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: silveriorueda@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

1. Introduction

The turbomachinery aerodynamic design process has been experiencing continuous progress using the Computational Fluid Dynamics (CFD) tools. The use of CFD-tools opens a new perspective in simulating the complex three-dimensional (3-D) turbomachinery flow field. Understanding the details of the flow motion and the interpretation of the numerical results require a thorough comprehension of fluid mechanics laws and the kinematics of fluid motion within the turbomachinery component. Kinematics is treated in many fluid mechanics texts.

The kinematics is the description of the fluid motion and the particles without taking into account how the motion is brought about. It disregards the forces that cause the fluid motion. As a result, in the context of kinematics, no conservation laws of motion will be dealt with. Consequently, the results of kinematic studies can be applied to all types of fluids and exhibit the ground work that is necessary for describing the dynamics of the fluid.

Apart from these theoretical and experimental endeavors, computational fluid dynamics (CFD) has been a powerful tool and increases in popularity as a method to model different types of fluids with the rapid advancement in computer hardware and physical understanding during the past decades.

Though empirical and experimental methods can generate reliable results with varied influencing factors, they still have their own restrictions: traditional theoretical analyses where calculation objects are always simplified are restricted by the nonlinearities of flow hydrodynamics to get analytical solutions for multiphase systems; experiments are restricted by specific reactor, fluid disturbance, human security and measurement precision.

CFD can provide satisfactory numerical solutions and thus engineers can test various numerical designs and compare the solutions without realistic experiments to find out some optimized proposals and provide more understanding of flow dynamics, heat and mass transfer. Therefore, the budget of realistic experiments could be significantly reduced.

2. Literature Review

The Navier–Stokes equations represent the equations of conservation of mass and momentum for a fluid in motion. However, they are complex, coupled and non-linear, and thus insoluble analytically for anything but the simplest cases. For most realistic cases, computational solution using numerical methods, i.e. CFD, is the only practical approach. Particularly