

DESIGN AND DEVELOPMENT OF SLAT FOR STOL FIXED WING UAV

Vijay P Sulakhe

Vehicles Research & Development
Establishment (VRDE),
Defense Research & Development
Organization (DRDO)
Ahmednagar, India
vpsulakhe@vrde.drdo.in

Nikhil B Nagare

Vehicles Research & Development
Establishment (VRDE),
Defense Research & Development
Organization (DRDO)
Ahmednagar, India
nikhilnagare7@gmail.com

ABSTRACT

A Slat is a device attached to leading edge of wing, used to improve aircraft aerodynamic efficiency by lowering separation of flow over wing and increasing the stall angle [7]. The need of high lift devices is to decrease the take-off and landing distance. Military forces acting in remote areas require unmanned aerial vehicles (UAV) for surveillance but main limitation is requirement of long runways, hence they are unable to use UAVs. In this paper it is demonstrated that UAV with slat requires short runway for operation. The computational simulation was carried out by commercial CFD software using Finite Volume Approach. The simulation was done at low subsonic flow and at various angles of attack using K- ω SST solver [3]. A comparison of aerodynamic characteristics of lift coefficient CL, drag coefficient CD and lift to drag ratio, L/D was made and it was found that the addition of the slat gave a larger lift curve slope and higher Lift-to-Drag Ratio in comparison to the baseline wing alone. Flight testing is done to validate the results obtained from CFD analysis. Range and endurance are also enhanced.

Keywords

Unmanned Aerial Vehicle (UAV), Slat, High lift devices, Aerodynamics, CFD, Autopilot.

1. INTRODUCTION

High-lift systems such as slats and flaps play an important role in aircraft performance and they are widely used during takeoff and landing. Despite its important roles, conventional high-lift systems have some issues that affect the aircraft performance. Firstly, a high-lift system is a part of multi element airfoil where it has three elements; one element is the main body and two elements are the high-lift systems. Range and endurance of subsonic UAVs are defined, and often limited, for the performance of their high lift devices. These systems are generally quite complex, consisting of a leading-edge slat and two or three trailing-edge mini-flaps. High maintenance cost and their weight bring impulse to the designers to look for lighter devices with less mechanical complexity, without diminishing the performance regarding the classical ones. Nevertheless, to reach high CL during take-off, approximation and landing will be necessary new

developments in order to increase the lift and to control boundary layer separation. One of such possible technologies is the Slat.

Flow control was object of many investigations since Prandtl developed its boundary layer theory in 1904 [14]. On the recent years such research was motivated for developments in the fields of micro and nano electro-mechanics, materials and control looking for achieves successfully control in aerial vehicles without the employ of classical control surfaces.

In this paper, the design of airfoil is optimized by reducing the three-element airfoil into two-element airfoil. The slat and main body are combined. In our experiments a NACA 4412 airfoil with slat was tested by using Commercial CFD code and actual flight testing of slat mounted UAV. Autopilot Arduino 2.5 is used to collect flight data. Data obtained from autopilot is used for comparison of computational data and testing data.

The paper is organized as follows: Section II presents methodology. Section III discusses the numerical simulation, Section IV computational setup of slat-airfoil configuration. Section V details the simulation and flight testing results obtained for slat mounted UAV. Finally, Section VI proposes concluding remarks.

2. METHODOLOGY

In this work, first of all a generic model of the slat mounted on NACA 4412 airfoil is prepared in the CATIA V5 software. The geometry parameters are represented in the following figure:

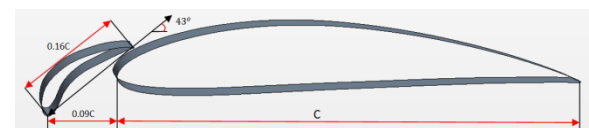


Fig.1. Slat Geometry

and this generic model is imported into the Star ccm+ to do the simulation of the coefficient of drag and coefficient of lift in the tunnel which is generated in the design module of the Star ccm+. After this the overset meshing technique is used to generate mesh on the surface of CAD

model [1] [2]. Aerodynamic evaluation of air flow over an object can be performed using analytical method or CFD approach. On one hand, analytical method of solving air flow over an object can be done only for simple flows over simple geometries like laminar flow over a flat plate. If air flow gets complex as in flows over a bluff body, the flow becomes turbulent and it is impossible to solve Navier-Stokes and continuity equations analytically.

