Computation fluid dynamics study on aerodynamics blade design of Advance Precision Composite (APC) slow flyer propeller

ABSTRACT

The current work presents a numerical method prediction using the commercial Computational Fluid Dynamic (CFD) of ANSYS FLUENT in order to determine the best propeller design which produce the nearest aerodynamics performance with the experimental data of a small-scale propeller. This study implements 2 different propeller design by changing the airfoil origin position (AOP) at each of blade design station which create the different design of propeller shape. The changing of AOP is referred in terms of percentage of 0% AOP and 25% AOP of the respective airfoil chord from the hub of the propeller blade. The study utilizes a tetrahedron meshing throughout the analysis with different turbulence models comparison. Multiple Reference Frame (MRF) technique was used to create the rotation of the propeller towards its local reference frame at 3008 revolutions per minute (RPM). The result of thrust, power coefficients and the efficiencies are validated with the experimental wind tunnel data. The results showed that the model with 25% AOP generated the nearest amount of thrust, power, and efficiency when compared to experimental data with 0.82%, 6.68% and -5.49% for lowest advance ratio of 0.236. Hence, using novel technique of CFD analysis provide a better platform in designing the best aerodynamics propeller blade design before fabricate the actual model of propeller.

Keyword: APC slow flyer; CFD; Multi reference frame; Propeller; Propeller blade; Unmanned Aerial Vehicle (UAV)