Proceedings of the 6th Annual World Conference
of the Society for Industrial and Systems Engineering,
Herndon, VA, USA
October 19-20, 2017

Aerodynamic Turbines Analysis by Computational Fluid Dynamics Analysis (CFD) with ANSYS Software

S Dominguez Rueda, K Escamilla, and I Escamilla Salazar

Universidad Autónoma de Nuevo León
Facultad de Ingeniería Mecánica y Eléctrica, Ave.
Universidad s/n, Ciudad Universitaria,
San Nicolás de los Garza, Nuevo León, México

Corresponding author's Email: silverioruedа@outlook.com

Abstract: The propeller blades are aerodynamics profiles such as the wings of an airplane, the design and manufacture of these are an engineering work, which looks for maximum performance in every rotation of a propeller (Jeong, 2012). Recently, with the advancement of simulation software, some of the events of this class can be displayed and explained, using CAD software packages, as a researching tool to estimate the flow fields and Turbine performance of new configurations.

In the analysis of previous research, it was concluded, that the energy of a propeller-moto system could be so much better if the angle varies with the blades impact on the air, getting maximum performance during cruise flight (Caboni, 2014). In order to find a final prototype that works optimally and present a remarkable innovation, it is necessary to analyzing different models of propellers taking as a variable the inclination angle and other factors.

Keywords: Turbines, Computational Fluid Dynamics, ANSYS