ABSTRACT

Wind turbines have become a prominent feature of the global energy landscape, symbolizing the transition towards sustainable and renewable energy sources. The design and aerodynamic performance of the wind turbine blades, in turn, heavily rely on the characteristics of the airfoils they employ. The interaction between airfoils and the wind is complex, involving phenomena such as lift, drag, stall, and flow separation. Understanding these aerodynamic characteristics is fundamental to the design, operation, and performance assessment of wind turbines. In this project work, the aerodynamics characteristics of NREL phase VI wind turbine blade is analysed. ANSYS FLUENT software is used for computational fluid dynamic analyis to extract the pressure distribution to compare the aerodynamic factors, such as lift, drag, moment and torque. The flow behavior is analysed at the different velocity ,i.e, 7m/s, 10m/s and 15m/s. The flow around the blade and the separation point also discussed. The pressure co efficient values are compared at different radial locations.