Journal of Construction and Engineering Technology
Journal homepage: www.journal.ferpi.uz

AERODYNAMIC STUDY OF THE CHARACTERISTICS OF PEDESTRIAN OVERSTANDS UNDER WIND LOAD

Kurbonova Nilufar Ulugbek kizi

Fergana polytechnic institute
Madaliev.me2019@mail.ru,

Abstract

To improve the safety and ease of use of pedestrian overpasses in strong winds, aerodynamic optimization of building shapes is considered the most effective approach. Aerodynamic optimization aims to solve a problem at its source, as opposed to structural optimization, which aims to improve the wind resistance of a structure. With the advancement of technology, buildings with complex plan shapes such as square and round, round and hexagonal, square and octagonal, etc. have been considered by many architects keeping in mind the aesthetics of the building. The wind flow around a high-rise building with complex plan shapes having different height ratios is different from what we get when analyzing a conventional shape. Since data on such buildings is not available, the need for wind tunnel or CFD testing becomes important to analyze the effects of wind on such buildings. CFD (computational fluid dynamics) for determining wind response is becoming extremely popular.

Keywords: Navier–Stokes equations, separated flow, SST model, Comsol Multiphysics

Introduction

In the design of high-rise structures, wind is considered as one of the important horizontal forces that has a significant influence on the response of the building. Wind puts pressure on the surface of a structure, creating forces that can cause it to sway or vibrate. These horizontal forces must be resisted by the structural system to ensure the stability and safety of the building. There are several factors that influence the force of wind on high-rise structures. These include the height and shape of the building, the terrain and surrounding buildings, and wind speed and direction. When designing a building, it is also necessary to take into account the local wind climate and the possibility of wind gusts or extreme wind events. Engineers use various methods to account for wind force. Wind tunnel tests are usually carried out to study the wind flow around a building and accurately estimate the pressure distribution. Computer modeling and computational fluid dynamics (CFD) are also used to model and analyze the effects of wind on a structure. The structural system of a high-rise building is designed to resist wind forces in various ways. These may include the use of bracing systems, shear walls or outriggers to provide rigidity and reduce lateral deflection. Damping devices such as tuned mass dampers or dampers using fluid or viscoelastic materials can also be used to reduce a building's response to wind-induced vibrations. In general, wind is a critical factor in the design of high-rise structures, and engineers must carefully analyze and design a building to ensure its stability and safety under varying wind conditions.

Computational Fluid Dynamics, or CFD (Computational Fluid Dynamics), is a method and toolkit for solving fluid dynamics equations on a computer. CFD is used to model and analyze the movement of liquids and gases, and to study and optimize various engineering problems related to fluid dynamics. CFD is based on physical and mathematical equations that describe the properties of moving liquids or gases. These equations are solved numerically using approximations and finite element, finite volume, or finite difference methods. CFD can be used to simulate various hydrodynamic phenomena, such as the flow of air or water around a body, the

Published online: 20 February 2024
https://doi.org/10.5281/zenodo.10947077
©Fergana Polytechnic Institute
ISSN **2181-4473**

distribution of flow structure, heat exchange between liquids and solids, mixing of substances in liquids, and many others. The benefits of CFD include the ability to quickly and cheaply study various designs or optimize system parameters without the need for physical simulation in a laboratory or test bench. CFD also makes it possible to study phenomena that cannot be reproduced in the laboratory, such as high-speed or complex three-dimensional flows [1–2].

One of the big challenges encountered when using CFD is turbulence modeling. Turbulence modeling involves describing and predicting the motion of turbulent flow. Turbulence is the stochastic, chaotic and complex movement of a liquid or gas, which is characterized by unpredictable and random fluctuations in speed and pressure.

There are two main approaches to modeling turbulence:

Physical models: These models are based on the Navier-Stokes equations, which describe the movement of a liquid or gas. Such models can be very accurate, but require large computational resources and time to solve the equations. Examples of physical models include direct numerical simulation (DNS) and Reynolds Averaged Equations (RANS).

Statistical Models: These models are based on statistical patterns observed in turbulent flows. They allow the prediction of average velocity and other flow characteristics without the need to solve the detailed Navier-Stokes equations. Examples of statistical models include k-ε models and k-ω models.

