Evaluation of CFD turbulence models for simulating external airflow around varied building roof with wind tunnel experiment

Georgios K. Ntinas\textsuperscript{2,3}, Xiong Shen\textsuperscript{1} (\textsuperscript{*}), Yu Wang\textsuperscript{4}, Guoqiang Zhang\textsuperscript{2}

\textsuperscript{1}. Tianjin Key Laboratory of Indoor Air Environmental Quality Control, School of Environmental Science and Engineering, Tianjin University, Tianjin 300072, China
\textsuperscript{2}. Department of Engineering, Faculty Sciences and Technology, Aarhus University, Blichers Allé 20, 8830 Tjele, Denmark
\textsuperscript{3}. TUM School of Life Sciences Weihenstephan, Research Department Plant Sciences, Technische Universität München, 85354 Freising, Germany
\textsuperscript{4}. Key Laboratory of Agricultural Engineering in Structure and Environment, Ministry of Agriculture College of Water Resources and Civil Engineering, China Agricultural University, P.O. Box 67, Beijing 100083, China

Abstract
Detailed airflow information around a building can be crucial for the design of naturally ventilated systems and for exhaust air dispersion practices in agricultural buildings like greenhouses and livestock buildings. Full-scale measurements are cost-intensive and difficult to achieve due to varied wind conditions. A common method to gain insight of flow field under different wind conditions is the numerical simulation by using computational fluid dynamics (CFD). Still, evaluation of a CFD models' performance and validation of its predictions with high quality experimental data is necessary before the model is used in practice. In this research three types of common agricultural buildings, arched-type, pitched-type and flat-type roof, were examined by conducting experiments in a wind tunnel with controlled airflow conditions, in order to validate different 3D turbulence models for predicting airflow patterns. The focus of this work was the detailed description of the external airflow field over the varied roof geometries and especially the velocity distribution and turbulent kinetic energy in the wake of each building. Experimental measurements of velocity were performed with a Laser Doppler Anemometer (LDA) and were compared with 3D RANS turbulence models' simulation results. A reasonable agreement was found between experimental and simulation results concerning the velocity and the turbulence kinetic energy with CFD models slightly underestimating these magnitudes. The $k$-$\varepsilon$ series turbulence models and especially the standard $k$-$\varepsilon$, RNG $k$-$\varepsilon$ and Realizable $k$-$\varepsilon$ models presented good agreement concerning velocity contours; however, high prediction error occurred over the roof of the buildings compared to the average values.

Keywords
turbulence, velocity, LDA, CFD, RANS

Article History
Received: 9 October 2016
Revised: 14 March 2017
Accepted: 21 March 2017

© Tsinghua University Press and Springer-Verlag GmbH Germany 2017

1 Introduction
The external airflow field around naturally ventilated buildings has a severe impact on the dispersion of contaminants exhausted from the buildings. Naturally induced flows can be complex even for a simple building geometry requiring thorough examination of the exterior wind environment as well as its interaction with buildings’ openings for ventilation (Karava et al. 2011). The airflow field around the roof outlets is important for the contaminant transport. Once the outlet is located in the flow region with large vortex, the contaminant may concentrate in the region and make it difficult to disperse. Moreover, the contaminant air can be easily removed from the roof when negative pressure prevails at the roof flow region and positive pressure at the inlet. The design of the inlet and outlet positions should
consider the external air field. In some cases, contaminant air may return to the air inlet from the external field, leading to the self-pollution problems (Wu et al. 2012). The velocity and turbulence around the roof outlet are important for the contaminant release. High values of velocity and turbulence kinetic energy are proved to enhance the contaminants dispersion from the outlet.

In practice, in-situ experiments, laboratory experiments and CFD simulation tools can be applied to investigate the external airflow conditions. In-situ experiments can be very costly and time consuming and difficult to repeat because the external wind conditions change dynamically and rapidly all the time. Laboratory experiments can be an alternative way, even though they may suffer from dynamic similarity issues. Instead, computational fluid dynamics (CFD) provides an approach to investigate the external airflow distributions. It calculates the flow field by solving the governing equations such as the Navies–Stokes equations. CFD simulation studies on external flow around ribs and rural buildings have been suggested by many researchers (Yanaoka et al. 2007; Tamura et al. 2008; Tominaga and Statthopoulos 2010; Murakami and Mochida 1988; Shen and Hao 2006; Dados et al. 2011; Ntinas et al. 2014; Vogiatzis et al. 2014).

