J. Wind Eng. Ind. Aerodyn. 124 (2014) 99–108

Contents lists available at ScienceDirect

Journal of Wind Engineering and Industrial Aerodynamics

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

# Numerical and experimental study of the flow through a geometrically accurate porous wind barrier model

Marijo Telenta*, Jožef Duhovnik, Franc Kosel, Viktor Šajn

*University of Ljubljana, Faculty of Mechanical Engineering; Aškerčeva 6, Ljubljana 1000, Slovenia*

ARTICLE INFO

*Article history:*
Received 20 May 2013
Received in revised form
3 October 2013
Accepted 25 November 2013
Available online 27 December 2013

*Keywords:*
Porous wind barrier
Accurate three-dimensional model
URANS
Bar inclination angle

ABSTRACT

A method in which the complex flow near and through the openings of a porous wind barrier is treated at a detailed level. The flow characteristics of the turbulent wake behind the barrier are experimentally and numerically investigated. The wind barrier is accurately geometrically represented with a three-dimensional model in the numerical simulation. Barrier models consisting of horizontal bars with different inclination angles are considered. The unsteady Reynolds-averaged Navier–Stokes (URANS) computation is applied because the flow is not statistically stationary. The shear stress transport (SST) $k$–$\omega$ turbulence model is used because it shows good behavior in adverse and separated flows. In addition to the three-dimensional URANS numerical study, an experimental study is performed to confirm the numerical data. The aim is to conduct an experimental and numerical study of a fluid flow through the geometrically accurate three-dimensional barrier model and analyze the bar inclination effect on the wake characteristics behind the barrier. As the bar inclination angle decreases, the bleed flow gets stronger, which results in a smaller reduction of the mean streamwise velocity. In addition, the turbulence intensity decreases in the shelter wake with a decreasing bar inclination angle.



## 1. Introduction

Wind barriers are artificial structures designed for wind protection. Their function is to reduce the wind velocity within a certain distance. They are the earliest devices used to provide shelter by decreasing the wind speed in a defined zone behind the barrier. The initial goal of the wind barrier is to reduce hazards caused by the wind, for instance, soil erosion, snow drift, dust emission from open storage, etc.

A number of earlier papers experimentally and numerically investigated the flow behind porous barriers. The experimental studies examined a range of barrier porosities from 0% to 50%. The Bradley and Mulhearn (1983) experiment gave detailed measurements from full-scale field trials for a 50% porous barrier. The barrier in this experiment consisted of vertical wooden bars with dimensions of $1.2\ \text{m} \times 0.08\ \text{m} \times 0.01\ \text{m}$ and 50% porosity. In Lee and Kim (1999), the detailed velocity and turbulent fields behind a porous fence were measured using particle tracking velocimetry (PTV). The Reynolds number in this research was based on the barrier height and was 8360. Dong et al. (2007) used a similar measuring method, particle image velocimetry (PIV), in which the Reynolds number was approximately 10,000. Also, artificial wind barriers made of nets were examined in the previous studies. Pressure drop through the screen was expressed by the wire diameter size in Wakeland and Keolian (2003). Also, in works of Van Renterghem and Botteldooren (2002) woven windscreens with 32% porosity made of polyester were used as a scale model for canopy of trees. The focus of the present research is a barrier with horizontal bars. Thus, the data from the above-mentioned experiments are not applicable.

Previous numerical studies modeled fluid flows through complex geometries but excluded the details of the wind barrier geometry. The two-dimensional simulations were based on the Reynolds-averaged Navier–Stokes (RANS) equations using different turbulence closer models.

In the work of Wilson (1985), the barrier was modeled as a momentum extraction in the momentum equation using the Reynolds-stress closure scheme. The $k$–$\varepsilon$ closure scheme was used, which gave slightly less satisfactory results than the Reynolds-stress scheme. It is reported that satisfactory estimates of the pattern of the turbulent kinetic energy behind the barrier were obtained. However, all simulations failed to predict the sharp speed-up over the barrier.

Santiago et al. (2007) determined the optimum barrier porosity utilizing RANS simulations with three variants of the $k$–$\varepsilon$ turbulence closure models. The turbulence closure models were compared using the same numerical procedure and parameters to evaluate their performance. The permeable barrier was assumed

* Corresponding author. Tel.: +386 147 714 28.
*E-mail address:* telenta@lecad.si (M. Telenta).

0167-6105/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.jweia.2013.11.010

to be a thin porous medium. The physical effect of the porous barrier's presence inside the flow was represented by a pressure drop through the barrier, thus creating a momentum sink. The paper stressed the necessity to link the resistance coefficient of the porous medium with the barrier porosity for modeling purposes.

Packwood (2000) compared wind tunnel data with CFD models of two-dimensional porous barriers. Two turbulence models were used, $k$–$\varepsilon$ and Reynolds stress. A new method was developed for determining the porous barrier resistance, which was modeled as a porous region.

Fang and Wang (1997) numerically investigated the two-dimensional turbulent flow around a porous barrier using a weakly compressible flow method. A large-eddy simulation method with the sub-grid scale turbulence model was applied to take into account the turbulent effects.

Huang et al. (2012) noted that engineers were using the drag law (pressure drop) to represent the porous effects to lower the computational cost. In this approach, only the spatial average flow is represented around the barrier using the drag law. Moreover, the application of the drag law uses drag coefficients that need to be correctly calculated. However, very little drag data of porous barriers are available. This approximation does not include the geometric representation of the barrier. Thus, the interaction between the fluid flow and the openings in the barrier is omitted.