Turbulence modeling capabilities are widely used in various fields such as aerodynamics, fluid dynamics, meteorology and engineering applications. They help in predicting the behavior of turbulent flows, which can be important for design optimization or weather and climate forecasting [3–4].

The most common is the Reynolds approach. Based on this approach, a system of Reynolds-averaged Navier-Stokes (RANS) equations is obtained. To solve the RANS equation, the Menter k-ω SST (Shear Stress Transport) turbulence model was used [5–6]

The Menter k-ω SST (Shear Stress Transport) model is a turbulence model used in computational fluid dynamics (CFD) simulations. It is an extension of the k-ε and k-ω models, and is generally considered more accurate and realistic for a variety of problems. Today, with the help of these m

2. SST turbulence model.

Menter's shear stress transfer (SST) model [5-6] is a combination of the k-ε and k-ω models. For the wall layer, k-ω is used, for the outer region - k-ε. This model is currently very popular and is included in many CFD packages.

$$\begin{cases} (\mathbf{U}\cdot\nabla)k = \nabla[(\nu+\sigma_k\nu_t)\nabla k] + P - \beta^*\omega k, \\ (\mathbf{U}\cdot\nabla)\omega = \nabla[(\nu+\sigma_\omega\nu_t)\nabla\omega] + \frac{\gamma}{\nu_t}P - \beta\omega^2 + 2(1-F_1)\frac{\sigma_{\omega 2}}{\omega}\nabla\omega\nabla k. \end{cases} \quad (1)$$

Here k is the specific turbulent kinetic energy ($m^2\ s^{-2}$), ω is the specific rate of turbulent dissipation (s^{-1}). Other values are presented in works [5-6].

3. Mathematical statement of the problem

When studying the aerodynamics of buildings, a high-rise building is defined as a building whose height exceeds the width of the leeward façade by three or more times. In Fig. 1 shows a view of the simulator of pedestrian overpasses.

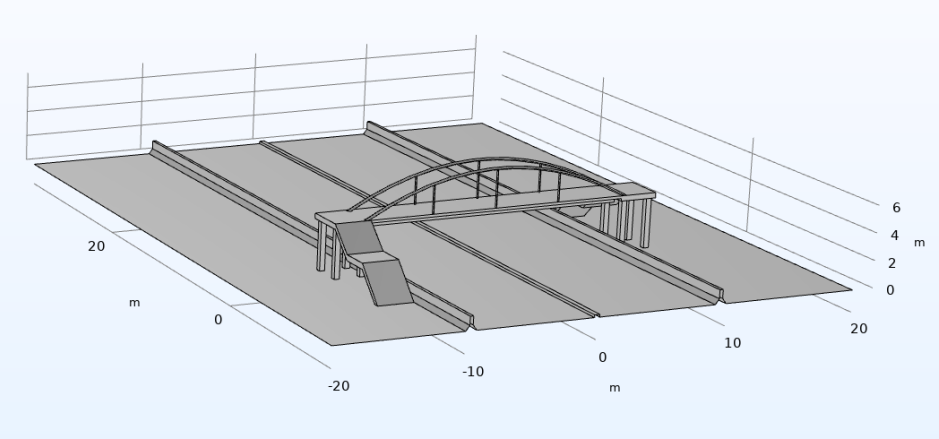


Fig. 1. Pedestrian overpass view in the simulator.

To study the process, the geometry was divided into 428,188 non-structural computational meshes, which were presented in Fig. 2.

