

CFD analysis of a UAV plane

Mukundjee Pandey^{1*} and Jagadish Dash²

¹Mechanical Engineering Department, Centurion University of Technology and Management, Odisha, India

²Bachelor of Technology, Mechanical Engineering, Centurion University of Technology and Management, Odisha, India

*For Correspondence: mukundjee.pandey@cutm.ac.in

ABSTRACT:

Unmanned aerial vehicles (UAVs) are playing major roles in various application of this cutting-edge technology. UAVs can be operated without the physical presence of a human being. In this article, 3D Experience, Simulia software is used to analyze the UAV using CFD techniques. The 3D model was designed in CATIA and CFD analysis was conducted with the help of Simulia software. Investigation is displayed in steady state 3D computational fluid elements (CFD) at variation of velocity from 80km/hr. to 500km/hr.

Keywords: UAV, CFD analysis, drone, plane

INTRODUCTION:

Yakinthos et al. presented different phases of aerodynamic design for the aerodynamic design of fixed-wing UAVs. The CFD simulation was done using BETA CAE software coupled with the CFX flow solver to find out the stability of UAV[1]. Ebubekir et al. investigated UAV design, analysis and production incorporating SLS and CFD analysis[2]. Pastor et al. studied a hardware/software architecture designed to operate as a flexible payload and mission controller in a mini/micro UAV[3]. Mairaj et al. described the current challenges and issues in research for drone simulators[4]. Atmaca et al. presented UAV model and its cfd analysis was conducted[5]. Carvalho presented low subsonic aerodynamic theories, through CFD analysis, in order to optimize the main wing of a MALE UAV[6]. Naresh et al. described lift and drag coefficients with different Mach number, velocity and pressure with help of *ANSYS ICEM CFD analysis*[7]. Suresh et al. represented drone which was capable to travel in both air and water[8]. Manikantissar et al. presented CFD analysis of delta wing UAVs with help of ANSYS CFX software[9]. Khuntia et al. described advantages of CFD analysis to analyse strength and stiffness of UAV wing. CFD was done with help of ANSYS software [10]. Sreelakshmi et al. investigated CFD analysis at different Mach numbers to calculate drag, lift coefficients of UAVs [11]. Kumar et al. studied NACA 651-212 model in wind tunnel and its simulation was performed in Ansys Fluent. Both the simulation and experimental results were compared [12].

PROCEDURE: