ETMM

ETMM13 Rhodes, Greece

15-17 September, 2021.

CONFERENCE PROCEEDINGS

CFD EVALUATION OF AIRFLOW PATTERNS AROUND BEACH HOUSES WITH DIFFERENT WIND FACING SIDES

P. Pourteimouri [1], G.H.P. Campmans [1], K.M. Wijnberg [1], S.J.M.H. Hulscher [1]
[1] University of Twente, Water Engineering and Management

p.pourteimouri@utwente.nl

Abstract

The attractiveness of beaches to people has led, in many places, to the construction of buildings at the beach-dune interface. Buildings change the local airflow patterns which, in turn, alter the sediment transport pathways and magnitudes. This induces erosion and deposition patterns around the structures. In this study, a numerical model is developed using the open-source computational fluid dynamics solver OpenFOAM. First, the impact of wind facing surface on the near-surface airflow patterns is investigated. Second, the near-surface horizontal divergence of the velocity field is calculated to interpret the impact of changes in airflow patterns on potential erosion and deposition patterns around the buildings.

1 Introduction

Coastal zones worldwide offer a wide variety of valuable resources and recreational activities, and they have always been attractive to human populations. Rapid population growth in coastal areas leads to an increased demand for construction of restaurants, sailing clubs, beach houses and pavilions at the beach-dune interface. These structures act as obstacles for the wind flow and change the local airflow patterns close to the building which, in turn, alter the sediment transport pathways. They might influence the aeolian sand dunes and their functioning as a natural buffer zone over a longer time scale. Therefore, investigating the impact of buildings' positioning at the beach is of high importance to safeguard their functioning as flood defense. The present study aims to quantitatively describe the impact of different building's positioning on airflow patterns, focusing on near-bed flows that will impact wind-driven sand transport.

2 Methodology

Computational Fluid Dynamics (CFD) uses a set of numerical algorithms that enables the computer to predict the flow motion by solving the Navier-Stokes equations over the computational domain. In this study, an open-source CFD solver, OpenFOAM, was used to solve the Reynolds-averaged Navier-Stokes equations (RANS) for airflow around buildings with different wind facing sides. The so-called simpleFoam solver was used which is used for steady-state and turbulent flow simulations. The commonly used $k-\varepsilon$ turbulence closure model which has a relatively low computational cost was implemented to solve the turbulent flow structures that are formed around the structure. The computational domain is $150\ m$ in length, $150\ m$ in width and $50\ m$ in height. The buildings have a length of $6\ m$, width of $2.5\ m$ and height of $2.5\ m$. The row of ten buildings is located at the center of the domain. The buildings' dimensions and their number in a row are selected based on the beach houses at Kijkduin beach, the Netherlands shown in figure 1. The logarithmic velocity profile, u, turbulent kinetic energy, k, and turbulence dissipation rate, ε, proposed by Richards and Hoxey (1993) were implemented as the inlet boundary conditions. The aerodynamic roughness length of $y_0 = 0.00001$ m is applied for the bed surface. The reference velocity, $U_{ref} = 17\ m/s$, at a reference height of $y_{ref} = 1.8\ m$ is used at an angle $\theta_W = 45°$. The no-slip boundary condition was used for the bottom of the domain and buildings' faces. Figure 2 shows the schematic representation of the computational domain. The values of the geometric parameters shown in figure 2, are given in table 1.

Figure 1. A row of beach houses at Kijkduin beach, the Netherlands (source: google earth).

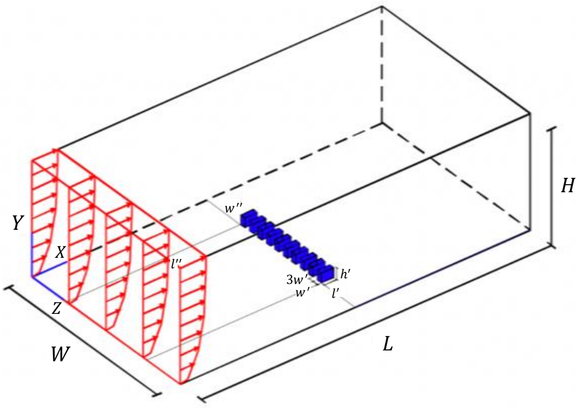

Figure 2. Numerical computational domain including a row of ten buildings.

Table 1
Values of the geometric parameters of the computational domain and row of buildings.