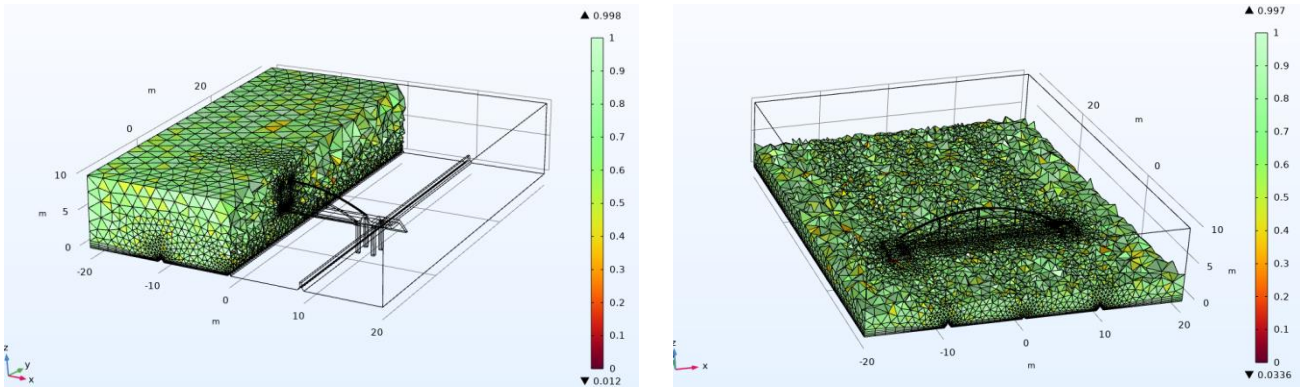


Fig. 2. Non-structural computational mesh.

4. Solution method and boundary conditions

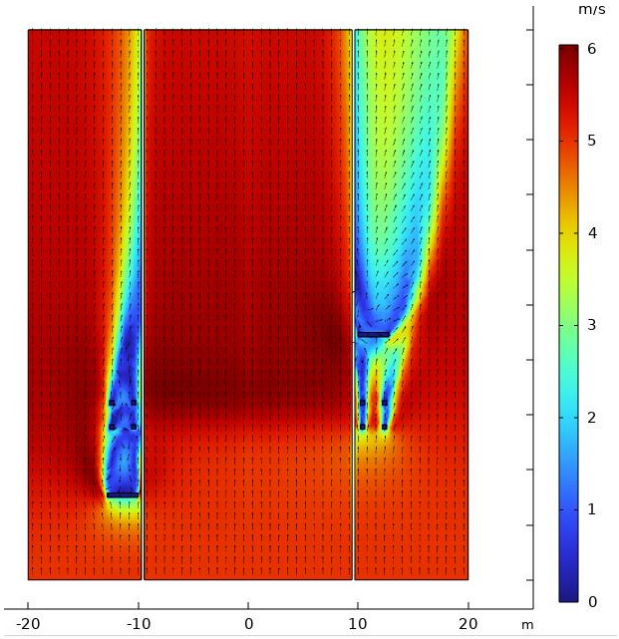
COMSOL Multiphysics offers a range of solvers to solve different types of physics problems. The choice of solver depends on the type of physics being modeled, the complexity of the problem, the desired accuracy, and the available computing resources. Standard COMSOL Multiphysics solvers were used for the standard SST turbulence model. The average wind speed in Tashkent is 15 m/s. The following boundary conditions were set for the SST model

$$\omega_{\infty} = 10\frac{U_{\infty}}{L}, \quad k_{\infty} = 0.1\frac{\nu_{\infty}U_{\infty}}{L}. \tag{2}$$

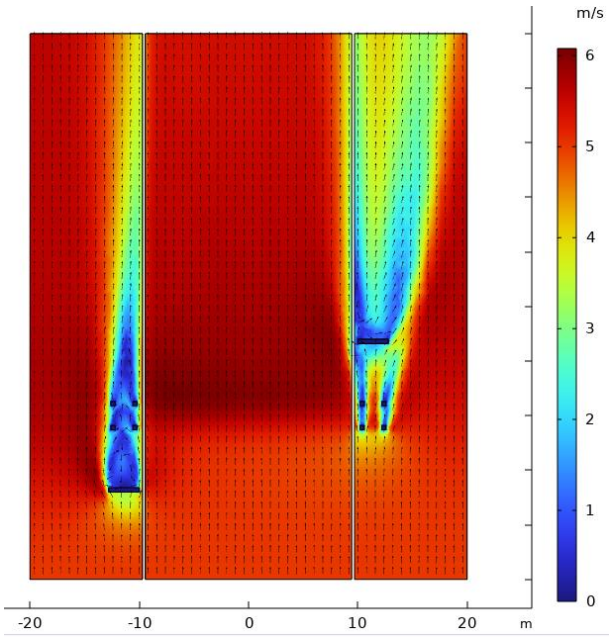
L — characteristic size of a streamlined body. The remaining boundary conditions were specified in the standard way.