On the other hand, obtaining direct numerical solution of Navier-stoke equation is not yet possible even with modern day computers. In order to come up with reasonable solution, a time averaged Navier-Stokes equation is being used (Reynolds Averaged Navier-Stokes Equations – RANS equations) together with turbulent models to resolve the issue involving Reynolds Stress resulting from the time averaging process. In present work the SST k- ω turbulence model is selected to analyze the flow over the slat model. This k- ω SST turbulence model is very robust, having reasonable computational turnaround time, and widely used by the aerospace industry.

Slat mounted aircraft is tested to verify the results obtained from computational approach. Possible effects on aircraft performance are calculated and compared with aircraft without slat[15].

3. NUMERICAL SIMULATION

The numerical simulation was carried out using a constant pressure-based solver and Transition SST (2 equations) turbulence model, which was shown to perform better than Spalart-Allmaras in our early study [4]. The governing equations used a second-order upwind discretization and solved using the simple algorithm. The calculations are done in the range of angles of attack from 0° to 45° at Re=5.25x10E5.

3.1 Governing equations

Based on the Reynolds-averaged Navier-stokes equation, the governing equations can be given as:

Continuity equation,

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i) = 0$$

Momentum equations,

$$\frac{\partial}{\partial t} (\rho u_i) + \frac{\partial}{\partial x_j} (\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j}$$

Energy equation,

$$\frac{\partial}{\partial t} \left[\rho \left(h + \frac{1}{2} u_i^2 \right) \right] + \frac{\partial}{\partial x_j} \left[\rho u_j \left(h + \frac{1}{2} u_i^2 \right) \right] = \frac{\partial p}{\partial t} + \frac{\partial}{\partial x_j} \left(u_i \tau_{ij} + \lambda \frac{\partial T}{\partial x_j} \right)$$

where τ_{ij} is the stress tensor and h is total enthalpy given by

$$\tau_{ij} = \mu \left(\frac{\partial u_i}{\partial x_j} + \lambda \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \mu \frac{\partial u_i}{\partial x_i} \delta_{ij}$$

$$h = c_p T$$

Ideal gas state equation,

$$P = \rho RT$$

The finite volume method is used to solve the two dimensional unsteady Reynolds-averaged Navier-Stokes equations. The momentum equations are discretized using second order upwind scheme options. Viscous terms are discretized using central differences. Velocity and pressure are coupled and solved in all computations. The temporal integration is accomplished with one order implicit scheme. The velocity was set to zero on the airfoil surface. The ground was set as a moving wall with the velocity same as free air to simulate the flight more accurately. A pressure far-field boundary condition is used for the far field flow condition.

3.2 Turbulence modeling

To close the Reynolds-averaged Navier-Stokes equations, SST k- ω SST turbulence model was used in this study which can effectively blend the robust and accurate formulation[6]. The k- ω SST model by Menter is a hybrid of the original k- ω model by Wilcox, and the standard k- ω model by Jones and Launder. It is known that the Wilcox k- ω model is superior to the k-epsilon model in wall bounded flows. The k- ω model does not involve damping functions at the wall and allows simple Dirichlet boundary conditions to be specified. This simplicity makes the model more numerically stable than other two -equation models. Furthermore the behavior of the k- ω model in the logarithmic region is superior to that of the k-epsilon model in equilibrium adverse pressure gradient flows. The k and ω transport equations are:

$$\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_j} (\rho k u_j) = \tau_{ij} \frac{\partial u_i}{\partial x_j} - \beta^* \omega k + \frac{\partial}{\partial x_j} \left[\left(\mu + \sigma_k \mu_t \right) \frac{\partial k}{\partial x_j} \right]$$

$$\frac{\partial}{\partial t} (\rho \omega) + \frac{\partial}{\partial x_j} (\rho \omega u_j) = \frac{\gamma}{v_t} \tau_{ij} \frac{\partial u_i}{\partial x_j} - \beta \rho \omega^2 + \frac{\partial}{\partial x_j} \left[\left(\mu + \sigma_\omega \mu_t \right) \frac{\partial \omega}{\partial x_j} \right] + 2(1 - F_1) \rho \sigma_{\omega 2} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}$$

The constants ϕ of the model are calculated from the constants ϕ_1 and ϕ_2 as follows:

$$\phi = F_1 \phi_1 + (1 - F_1) \phi_2$$

The constants of set 1 (ϕ_1) applied to flow in the boundary layer are:

$$\sigma_{k1} = 0.85, \sigma_{\omega 1} = 0.5, \beta_1 = 0.075, \beta^* = 0.09,$$

$$k = 0.41, \gamma_1 = \beta_1 / \beta^* - \sigma_{\omega 1} k^2 / \sqrt{\beta^*}.$$

