College of Engineering and Computer Science

Aerodynamic Analysis of a Propeller in a Turbulent Boundary Layer Flow

Felipe Lachowski, Dr. Glegg

Mechanical Engineering; Florida Atlantic University

Simulating the exact chaotic turbulent flow field about any geometry is a dilemma between accuracy and computational resources, which has been continuously studied and compromised for just over a hundred years. This thesis explores numerical procedures and applies them along with the Reynolds' Averaged Navier-Stokes equations and Spalart-Allmaras turbulence model to approximate the otherwise chaotic flow ingested by a Sevik type rotor. The complete CFD process is detailed to provide information at every phase in the development of results ranging for geometry creation to solution post-processing. A multiple reference frame fluid model is utilized by ANSYS - FLUENT throughout this thesis and results compared to experimental data. Three interesting rotor configurations are studied including an open rotor with no flow obstruction, an open rotor near a plate, and an open rotor near a plate with an additional thick boundary layer. Furthermore, studies are made to determine the variation in velocity profiles of the ingested turbulent flow due to varying flow conditions. As a result, this study is indicative that adding a plate near a rotor and allowing a thick boundary layer to be ingested could provide improvements in rotor performance at off peak performance range and possibly reduce noise levels within the entire operational range.