CFD models and specifically turbulence models have been widely used to predict airflow patterns around obstacles. However, the selection of the proper turbulence model for the simulation of such flow field is a challenge. The use of a laminar model which discards the turbulence items of the Navier–Stokes equation, leads to high-speed computation but poor prediction results (Stamou and Katsiris 2006). The Spalart–Allmaras (S–A) model, in which only one equation is used to describe the turbulent items of the fluid (ANSYS 2009), leads also to high-speed iterations but even though in some applications it has shown good results in airflow patterns prediction (Gilkeson et al. 2009), in most cases its accuracy was not satisfactory (Stamou and Katsiris 2006; Ramponi and Blocken 2012). The most common CFD models applied also in the industry though are the two- equation RANS models (Norton et al. 2007; Bournet and Boulard 2010). The standard $k$–$\varepsilon$ model has been widely used in several applications presenting modest calculation speed but it may fail to predict the airflow in non-isotropic turbulence (Norton et al. 2007). To overcome this weakness, two upgraded models, the RNG $k$–$\varepsilon$ and Realizable $k$–$\varepsilon$ models were developed (Yakhot et al. 1992; ANSYS 2009). Even though the RNG $k$–$\varepsilon$ model is recommended by several scientists; however, it is not certain still, which $k$–$\varepsilon$ model is performing better (Bournet and Boulard 2010). Tominaga et al. (2015) reported that RNG $k$–$\varepsilon$ model exhibited the best overall performance in the prediction of the velocity and turbulent kinetic energy around the pitch roof building, but the prediction accuracy for those behind the building is rather poor. Another category of two-equation RANS models are the $k$–$\omega$ models, with standard $k$–$\omega$ model being able to calculate in a different way the turbulence viscosity comparing to the standard $k$–$\varepsilon$ model. The standard $k$–$\omega$ model is very sensitive to the near-wall grid distribution because of the fully turbulent condition in the region (ANSYS 2009), and therefore it is more suitable for high turbulent airflow conditions, such as the stirred and impinging flow (Hu 2012). The SST $k$–$\omega$ model is combining advantages of the standard $k$–$\varepsilon$ and standard $k$–$\omega$ models by employing a dump function that selects specific CFD turbulence models for specified turbulence conditions (Norton et al. 2007; Hu 2012). Concerning airflow patterns around buildings, the SST $k$–$\omega$ model has shown high accuracy (Ramponi and Blocken 2012). The RSM model, which is a seven-equation model, is characterized by low computational speed compared to the previous turbulence models, but it has the ability to predict the indoor airflow patterns with good accuracy (Norton et al. 2007). However, this is not always the case as in some cases it may lose accuracy (Bartzanas et al. 2007). The direct numerical simulation (DNS) which directly solve the Navier–Stokes equation was applied to simulate the turbulent flow over flat roof building (Fragos et al. 2010). However, the simulation was in two-dimensional and was quite time consuming and not suitable for the industrial application. The Large Eddy Simulation (LES) calculate the airflow by ignoring the smallest length scales via low-pass filtering of the Navier–Stokes equations. It has reported its high performance in simulating the airflow passing the flat-roof building while it was the most computationally expensive to resolve (Rodi 1997).

In the light of the above, it is still not clear which CFD model is more suitable to predict the external airflow patterns around a buildings’ roof. Nevertheless, the examination of the external airflow patterns around buildings with varied roof geometries would help to investigate the odor and particle dispersion as well as wind environment. In this aspect several CFD tools can be utilized, but require to be validated first.

Therefore, the aim of this study was to test the accuracy of varied CFD turbulence models in three types of common agricultural buildings with arched-type, pitched-type and flat-type roof, respectively. These roof geometries are among the most common commercial agricultural structures in practice (livestock buildings and greenhouses) and useful tools exist for their design and construction like the Eurocode “EN 13031-1:2001” (CEN 2001). Hence, we compared the simulation with the wind tunnel experimental results. In our study, the velocity and the turbulence kinetic energy were measured and compared with simulation results by varied turbulence models.