Parameter	Value $[m]$
$\boldsymbol{L}$	150.00
$\boldsymbol{W}$	150.00
$\boldsymbol{H}$	50.00
$\boldsymbol{l'}$	6.00
$\boldsymbol{w'}$	2.50
$\boldsymbol{h'}$	2.50
$\boldsymbol{l''}$	72.00
$\boldsymbol{w''}$	28.75

3 Results and Discussion

In this study, the horizontal flow divergence is used to estimate erosion and deposition patterns around buildings. The velocity field and the horizontal flow divergence patterns at a near-surface plane, $y = 0.25\,m$, for three different buildings configurations are shown in figure 3. The buildings rotate around their center from $\theta_B = 0°$ to $\theta_B = -45°$ and $\theta_B = 45°$, providing the largest to lowest wind facing sides. The inter-distance between two neighbor buildings is three times one building's width.

The results of $\theta_B = 0°$ show that the flow is divided into two branches at the lower left corners of the buildings. A large recirculation region forms just behind the buildings. The wind speed results show the highest flow blockage for $\theta_B = -45°$ and $\theta_B = 0°$. This results in additional depositions just in front of the windward faces of the buildings (Figure 3B, red shaded colors). For $\theta_B = -45°$, the most intensive erosion is expected to happen in front of the windward faces of the buildings (Figure 3B, green shaded colors). Furthermore, the horizontal flow divergence patterns show that the deposition tails behind the buildings form parallel to the wind direction and the highest amount of sedimentation is expected to occur behind the buildings with $\theta_B = -45°$. Results of the near-surface velocity field and the horizontal flow divergence patterns show that for buildings with $\theta_B = 45°$, the minimum flow blockage happens at the inter-distance between buildings. In addition, two counter-rotating vortices form just behind the leeward face of the buildings that are equal in size.

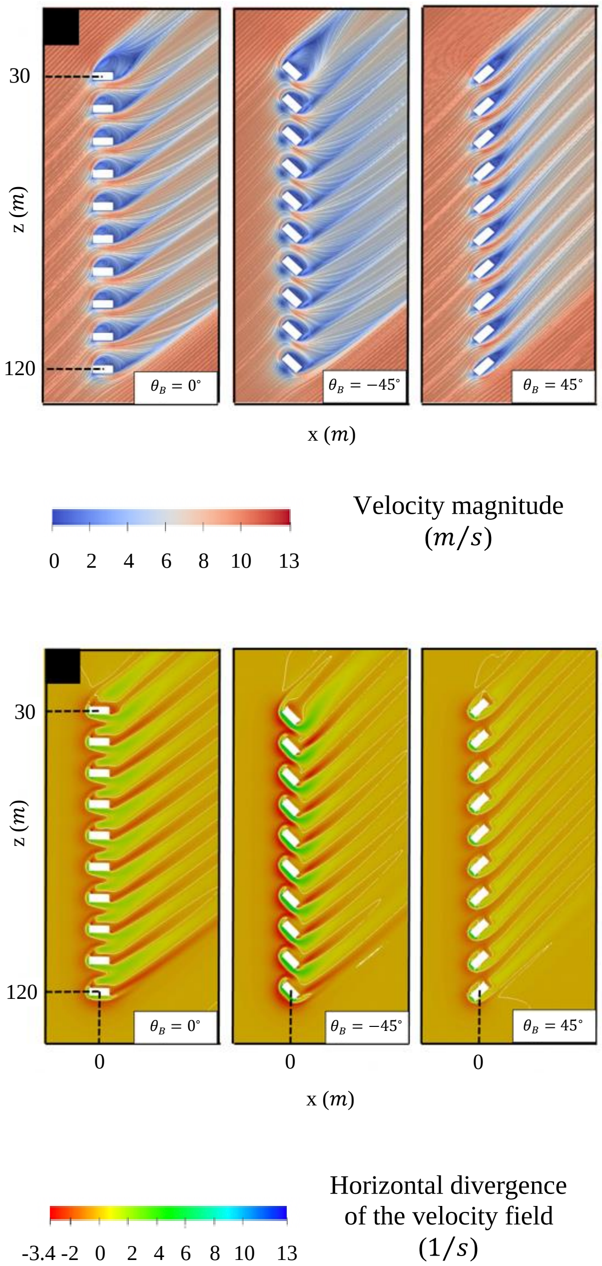

Figure 3. A) Horizontal velocity field around the buildings at $y = 0.25\,m$ (the white lines are streamlines), B) Erosion and deposition patterns around the buildings at $y = 0.25\,m$ (the while lines are zero contours).

Acknowledgements

This PhD project is part of a ShoreScape project that is funded by Netherlands Organisation for Scientific Research (NWO) and co-funded by Rijkswaterstaat (RWS) and Hoogheemraadscap Hollands Noorderkwartier (HHNK).

References

Richards, P. J., & Hoxey, R. P. (1993). Appropriate boundary conditions for computational wind engineering models using the k-ε turbulence model. In Computational Wind Engineering 1 (pp. 145-153). Elsevier.