Proceedings of the 5th Annual World Conference
of the Society for Industrial and Systems Engineering,
San Francisco, CA, USA
October 13-14, 2016

Analysis of Aerodynamic Turbines for Analysis Computational Fluid Dynamics (CFD) with ANSYS

S. Domínguez Rueda, K. Escamilla Salazar, 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

Author Note: In this paper they are reflected efforts by me and my two counselors, they are a great example for me and I have consistently supported the entry into the field of research, at an early stage of my training in my 3rd year of university.

Abstract: The computational fluid dynamics (CFD) is a sophisticated method that uses mathematical equations and algorithms to simulate the flow of fluids, heat and mass transfer, and other related phenomena. In recent years the CFD modeling and simulation technology has grown significantly in the manufacturing sector, because as is often the only means to design, analyze and optimize complex manufacturing processes.

The importance of computational fluid dynamics (CFD) as an important tool for analysis and design of fluid problems today is unquestionable. Using CFD plays an important role in fluid mechanics, due to the progress of different numerical methods and computer capabilities. The design and analysis of different types of systems as in this case a turbine aerodynamics are performed using 3D-Navier Stokes equations, which feed or CFD software program to predict the behavior of the analyzed model.

This paper is intended to carry out an analysis from computational fluid dynamics (CFD) applied to aerodynamic turbines, just taking analysis in ANSYS program from the finite element method FEM.

Keywords: Aerodynamic Turbines, CFD, FEM