International Journal of Technical Innovation in Modern Engineering & Science (IJTIMES)
Impact Factor: 3.45 (SJIF-2015), e-ISSN: 2455-2585
Volume 4, Issue 5, May-2018

Computational Analysis of a UAV and Developing an Improved Design to Enhance its Performance by Reducing Drag Force

Disha Dewangan[1]
[1] *Department of Industrial & Production Engg., Institute of Technology Guru Ghasidas Vishwavidyalaya,*
Dewangan.disha@gmail.com[1]

Abstract— ***The present work aims to investigate and optimize the design of a self-propelled Unmanned Arm Vehicle namely UAV by using Ansys Fluent for its analysis. Design of the chosen UAV was based on HAVOC RC which is commercially available online. It has a high-wing configuration with zero dihedral angle, propelled by a single thruster powered by a Brushless DC Electric Motor and controlled wirelessly by Radio Transmitter and Receiver. The multipurpose Airplane was modeled by using Creo Parametric (Pro-E) and later on analyzed for its performance on Ansys Fluent. The modifications in the design of critical parts like wing tips, fuselage were done and later on analysis has done in order to improve its aerodynamic efficiency. It was found that the efficiency for cruise flight is better in the optimized version than in the original design. With a fewer modifications over a design the aerodynamic performance of an aircraft can be significantly improved.***

Keywords: Computational Fluid Dynamics (CFD), Drag, Lift, UAV, Aerodynamic Efficiency.

INTRODUCTION

In past two decades extensive researches were carried out to develop the UAVs field and it can be utilized for different missions such as surveillance and reconnaissance. Airplane has three sources of drag: (i) profile drag which is related to skin friction caused by flow of air over the aircraft surface (ii) induced drag which is the result of lift generation for finite wingspan and (iii) the compressibility effect caused by high speed aerodynamics. Here, profile and induced drag are related to wing shape. Thus wing shape can play an important role in the reduction of drag. Besides various wingtip devices like winglet can bring added advantage to the wing shape. The function of winglet is to reduce induced drag. The drag breakdown of a typical transport aircraft shows that the lift-induced drag may become as much as 40% of the total drag at cruise conditions and 80–90% of the total drag in take-off configuration. Hence, reducing induced drag will make the system more fuel economic.

Method

HAVOC RC UAV

The present work is based on optimization of design of a sailplane by Havoc RC. It has the following specifications:

Type: Sailplane	
Dimensions	
Wingspan	1 m
Length	0.965 m
Height	0.2 m
Weight	
Maximum weight	1.8 kg
Wing Characteristics	
Airfoil Profile	NACA S7055
Wing Span (b)	1 m
Mean Chord Length (C)	0.183 m
Aspect Ratio (b/C)	5.464
Flight Characteristics	
Density of Flow	1.225 kg/m^3
Flow velocity (V)	20 m/s max.
Mach Number (M=V/c)	0.058
Flow type	Incompressible
Propulsion	
Brushless D.C. Motor	1800 kV
Maximum Current	30 A

As it is easier to control, slow and stable, the plane is used by beginner hobbyist to learn to fly drones.

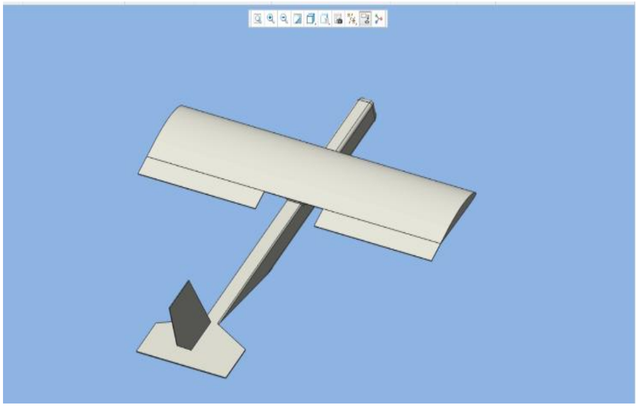

Fig. 1 Havoc RC UAV

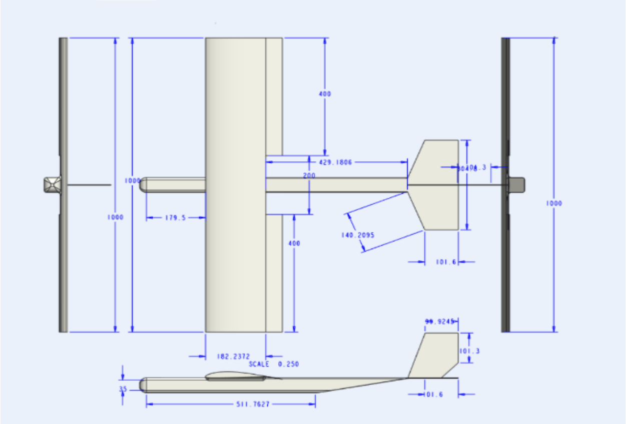

Fig. 2 Complete Plan

Mathematical Model

Calculation of Flight Loads

Analysis is performed with the ANSYS CFX solver, according to the turbulence and Navier stroke equation methods for 100 iterations. In this study an upstream velocity of 34.3 m/s & 65.17 m/s is specified at the inlet boundary. All other boundary conditions remain the same as wall. In order to determine the necessary loads, the Mach no, temperature and velocity of the UAV is to be studied and the lift and drag calculation is performed.

