The 2015 World Congress on
Advances in Civil, Environmental, and Materials Research (ACEM15)
Incheon, Korea, August 25-29, 2015

Parametric numerical study of wind barrier shelter

Marijo Telenta*[1], Milan Batista[2], M.E. Biancolini[3], Ivan Prebil[1], and Jožef Duhovnik[1]

[1]*Faculty of Mechanical Engineering, University of Ljubljana, Aškerčeva 6, 1000 Ljubljana, Slovenia*
[2]*Faculty of Maritime studies and Transport, University of Ljubljana, Pot pomorščakov, 6320 Portorož, Slovenia*
[3]*Department of Enterprise Engineering, University of Rome "Tor Vergata", Via Politecnico 1, 00133 Roma,Italy*

(Received keep as blank , Revised keep as blank , Accepted keep as blank)

Abstract. This work is focused on a parametric numerical study of the barrier's bar inclination shelter effect in crosswind scenario. The parametric study combines mesh morphing and design of experiments in automated manner. Radial Basis Functions (RBF) method is used for mesh morphing and Ansys Workbench is used as an automation platform. Wind barrier consists of five bars where each bar angle is parameterized. Design points are defined using the design of experiments (DOE) technique to accurately represent the entire design space. Three-dimensional RANS numerical simulation was utilized with commercial software Ansys Fluent 14.5. In addition to the numerical study, experimental measurement of the aerodynamic forces acting on a vehicle is performed in order to define the critical wind disturbance scenario. The wind barrier optimization method combines morphing, an advanced CFD solver, high performance computing, and process automaters. The goal is to present a parametric aerodynamic simulation methodology for the wind barrier shelter that integrates accuracy and an extended design space in an automated manner. In addition, goal driven optimization is conducted for the most influential parameters for the wind barrier shelter.

Keywords: parametric numerical simulation; wind barrier shelter; RBF morph; CFD simulation; mesh morphing

1. Introduction

Wind barriers are the earliest devices used to control wind flow. They are utilized in numerous shelter applications to improve windy conditions that would be appropriate for human needs. For many years, wind barriers have been employed to prevent strong wind effects in secluded areas. In particular, wind barriers can offer optimal protection if used correctly and efficiently. They have been studied since the 1940s in order to find optimum shelter protection. The basic purpose of the wind barrier is to reduce the wind velocity within a certain distance. There are two main types of wind barriers: solid and porous. Primarily porous wind barriers are exploited as turbulence manipulators. It has been found that the most important feature of the wind barrier in wind protection is its porosity (Dong, et al. 2007). Currently, porous wind barriers are being used for wind protection applications of ground vehicles in crosswind conditions.

A number of earlier papers experimentally and numerically investigated the flow behind porous barriers. The experimental studies examined numerous barrier porosities, ranging from 0% to 50% porosity. The Bradley-Mulhearn experiment by Bradley and Mulhearn (1983) gave

∗Corresponding author, Ph.D., E-mail: marijo.telenta@lecad.fs.uni-lj.si
Note: Presented the paper published in "Wind and Structures, An International Journal", Vol. 20, No, 1, 75-93.

detailed measurements from full-scale field trials for a 50% porous barrier. Detailed velocity and turbulent fields behind a porous fence were measured using particle tracking velocimetry (PTV) in Lee and Kim (1999). (Dong, et al., 2007) used a similar measuring method, that is particle image velocimetry (PIV). The Reynolds number was approximately $R_e = 1 \times 10^4$ for both measurements. Also, in previous work artificial wind barriers made of nets were analyzed. In particular, pressure drop through the screens was expressed by the wire diameter size in Wakeland and Keolian (2003). Woven windscreens with 32% porosity made of polyester were used as a scale model for canopy of trees in work of Van Renterghem and Botteldooren (2002).

Computational Fluid Dynamic (CFD) simulation software is used to obtain insight into the flow mechanisms that contribute to the aerodynamic forces and moments acting on the vehicle. Current state-of-the-art of the aerodynamic CFD simulation process consists of CAD model creation, mesh generation, solver set-up, and solver calculation of the flow structures. Drag and lift forces, flow pathlines, and pressure distribution are extracted from the numerical solution. One of the objectives of the CFD analysis is to find the shape parameter combination to achieve optimum goal. Consequently, a large design space with numerous shape arrangements needs to be evaluated. The aerodynamics optimization simulation process, if not automated, would require significant time and effort. The paper Khondge and Sovani (2012) reports a process in which a large set of design alternatives are considered. The process combines morphing, advanced CFD solver, high performance computing, and process automaters. This methodology is used for parametric numerical simulation of barrier shelter in the current work.

Previous studies modeled fluid flow through porous geometries while not considering the details of the barrier's geometry. The main focus was to define a suitable resistance model for a given geometry of a barrier. Previous work, Packwood (2000), Fang and Wang (1997), and (Huang *et al*. 2012), used 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. A deeper understanding of the turbulent structure dynamics is required to evaluate the barrier sheltering effect. Author's previous work (Telenta *et al.* 2014) addressed this issue. URANS numerical simulations, verified with experimental data, were done in which fluid flow was simulated through geometrically accurate three-dimensional barrier model in order to resolve the flow near and through the porous barrier. The objective was to investigate the interaction between the bleed flow and the reverse flow for different barrier configurations. Present paper extends the research scope with parametric study of the bar angle influence on the barrier shelter.

Parametric numerical study in this work combines the mesh morphing and design of the experiments in automated method without re-meshing. Mesh morphing is emerging as a meaningful approach for the definition of a shape parametric CFD model. New shapes are generated by deforming the mesh of the baseline CFD model, i.e. just updating nodal positions, which requires a negligible computational time compared to any re-meshing procedure. Importantly, preserving the same mesh structure eliminates the re-meshing noise that can be confused with the effect of the design parameters.

Several algorithms have been explored for this task. A common and well-established technique, the Free Form Deformation (FFD) by Sederberg and Parry (1986) method, deforms volumes and controls their shape using a trivariate Bernstein polynomial. The method is mesh-less, so it can be easily implemented in parallel partitioned meshes with hybrid elements. It allows the definition of new interesting shapes but it lacks accurate local surface control. Such accurate control can be achieved using mesh-based methods, for example in the pseudo-solid method (Masud et al., 2007), where an elastic FEM solution is used to propagate the