The constants of set 2 (ϕ_2) applied to flow in the boundary layer are:

$$\sigma_{k2} = 1.0, \sigma_{\omega 2} = 0.856, \beta_2 = 0.0825, \beta^* = 0.09,$$

$$k = 0.41, \gamma_2 = \beta_2 / \beta^* - \sigma_{\omega 2} k^2 / \sqrt{\beta^*}.$$

The blending function F_1 is defined as

$$F_1 = \tanh(\arg_1)^4$$

$$\arg_1 = \min \left[\max \left(\frac{\sqrt{k}}{0.09 \omega y}, \frac{500 v}{\omega y^2} \right); \frac{4 \rho \sigma_{\omega 2} k}{CD_{k\omega} y^2} \right]$$

$$CD_{k\omega} = \max \left(2 \rho \sigma_{\omega 2} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}; 10^{-20} \right)$$

where y is the distance to the solid surface.

The Eddy viscosity is defined as

$$\nu_t = \frac{\alpha_1 k}{\max\left[\frac{u_1 k}{\Omega F_2}, \alpha_1\right]}; \quad \alpha_1 = 0.31$$

where Ω is the absolute value of the vorticity [10]

$\Omega = \left| \frac{\partial u}{\partial y} \right|$ and F_2 is given by

$$F_2 = \tanh(\arg_2^2)$$

$$\arg_2 = \left(\frac{2\sqrt{k}}{0.09\omega y}; \frac{500\nu}{\omega y^2} \right)$$

4. COMPUTATIONAL SIMULATION SET UP

The assumptions made in present simulation have the air flow is steady state with constant velocity at inlet, constant pressure outlet, no slip wall boundary conditions at the model surfaces, and in-viscid flow wall boundary condition on the top, sidewalls and ground face of the virtual wind tunnel.

The solver was segregated flow 2nd order convection method for pressure-velocity linked equations algorithm. This algorithm is an iterative procedure for solving equations for velocity and pressure, for steady-state and the under relaxation factors for velocity and pressure are set as 0.7 [5] and 0.3 respectively and for the turbulent kinetic energy, turbulent viscosity is set to 1. For the discretization the pressure was kept as standard, while the other parameters Momentum, Turbulent Kinetic Energy, Turbulent dissipation Rate and Energy were retained as Second Order Upwind [11]. Monitoring the convergence during the solution was dynamically checked by force coefficient values and convergence through residuals. The data were printed, reported and displayed in plots of lift, drag, and moment coefficients, and residuals for the solution variables. In the Force Monitors, the Force vectors Lift and Drag had to define with relative to the free stream direction.

When calculating force coefficients the reference values should be given to get the actual results. So the Reference Values of area, length, pressure, density, temperature and velocities were given as 0.003 sq. m., 0.3 m, 101325 Pa, 1.225 kg/cu.m., 300 K, 25 m/s (subsonic Mach 0.07).

The numbers of iterations were set to 1000 to be performed in the Number of Iterations field. Then the Star-ccm+ began with the calculations starting at iteration 1, using the initial solution. Then the graphs were plotted, printed and written (in separate data files). When the observations in the graphs of Coefficient of Lift vs iterations and Coefficient of Drag vs iterations were converged, the simulations were stopped.

5. RESULTS

5.1 CFD Simulation

Results showing the flow field developments are presented as velocity vector, static pressure contour and pressure coefficient plots using Star ccm+ as post processors. Further, aerodynamic characteristics such as lift and drag coefficients are also presented as obtained from the resulting pressure and shear stress distributions at different flight conditions [12]. Conventional NACA 4412 airfoil and the modified airfoil configuration were evaluated for a given set of inlet conditions.

The operating conditions matching the standard atmospheric values of pressure, temperature, and density were kept the same for all the cases during the study.

