Abstract

The flow around bluff bodies is a complex phenomenon that has been the subject of intensive research for some time, particularly in the context of wind turbine aerodynamics. Boundary layer separation, an undesirable effect in flow, leads to changes in velocity distribution and the formation of vortices. Despite extensive research on flow control, there remains a need to develop new solutions to optimize wind turbine profiles.

This study reviews the literature on flow around airfoil profiles and methods for controlling it, including the use of vortex generators and microcylinders. The development of wind energy as a renewable, low-emission CO_2 energy source focuses on optimizing wind turbines for power output and aerodynamic efficiency. Key challenges include managing power under varying wind conditions and minimizing dynamic loads that can lead to premature wear of turbines. To address these challenges, various flow control methods are employed, including both active and passive flow control. Passive flow control methods, such as research on micro-cylinders and bluff bodies, involve generating characteristic vortex structures that improve the aerodynamic properties of wind turbines.

This work presents experimental results concerning the control of boundary layer separation using bluff bodies (microcylinders). The studies were conducted on the NACA 0012 aerodynamic profile, a classic symmetric model. Advanced Particle Image Velocimetry (PIV) was used for precise measurement of velocity fields in the flow and analysis of boundary layer separation phenomena. Additionally, an aerodynamic balance was employed to calculate key aerodynamic coefficients, such as the lift coefficient C_L [-], the drag coefficient C_D [-], and aerodynamic efficiency L/D [-].

In the initial phase of the research, the focus was on analyzing the impact of the angle of attack on boundary layer separation. Angles of attack were examined in the range from 13° to 19° to identify the angle at which separation was most prominent and intense. After thorough investigation, an angle of attack of 17° was selected for further analysis. The next stage involved introducing microcylinders into the flow. Initially, circular cross-section microcylinders of various sizes d/c [-] were studied. Five sizes were analyzed: $d/c = 0.005, 0.010, 0.015, 0.020, 0.025$ [-] at different positions relative to the profile. This part of

the research aimed to examine how variations in the position and size of the micro-cylinder affected boundary layer separation and aerodynamic coefficients. Following the analysis of these results, further studies were conducted on microcylinders of size $d/c = 0.015$ [-] but with different cross-sectional shapes (e.g., square and triangular). The purpose of changing the shape of the microcylinder was to investigate how the geometry of the microcylinder affected boundary layer separation. The results are presented as contour maps of normalized longitudinal velocity U_x/U_∞ [-], transverse velocity U_y/U_∞ [-], turbulent kinetic energy $TKE/(U_\infty)^2$ [-], and vortex evolution over time. Based on measurements with the aerodynamic balance, the lift coefficient C_l [-] and the aerodynamic drag coefficient C_d [-] which are crucial parameters describing the aerodynamic efficiency of the investigated configurations, were calculated. These results were further enriched by analyzing the surface area of the boundary layer separation bubble in cases where the addition of the microcylinder did not brought beneficial effects. This allowed for a precise determination of situations where the presence of the microcylinder did not improve aerodynamics, providing additional insights into optimal configurations.

The analysis revealed that microcylinders, despite their small size, played a significant role in controlling the flow around the NACA 0012 profile. The generation of local disturbances around the microcylinders prevented boundary layer separation on the profile surface, leading to flow stabilization and reduced aerodynamic drag. This phenomenon is particularly evident as reduced turbulent kinetic energy $TKE/(U_\infty)^2$ [-].

In practice, the use of microcylinders in aerodynamic designs, such as those found in aviation or automotive engineering, can lead to significant improvements in energy efficiency by reducing aerodynamic drag and enhancing flow stability around objects. Furthermore, understanding the impact of different microcylinder shapes on flow can provide valuable insights for further research and optimization in the design of aerodynamic components based on specific engineering requirements.