5. Results and its discussion

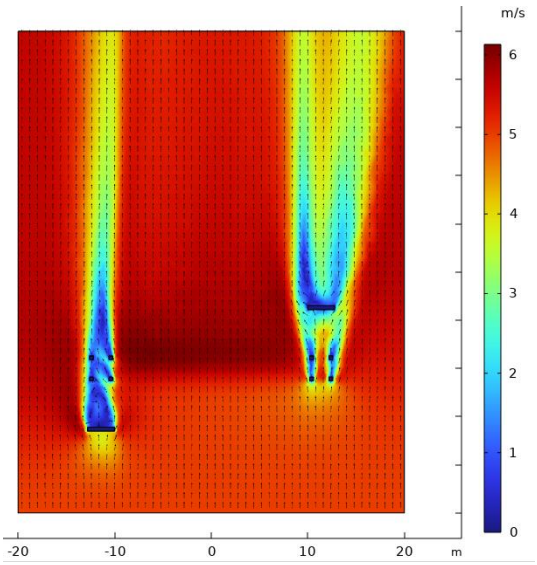
In Fig. Figure 3 shows speed contours and vector direction at different heights from the earth's surface.



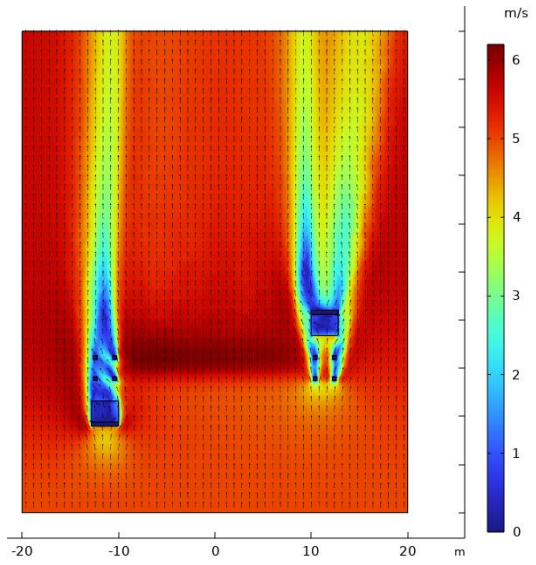
H=0.5 m



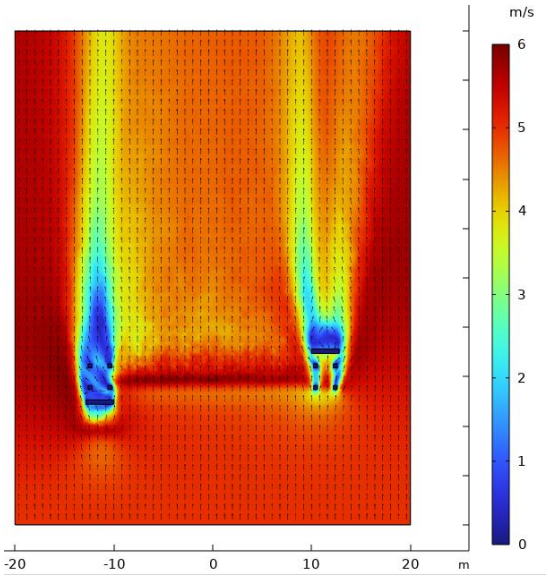
H=1 m



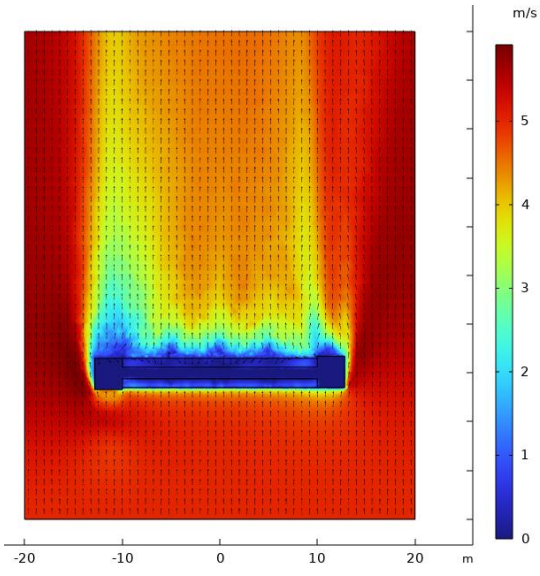
H=1.5 m



H=2 m



H=3 m



H=4 m

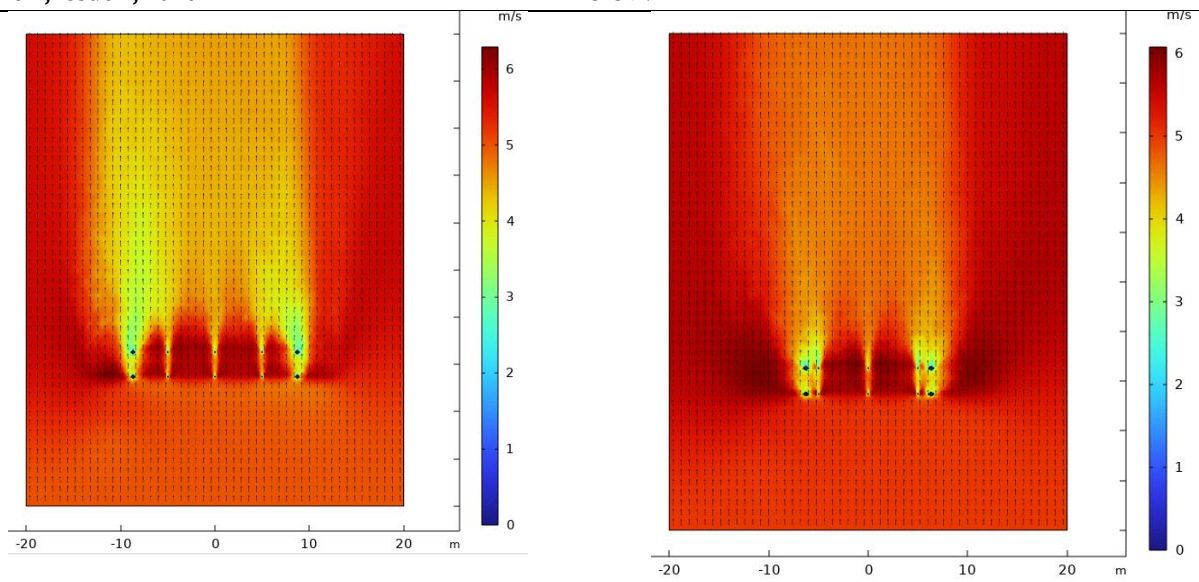


Fig. 3 Isolines of the velocity vector at different heights from the earth's surface.

According to the data presented in Figure 3, it can be seen that the nature of the movement of air masses changes depending on the height. This indicates the presence of a vertical wind flow structure. At lower altitudes there is likely to be more turbulent or rather irregular air movement, while at higher altitudes a more steady and uniform flow may predominate. Such differences in air flow patterns at different heights can be significant when designing structures exposed to wind, such as pedestrian overpasses. Given these data, it is necessary to adapt the design taking into account the vertical structure of the wind flow to ensure the safety and comfort of users.

In Fig. Figure 4 shows the speed and pressure isolines of the central part of the building.

