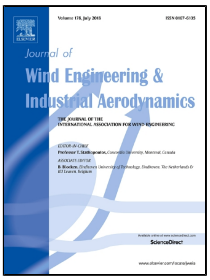
Accepted Manuscript



A review of computational fluid dynamics (CFD) simulations of the wind flow around buildings for urban wind energy exploitation

Francisco Toja-Silva, Takaaki Kono, Carlos Peralta, Oscar Lopez-Garcia, Jia Chen

PII:	S0167-6105(18)30227-7
DOI:	10.1016/j.jweia.2018.07.010
Reference:	INDAER 3675
To appear in:	*Journal of Wind Engineering and Industrial Aerodynamics*
Received Date:	28 March 2018
Accepted Date:	14 July 2018

Please cite this article as: Francisco Toja-Silva, Takaaki Kono, Carlos Peralta, Oscar Lopez-Garcia, Jia Chen, A review of computational fluid dynamics (CFD) simulations of the wind flow around buildings for urban wind energy exploitation, *Journal of Wind Engineering and Industrial Aerodynamics* (2018), doi: 10.1016/j.jweia.2018.07.010

A review of computational fluid dynamics (CFD) simulations of the wind flow around buildings for urban wind energy exploitation

Francisco Toja-Silva[1,2,*], Takaaki Kono[3],Carlos Peralta[4], Oscar Lopez-Garcia[5], Jia Chen[1].

1 Environmental Sensing and Modeling, Technische Universität München (TUM). Theresienstr. 90, Munich 80333, Germany.

2 Barcelona Supercomputing Center (BSC). Nexus II Building - Jordi Girona 29, 08034 Barcelona, Spain.

3 Research Center for Sustainable Energy and Technology, Kanazawa University. Natural Science and Technology Hall 3, 3B416 Kakuma-machi, Kanazawa-shi, Ishikawa 920-1192, Japan.

4 Wobben Research and Development Management Support GmbH. Teerhof 59, Bremen 28199, Germany.

5 Escuela Técnica Superior de Ingenieros Aeronáuticos, Universidad Politécnica de Madrid. Pl. Cardenal Cisneros 3, Madrid 28040, Spain.

* Corresponding author: frantojasilva@yahoo.es

Abstract:

This article presents a review on computational fluid dynamics (CFD) applied to urban wind energy exploitation. The content comprises technical CFD aspects relevant for this application and the current state-of-the-art in building aerodynamics applied to urban wind energy. The majority of studies (more than 50% of the respective criteria) used Reynolds-averaged Navier-Stokes (RANS) turbulence models, the commercial solver ANSYS, simulated a full-scale geometry and studied an isolated building. For RANS, at least second order-accurate discretization schemes must be used, to improve turbulence kinetic energy prediction. In large-eddy simulation (LES) studies, a blending scheme is often needed to avoid numerical instability. Urban wind flow is very complex (i.e. detachment, stagnation), and rigorous validation and verification processes are needed, because only sophisticated turbulence models are able to yield acceptable results. The building-roof shape was optimized for the wind energy exploitation attending to both turbulence intensity and wind velocity. Conventional roof and roof edge shapes were studied, as well as the compatibility with the installation of solar panels. Wind turbines sitting was also discussed. Few simulations of wind turbines installed on building roofs were conducted using wind turbine models, whereas real geometries of vertical axis wind turbines were simulated and optimized.

Keywords:

Building aerodynamics, CFD, computational wind engineering, LES, RANS, urban wind energy.