1) Mach number: The Mach number (M or Ma) is a dimensionless quantity gives the ratio off-low velocity past over the boundary in the domain to the speed of the sound.

$M=V/c$

Where,

M- Mach number.

V- Flow velocity over boundaries.

c- Speed of sound = $(\gamma * R * T)^{0.5} = 340$ m/s

Where,

γ is the density of air 1.4

R is the gas consistent 287 J/kg K

T is the temperature of air (25°C) in K

2) Drag Force

The drag force acting on a body in fluid flow can be expressed as

$FD = 1/2\, C_D \rho V^2\, A$………………….. (1)

3) Lifting Force

The lifting force acting on a body in a fluid flow can be expressed as

$F_L = 1/2\ C_L\ \rho V^2\ A$ ………………….. (2)

Where,

F_D = drag force (N),

C_D = drag co-efficient,

F_L = lifting force (N),

C_L = lifting co-efficient,

ρ = density of fluid (kg/m^3),

V = flow velocity (m/s),

A = body area (m^2).

CFD Numerical Study

The Numerical simulations used a model of the Havoc RC prepared on CAD software Creo Parametric (Pro-E) which was later imported to ANSYS Fluent software for CFD analysis. Where the model was meshed using Orthogonal mesh with quality ranging from 0 to 1. The total number of elements were found to be 1128375 and the total number of nodes were 200619.

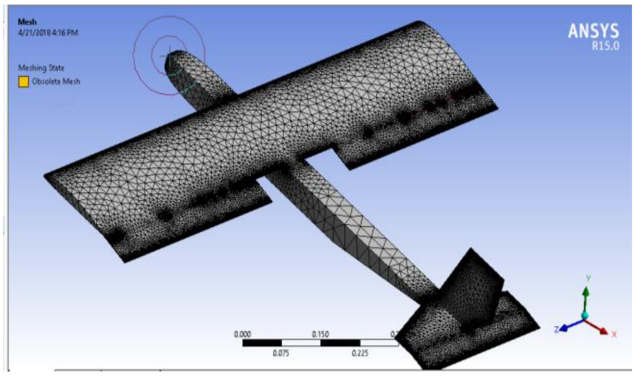

Fig. 3 Meshing

The CFD simulation is defined using k-ε model since airflow turbulence around the body. The transport equations for the standard k-ε model are expressed as follows:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b - \rho\epsilon - Y_M + S_k$$

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}(\rho \varepsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_{1\varepsilon}\frac{\epsilon}{k}(G_k + C_{3\varepsilon} G_b) - C_{2\varepsilon}\,\rho\frac{\varepsilon^2}{k} + S_\varepsilon$$

The boundary conditions were specified as:

M= 0.058

V=20 m/s

Angle of attack (α): 10 ^{o}

Density of Flow: 1.225 kg/m^3

Frontal Area= 0.667 m^2

Design Modification

As we know that a significant improvement in lift and drag characteristics can be improved by incorporating the following changes in the design

(i) Decreasing the aspect ratio (b/C) of the wings.

(ii) Rounding off the edges and making the cross-section of fuselage as round as possible.

The new Design is as shown:

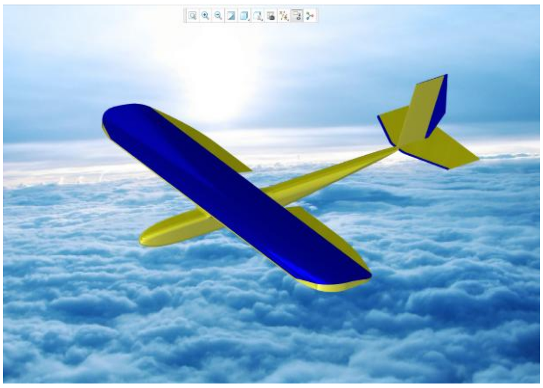

Fig. 4 Modified Havoc RC

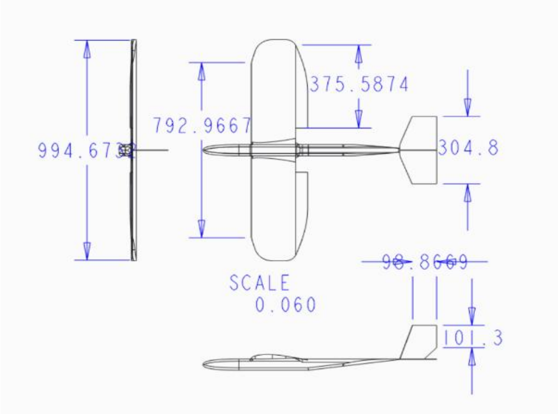

Fig. 5 Detailed Plan of Modified Model

The new design was again modelled by Creo Parametric and then geometry was imported to Ansys for CFD analysis, where it was meshed again and analyzed. Both models were compared for results on parameters such as coefficient of lift, coefficient of drag and pressure distribution.