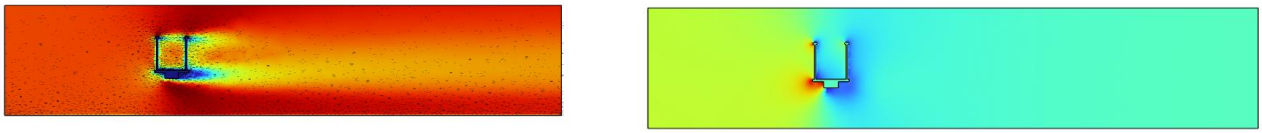
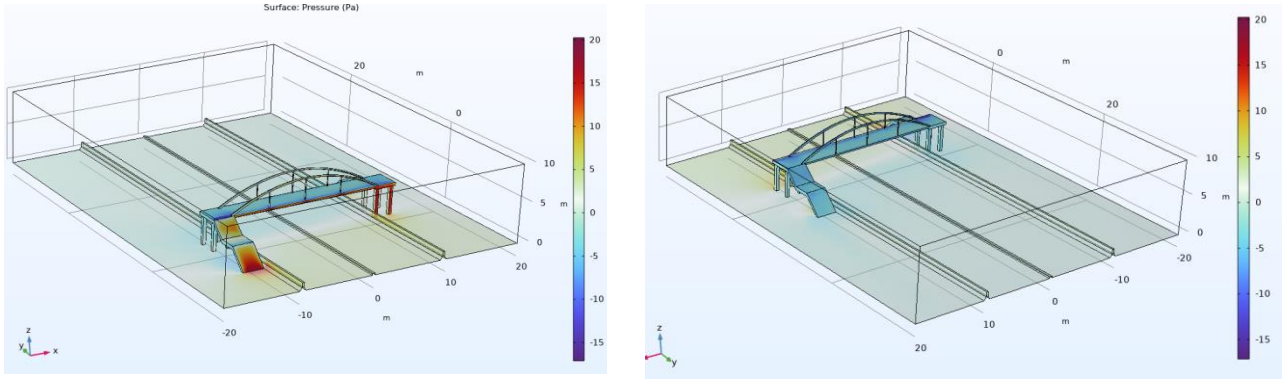


Fig. 4 Velocity and pressure contours of the central part of the building.

Figure 4 shows the vortices. Figure 4 also shows a sharp drop in pressure after the building.

In Fig. Figure 5 shows pressure isolines in the surface of the building.

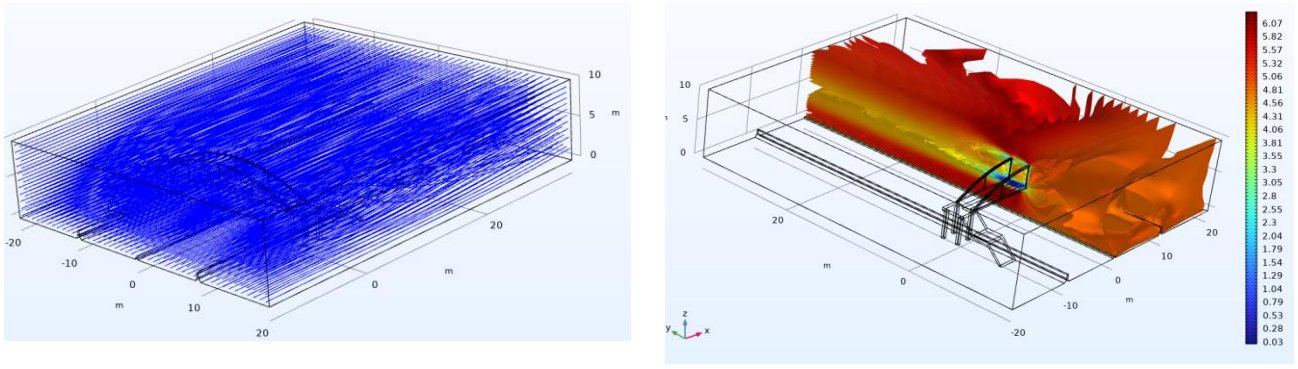


View from the front Back view

Fig. 5 Pressure contours in the building surface.

From the graph presented in Figure 5, it is obvious that with an increase in the speed of movement of air masses, a decrease in air pressure is observed. This corresponds to the physical laws that describe the relationship between speed and pressure in the atmosphere. As air flow speed increases, the kinetic energy of air molecules increases, causing them to collide more intensely with the surfaces on which they fall. This in turn creates lower pressure on these surfaces. Understanding this relationship is important when designing various structures and engineering systems exposed to wind. For example, when designing buildings or structures, it is necessary to take into account changes in air pressure depending on wind speed to ensure their stability and safety. It can also be important when designing aerodynamic systems such as airplane or car wings to optimize their efficiency and handling.

In Fig. 6 shows the streamline and velocity isosurfaces.



streamline isosurface

Fig. 6 shows the streamline and velocity isosurfaces.