There are considerable challenges to CFD codes when modeling separated flow and the porous barrier itself. Previous studies modeled fluid flow through porous geometries but did not consider the details of the geometry. The main focus was to define a suitable resistance model for a given geometry of a barrier.

A deeper understanding of turbulent structure dynamics is required to evaluate the sheltering effect. Previous studies used the Reynolds averaging method with turbulence closure for a two-dimensional fluid flow simulation in which the porous barrier was represented as a momentum sink. As stated in Bourdin and Wilson (2008), numerical methods utilizing the momentum sink approach for wind barrier modeling treat complex unresolved flow near and through the gaps at a superficial level. The goal of this research is to numerically simulate fluid flow through a geometrically accurate three-dimensional barrier model and resolve the flow near and through the porous barrier. In particular, the objective is to investigate the interaction between bleed flow and reverse flow for different barrier configurations.

## 2. Experimental apparatus and methods

The experimental study was conducted in a scaled wind tunnel simulation to confirm the numerical data. The experiment was carried out in a wind tunnel with test section dimensions of 0.355 m $\times$ 0.407 m $\times$ 1 m. The model was built of horizontal aluminum bars. The height of the bars was 20 mm, and the length was 327 mm. The barrier height was $H=145$ mm and had a flat end at the top. The shape ratio, which is the ratio between the barrier thickness and height, was 0.034; hence, the flow was a thin barrier flow. No roughness elements were placed upstream of the test section to create a boundary layer. The free-stream velocity was 20 m/s, and the corresponding Reynolds number based on the barrier height was $1.62e+05$. The Reynolds number was high enough that the aerodynamic coefficients were the same as a full scale model. The test model has a fixed shape and was rigid in the flow stream. Thus, the time averages of the force and moment coefficients are functions of a single parameter, the Reynolds number. This holds for flows of air at speeds up to a Mach number of 0.3, which was the case for this experiment. In this case, there was laminar flow with a low free-stream turbulence intensity in the wind tunnel, and the model's shapes were such that all the separation locations were geometrically determined. Hence, there was little dependence on the Reynolds number.

The stream-wise velocity profiles obtained were nearly flat. The barrier model was located in a uniform flow whose boundary layer thickness at the barrier location was approximately 0.02 of the barrier height. The barrier model was positioned at $2H$ downstream of the inlet of the test section. Two values were measured, the drag force $F_D$ and the lift force $F_L$ imposed by the fluid on the barrier. The smoke injection method was used to visualize the modification of the flow structure.

## 3. Barrier model

There are two basic types of wind barriers, solid and porous. Porous wind barriers are mainly used as turbulence manipulators and are exploited for many practical applications. There are a variety of porous barrier constructions, such as upright, horizontal, griddled, and screened. Upright and horizontal wind barriers are usually made from bars and are widely used because their construction is simple and they have a low cost. In addition, the bars can be easily replaced. Thus, a porous wind barrier made of a series of horizontal or vertical parallel bars is an economical choice. In this paper, four barrier models with different bar inclination angles are used. The inclination angles are 90°, 60°, 45°, and 30° relative to the horizontal axis for barrier configurations 1, 2, 3, and 4, respectively, as seen in Fig. 1. The geometric porosity of the barrier is defined as the ratio of the total open area between the bars and the total area occupied by the barrier. The barrier porosity is 30% for configuration 1.

### 3.1. Flow pattern characteristics around the wind barrier

The flow field behind the porous barrier is complex where massive separation and vortex shedding are found. As the flow approaches the wind barrier, a high velocity region is formed above the barrier. The flow momentum is transported to higher levels, resulting in a low wind velocity behind the barrier. Flow separation is initiated from the top of the barrier. A separation area is formed behind the barrier and is characterized by reverse flow. The shape of the separation area depends on the porosity, barrier construction, etc. The flow through the porous barrier consists of a bleed flow that passes through the gaps and the displacement flow situated above the barrier. As the porosity changes, the bleed flow and displaced flow vary. There is no bleed flow for a solid barrier.

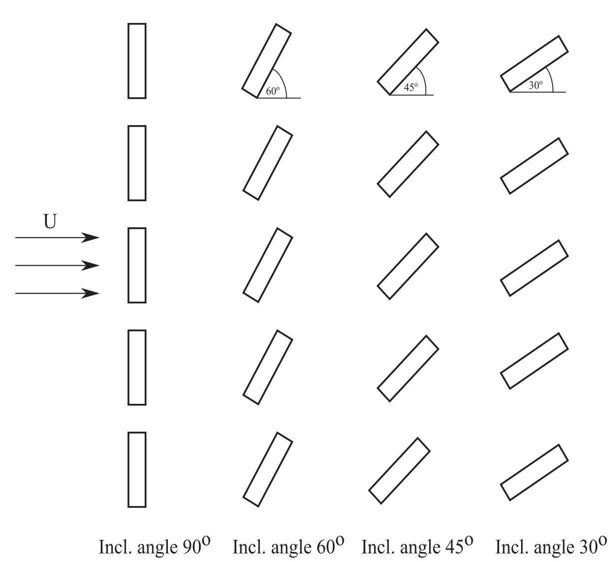


**Fig. 1.** Wind barrier configurations.