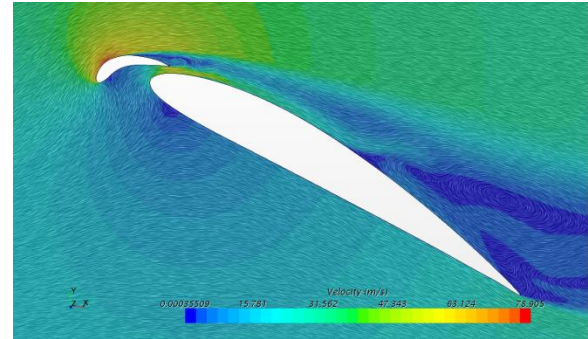


Fig.2. Velocity contour Slat-NACA4412 at $\alpha = 30^\circ$

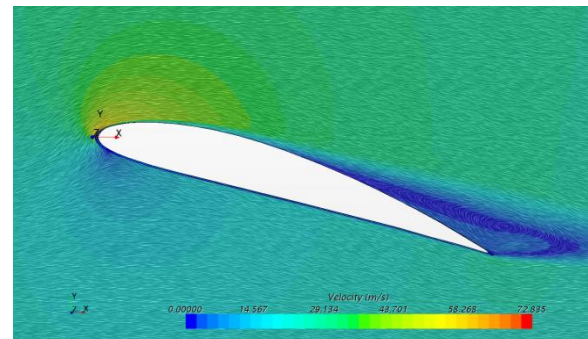


Fig.3. Velocity contour NACA 4412 at $\alpha = 15^\circ$

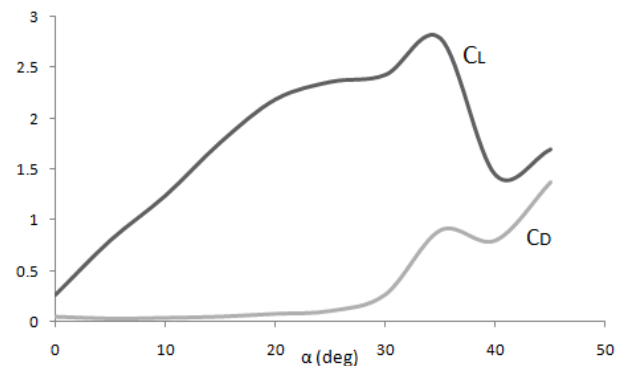


Fig.4. Variation of C_L and C_D with AOA (CFD results)

5.2 Flight tests

Low altitude flight tests of this configuration have been conducted. These tests have been conducted with three goals in mind: (i) to evaluate aerodynamic performance of the Slat mounted wing in realistic operating conditions, (ii)

determine the handling characteristics of the aircraft, and (iii) to obtain appropriate feedback gains for use in an autopilot system.



Fig.5. Flight Test



Fig.6. Slat mounted wing close view

To achieve altitude, the vehicles were outfitted with an electric power-plant mounted in a tractor configuration. The motor was Hacker 348 revolutions per Volt brushless motor with a Avionic controller and 6 cell battery providing up to 30 A of current. At an output of 6 cells and with a rated motor efficiency of greater than 82%, this provided up to 615 W of power to the propeller. Various sized Graupner CAM folding propellers were used as needed.

Numerical simulations were conducted using a conventional NACA 4412 airfoil considered as the base airfoil and the modified airfoil as an airfoil with slat. Results obtained show that lift coefficients are higher for the stepped airfoil than the base airfoil and the magnitude of the change in C_L increased with increase in the angle of attack. On the other hand, the drag values are higher for the slat airfoil. Based on the C_L and C_D values obtained for each of the cases, the take-off distance is calculated using the following formula:

$$S_{TO} = \frac{1.44 W_0^2}{g \rho S C_{L_{max}} [T - (D + \eta_r (W_0 - L))]}$$

The cruise Reynolds number is 5.25×10^5 , the velocity being 25 m/s. The use of slat yields about 66% higher C_L than the base airfoil and the L/D ratio increases by 5%. Fig.4. shows plot of variation of C_L and C_D with α . This increase in the L/D ratio yields about 20% increase in the endurance (E).

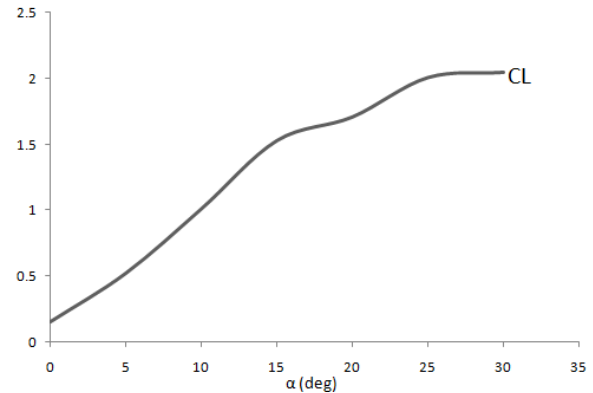


Fig.7. Variation of C_L with AOA (Flight test results)

$$R = \left(\frac{\eta}{C_s}\right) * \left(\frac{C_L}{C_D}\right) * \ln\left(\frac{W_0}{W_1}\right)$$

$$E = \left(\frac{\eta}{C_s}\right) * \left(\frac{C_L^3}{C_D}\right) * (2\rho S)^{\frac{1}{2}} * \left(W_1^{-\frac{1}{2}} - W_0^{-\frac{1}{2}}\right)$$



Fig.8. Screenshot of HUD on ground control Station

η was assumed to be 0.8 for all the calculations while c_s was assumed to be 2.27×10^{-7} per ft. Range was computed for both the base and slat airfoil configurations. The ratio $R_{\text{slat-airfoil}} / R_{\text{base airfoil}}$ was calculated to be around 1.1 which means use of slat during the cruise results in an increase of about 10% over the conventional NACA 4412 airfoil based wing originally used on the UAV.

The UAV has been test flown in this configuration as seen in Figure 5. Figure 8 shows the result of one of the flight test before stalling using the onboard sensor.

6. CONCLUSION

Computational fluid dynamics (CFD) simulations of the steady flow field around airfoil models with and without slat were presented and compared the simulated data to flight test. The Star ccm+ with the $k-\omega$ SST steady model is used for the simulations of aerodynamics. In this analysis, the coefficient of lift is increased by 66% and takeoff distance has been reduced by 40%. Lift curve slope increases from 0.068 to 0.096 due to slat. Hence, the Slat is the effective tool to reduce the runway length requirement, increases stall performance and decreases Landing speeds. Fig.2 shows flow is attached to upper surface due to slat. The effects of different aerodynamic high-lift devices on flow and its structure over a aircraft may be analyzed using CFD approach[8]. The objective is to increase aerodynamic lift acting on the aircraft and thus improve the overall efficiency of aircraft. Hence, the Lift force can be increased by using add-on devices on aircraft and flight performance of a aircraft can be improved.

7. ACKNOWLEDGMENT

The authors wish to thank Mr. H.M. Jadhav and Mr.A.S. Elumalai for their assistance during flight trials. We also thank to Maj. Gen. Ajay Gupta, Director, VRDE for giving us an opportunity to undergo research activities.

8. REFERENCES

- [1] Anderson, W.K. and Bonhaus, D.L., Airfoil design on unstructured grids for turbulent flows. AIAA J.37(2) (1999) 185–191.
- [2] Rogers, S.E., Roth, K., Nash, S.M., Baker, M.D., Slotnick, J. P., Cao, H.V., and Whitlock, M., \ Advances in Overset CFD Processes Applied to Subsonic High-Lift Aircraft," AIAA Paper 2000-4216, Aug. 2000.
- [3] Eppler, R, and D. M. Somers, 1980a, "A Computer Program for the Design and Analysis of Low-Speed Airfoils," NASA TM-80210.
- [4] Menter F R. Two-equation eddy-viscosity turbulence models for engineering applications. AIAA Journal, 1994, 32(8):1598-1605.
- [5] Moitra, A., Jun 2001, Validation of an Automated CFD Tool for 2-D High-Lift Analysis, AIAA; 2001-2401.

- [6] Cao, H. V., Su, T. Y., and Rogers, S. E., "Navier-Stokes Analysis of a 747 High Lift-Configuration," AIAA Paper 9812623, July 1998.
- [7] Whitcomb, R. T., "Methods for Reducing Aerodynamic Drag", NASA Conference Publication 2211, Proceedings of Dryden Symposium, California 1981.
- [8] Meredith, P.T. Viscous Phenomena Affecting High-Lift Systems and Suggestions for Future CFD Development, "High-Lift System Aerodynamics, AGARD CP-515, Paper No.19, Sept.1993.
- [9] Jameson, A., Aerodynamic design via control theory. J. Sci. Comput.3(1988) 233–260.
- [10] Wilcox, D. C., 1994, Turbulence Modeling for CFD, DCW Industries, Inc., La Cañada, CA.
- [11] Reuther, J., Jameson, A., Alonso, J.J., Remlinger, M.J. and Saunders, D., Constrained multipoint aerodynamic shape optimization using an adjoint formulation and parallel computers, Part 1. J. Aircraft 36(1) (1999) 51–60
- [12] Rogers, S. E., "Progress in High-Lift Aerodynamic Calculations," AIAA Paper 93-0194, Jan. 1993. Also published in the Journal of Aircraft, Vol31, No. 6, Nov. 1994, pp 1244- 1251.
- [13] Maughmer, M. D., Timothy, S. S., and Willits, S. M., "The Design and Testing of a Winglet Airfoil for Low-Speed Aircraft", AIAA Paper 2001-2478, 2001.
- [14] 'Flight Theory and Aerodynamics: A Practical Guide for Operational Safety' By Charles E. Dole, James E. Lewis.
- [15] Ta'asan, S., Kuruvila, G. and Salas, M.D., Aerodynamics design and optimization in one shot. AIAA Paper 92-0025 (1992).