International J.of Multidispl.Research & Advcs. in Engg.(IJMRAE),
ISSN 0975-7074, Vol. 6, No. III (July 2014), pp. 187-200

SIMULATION STUDIES OF AIR FLOW AROUND A NACA 0012 AIRFOIL USING ANSYS FLUENT

N. VEERA PRASAD[1] AND B. VEERABHADRA REDDY[2]
[1]PG student, [2]Professor, Dept. of Mech. Engg.,
G. Pulla Reddy Engineering College (Autonomous), Kurnool.

Abstract

The aim of this work is to evaluate the influence of angle of attack on lift and drag in aerodynamic vehicle. Aerodynamic analysis is done on flap wing using CFD simulations. The typical air speed at high angles of attack for air vehicles involve transitional Reynolds number regime, where laminar separation bubbles appear on the suction surface of the wing. Delaying, eliminating or controlling such separation bubbles are important aircraft design issues and measuring, modeling and predicting them are challenging research problems. The flow control is to manipulate a particular flow field with a small energy input typically aiming to increase lift and reduce the drag force. The NACA 0012 airfoil is a simple geometric model to overcome the boundary layer separation. Employing FLUENT software as the flow solver, the motions of flapping wing are simplified. Standard k-ϵ turbulence model is used for the analysis, will be carried out for different angles of attack.

Keywords: Airfoils, k-ϵ turbulence model, Lift, Drag, angle of attack.