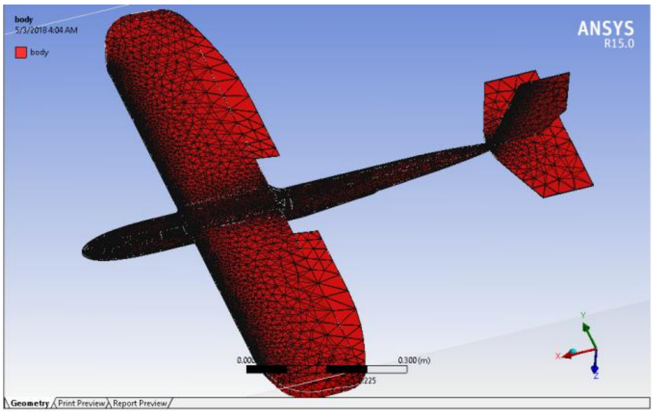

Fig. 6 Meshed Structure of Modified Design

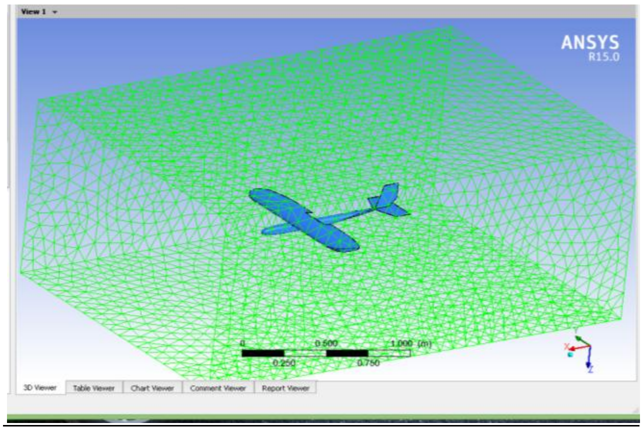

Fig. 7 Boundary around the geometry

Result

Fluent analysis of aerodynamic performance for full-scale Havoc RC in both cases were carried out for both original and modified version in the positive x-direction of the airflow. Lift and drag coefficient was measured based on aircraft at steady state level. Data of C_d were calculated and compared for both the UAVs. Result is summarized in the table:

Table 1.Simulation Aerodynamic Characteristic Result

Havoc RC		
	Original Design	**Modified Design**
Drag Coefficient (C_d)	0.8032	0.73985
Drag Force, F_D (N)	13.0527	12.071328

It has been found that the modified design has an optimized aerodynamic performance as compared to the original design as the drag force is reduced. This reduction in the Drag force has been achieved by modifying the design of the wing as well as that of the fuselage. The drag force has been reduced by 7% in modified design.

References

Sakhar Abudarag, Rashid Yagoub, Hassan Elfatih, Zoran Filipovic, “Computational Analysis of Unmanned Aerial Vehicle (UAV),” ICNPAA World Congress, AIP Conf. Proc. 1798, 020001-1–020001-10; doi: 10.1063/1.4972593J.

Mohsen Jahanmiri, “Aircraft Drag Reduction: An Overview” Research Report 2011, Research gate

S.M. Fraser, C. Carey, A.A.A. Moustafa, “Numerical and Experimental Analysis of Flow Around Isolated and Shielded Cubes” Appl. Math. Modelling, 1990, Vol. 14, November

S.A. Kalugade, Sangram Pawar, Suraj Chawan, “Aeroplane fuselage design modification to Overcome Pressure Drag resistance” Journal of Advances in Science and Technology March-2017

Kenji Tadakuma, Yasuhiro Tani, Shigeru Aso, “Effect of Fuselage Cross Section on Aerodynamic Characteristics of Reusable Launch Vehicles” Open Journal of Fluid Dynamics, 2016, 6, 222-233

R. Saim, S. Mohd., S.S. Shamsudin, M.F. Zulkifli, Z. Omar, “Computational Fluid Dynamic (CFD) analysis on ALUDRA SR-10 UAV with Parachute Recovery System” IOP Conf. Series: Materials Science and Engineering 243(2017) 012014