Figure 6 shows the change in the shape of air flow lines depending on factors such as wind speed and direction, as well as the geometry of surrounding objects. Changing the shape of flow lines allows us to visualize how air masses move around objects such as buildings, mountains or other obstacles. This is important for understanding how wind affects the environment and what its effects may be on various structures and objects. The shape of airflow lines may indicate the presence of turbulence, eddies, or areas of high pressure. It can also be useful in analyzing the aerodynamic characteristics of various objects and in designing to minimize the impact of wind on them. Studying the changes in airflow lines in Figure 6 allows engineers and designers to make more informed decisions in design and construction, such as when choosing the location of objects, optimizing shape and configuration, or taking measures to reduce the impact of wind on the environment.

6. Conclusion

When carrying out detailed design of pedestrian overpasses that are surrounded by wind flow, it becomes important to take into account and analyze the impact of this factor. The presence of such interaction becomes a key factor determining both the safety and comfort of using the overpass for pedestrians, as well as for those who are in its vicinity. It is important to emphasize that the effective inclusion of this information in the design process of pedestrian overpasses will provide a higher level of safety and comfort for users. Thus, by taking into account the above-mentioned factors and information, a more optimal solution can be achieved in the design of pedestrian overpasses, which will improve their functionality and ensure that the needs of users are met.

References

[1]. Orozco Murillo, W., Palacio-Fernande, J. A., Patiño Arcila, I. D., Zapata Monsalve, J. S., & Hincapié Isaza, J. A. (2020). Analysis of a Jet Pump Performance under Different Primary Nozzle Positions and Inlet Pressures using two Approaches: One Dimensional Analytical Model and Three Dimensional CFD Simulations. Journal of Applied and Computational Mechanics, 6(Special Issue), 1228-1244.

[2]. Hadad, K., Eidi, H. R., & Mokhtari, J. (2017). VOC level control by ventilation improvement of Flexography printing room using CFD modeling. Journal of Applied and Computational Mechanics, 3(3), 171-177.

[3]. Tsega, E. G., & Katiyar, V. K. (2019). A Numerical Simulation of Inspiratory Airflow in Human Airways during Exercise at Sea Level and at High Altitude. Journal of Applied and Computational Mechanics, 5(1), 70-76.

[4]. Sentyabov A.V., Gavrilov A.A., Dekterev A.A. Investigation of turbulence models for computation of swirling flows. Thermophysics and aeromechanics. 18:1, 73-85, 2011.

[5]. Menter F. R."Zonal two-equation k-ω turbulence models for aerodynamic flows". AIAAPaper 1993-2906.

[6]. Menter F. R., Kuntz M., and Langtry R. "TenYears of Industrial Experience with the SST Turbulence Model". Turbulence, Heat and Mass Transfer 4, ed: K. Hanjalic, Y. Nagano, and M. Tummers, Begell House, Inc., 2003, pp. 625 - 632.

[7]. Pasha, A. A. (2018). Study of parameters affecting separation bubble size in high speed flows using k-ω turbulence model. Journal of Applied and Computational Mechanics, 4(2), 95-104.

[8]. Malikov, Z. M., & Madaliev, M. E. (2021). Numerical study of a swirling turbulent flow through a channel with an abrubt expansion. Vestnik Tomskogo Gosudarstvennogo Universiteta. Matematika i Mekhanika, (72), 93-101.

[9]. Malikov, Z. M., & Madaliev, M. E. (2021). Mathematical modeling of a turbulent flow in a centrifugal separator. Vestnik Tomskogo Gosudarstvennogo Universiteta. Matematika i Mekhanika, (71), 121-138.

[10]. Erkinjon son M.M. Numerical Calculation of an Air Centrifugal Separator Based on the SARC Turbulence Model, J. Appl. Comput. Mech., 7(2), 2021, x–xx. https://doi.org/10.22055/JACM.2020.31423.1871

[11]. Spalart, P. R., Jou, W. H., Strelets, M., and Allmaras, S. R., "Comments on the Feasibility of LES for Wings and on a Hybrid, RANS/LES Approach," Advances in DNS/LES, Proceedings of 1st AFOSR International Conference on DNS/LES, Vol. 1, Greyden Press, Columbus, 1997, pp. 137–147