NUMERICAL STUDY ON THE EFFECT OF DIFFUSER GEOMETRY AND DESIGN FOR IMPROVING DAWT (DIFFUSER AUGMENTED WIND TURBINE) PERFORMANCE

Name : Adhana Tito Hary Wibowo
NRP : 2411 106 007
Program Study : S1- Engineering Physics
Department : Engineering Physics FTI-ITS
Supervisor : 1. Dr. Gunawan Nugroho, S.T., M.T.
             2. Ruri Agung Wahyuono, S.T., M.T.

Abstract

Numerical study was conducted on the effect of a diffuser geometry variation of the angle flange on diffuser DAWT performances using Computational Fluid Dynamics (CFD) Simulation. The focus objective of research is to analyze the flow simulation on the effect of the angle tilt geometry of a flat flange to the increased velocity profiles and static pressure in the diffuser that affected on diffuser’s performance. The verification results of simulation and computational conditions by comparing the quantitative data from Ohya’s experiment. Qualitative analysis of velocity contour, streamline contour and direction of the velocity vector contour. Quantitative analysis by determined the profile parameters increased speed and pressure coefficients. The simulation results of the streamline contours showed largest blockage air flow effects due to the flange at an angle of 0°. Correlation the formation of a wake appear at the rear side diffuser which caused by the formation of vortex flow increased. Velocity vector was given predictions on the direction of stream flow tip losses reduction at the backside of the flange. Quantitative data supported the performance of the diffuser airflow, showed increase in top speed in the interior of the diffuser angle of 90° at 9.82 m/s or 1.96 times freestream velocities, while the static pressure coefficient value is -2.85.

Keywords: Diffuser DAWT, flange, Velocity Profile, static Pressure, CFD.
Halaman ini sengaja dikosongkan