Verification of a Mild-Stall Design Airfoil with ANSYS FLUENT

Janewit Kitsin
Aeronautical Engineering Division
Defence Technology Institute
Bangkok, Thailand
janewit.k@dti.or.th

Pattadon Koanantachai
Aeronautical Engineering Division
Defence Technology Institute

d

***Abstract*—This paper describes an investigation of aerodynamic performance and a Mild-Stall characteristic of the designed airfoil for Fixed-Wing Unmanned Aerial Vehicle (Fixed-Wing UAV). The numerical simulations of Lift force and Drag force of the designed airfoil from numerical results by using Computational Fluid Dynamics (CFD) are compared with an experiment. The computational domain was discretized using an unstructured mesh of 7,100,000 elements. The steady and incompressible flow with a Realizable K-epsilon turbulence model is used in all study cases. The simulations were held at a Reynolds number of 1,000,000. Results show the force coefficients and also the velocity contours and streamlines at different angle of attack, from 0 to 20 degrees. The results support that this designed airfoil can achieve the Mild-Stall characteristic. Lift force is decreased slightly in the mild-stall region after the angle of attack generated the maximum Lift force. This can extend the range of flight lift coefficient and help to provide a safe flight in gusty wind.**

Keywords*—airfoil; wing; mild-stall; unmanned aerial vehicle; CFD

I. INTRODUCTION

Unmanned aerial vehicles (UAVs) have been produced and performed their operations in military and civil aviation industries for a past decade. The UAVs are designed in various models and sizes in order to achieve their purpose. The aerodynamic performance of the UAVs is depended by the shape of their fuselage and the shape of their airfoil. The overview of the design activity of civil c (high altitude long endurance) UAV (PW114) is provided by Goraj [1]. He reported that the cost of operation could be reduced by improvement of aerodynamic efficiency and optimization of aircraft structures. He redesigned the previous model (PW111) and developed a new airfoil shape with high aspect ratio via aerodynamics analyses for increasing the lift force meanwhile the drag force was reduced.

The study and development of the aerodynamics designed has played an important role in the UAVs optimization. There are many tools to design and investigate the flow phenomena and aerodynamic behaviors of the UAVs like Wind Tunnel [2,3] or Simulation programs [4]. In the present time, Computational Fluid Dynamics (CFD) has come to complement the experimental studying. CFD is the analysis of systems involving fluid flow and related phenomena by means of computer-based simulation. It can reduce cost in the tests and time for the generation of prototypes. The aerodynamic performance, lift force and drag force, can be accurately predicted by using a commercial CFD package as ICEM CFD and ANSYS FLUENT. The CFD simulation can also show the features and details that are difficult, expensive or impossible by measuring and visualizing in an experiment.

II. MATHEMATICAL MODELING

For a numerical simulation of fluid flows, the mass conservation equations and the rate of change of momentum and energy equations are solved by the Navier-Stokes equations. In case of turbulent flow, transport equations are also solved additionally. Reynolds-averaged Navier-Stokes (RANS) equations are generally applied for solving flow characteristics. The concept of RANS model is assumed that the turbulent scales of fluid motion are very small compared with the mean flow around the object and are included among addition unknown terms called Reynolds Stress equations [5,].

Fig. 1. Unmanned aerial vehicle (UAV)

The continuity equation for steady and incompressible flow is given by: