Quality assessment of Large-Eddy Simulation of wind flow around a high-rise building: Validation and solution verification. Computers & Fluids 79, 120–133.

Gousseau, P., Blocken B.and Stathopoulos, T., and Van Heijst, G., 2011. CFD simulation of near-field pollutant dispersion on a high-resolution grid: a case study by LES and RANS for a building group in downtown Montreal. Atmospheric Environment 45, 428–438.

Jia, H. and Kikumoto, H., 2022. Partially averaged Navier-Stokes simulation of flow around an isolated building model with a 1:1:2 shape. Building and Environment 223, 109506.

Klein, M., Sadiki, A., and Janicka, J., 2003. A digital filter based generation of inflow data for spatially developing direct numerical or large eddy simulations. Journal of Computational Physics 186, 652–665.

Liu, J., Niu, J., Du, Y., Mak, C. M., and Zhang, Y., 2019. LES for pedestrian level wind around an idealized building array—Assessment of sensitivity to influencing parameters. Sustainable Cities and Society 44, 406–415.

Okaze, T., Kikumoto, H., Ono, H., Imano, M., Ikegaya, N., Hasama, T., Nakao, K., Kishida, T., Tabata, Y., Nakajima, K., et al., 2021. Large-eddy simulation of flow around an isolated building: A step-by-step analysis of influencing factors on turbulent statistics. Building and Environment 202, 108021.

Tominaga, Y., Mochida, A., Murakami, S., and Sawaki, S., 2008a. Comparison of various revised k–ε models and LES applied to flow around a high-rise building model with 1: 1: 2 shape placed within the surface boundary layer. Journal of Wind Engineering and Industrial Aerodynamics 96, 389–411.

Tominaga, Y., Mochida, A., Yoshie, R., Kataoka, H., Nozu, T., Yoshikawa, M., and Shirasawa, T., Oct. 2008b. AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings. Journal of Wind Engineering and Industrial Aerodynamics 96, 1749–1761.