



## **INVESTIGATION OF PERFORMANCE OF A PROPELLER BLADE AEROFOIL USING COMPUTATIONAL FLUID DYNAMICS**

Aditya R Bhat

Department of Electronics and Communication Engineering

BMS College of Engineering

Bull Temple Road, Bengaluru, Karnataka, 560019

INDIA

---

**Abstract:** This paper aims to design and carry out an analysis of a propeller blade aerofoil and to study its velocity and pressure variations at subsonic speeds. The designed aerofoil is symmetric. The design and meshing processes of the aerofoil were done using AcuSolve software. The design that was generated was analysed using HyperView. The simulation process was carried out over a span of a thousand iterations. The velocity and the pressure plots were mapped.

**Keywords:** Aerofoil, CFD, Fluid flow, Velocity, Pressure.

---

### **I. Introduction**

Computational Fluid Dynamics, abbreviated as CFD, is a branch of fluid mechanics which makes use of algorithms and various numerical methods to solve and perform analyses of problems involving fluid flow. It is an area of fluid mechanics which deals with predicting fluid flow patterns, heat transfer, mass transfer, and such other parameters and phenomenon by solving the mathematical equations that govern these processes numerically. CFD provides mostly qualitative predictions of fluid flow by means of mathematical modeling using partial differential equations, numerical methods using discretization and solution techniques and various software tools. CFD gives an insight into flow patterns that are difficult, expensive or impossible to study using traditional (experimental) techniques. From a practical point of view, the usage of CFD simulation tools is very favourable over experimental methods. This is because CFD software can be used to study the flow for all desired quantities, and in high resolution in space and time for virtually any problem with realistic operating conditions. It can also be used to analyze the actual flow domain unlike in the experimental set ups where a laboratory scale model has to be built for analysis. But, it has to be noted here that any aerodynamic component design is completed by making use of the data acquired by both the Experimental and Simulation methods used to study fluid flow. The science of CFD leads to different technological areas which include grid generation, flow field discretization algorithms, efficient solution of large systems of equations, massive data storage and transmission technology methods and computational flow visualization.

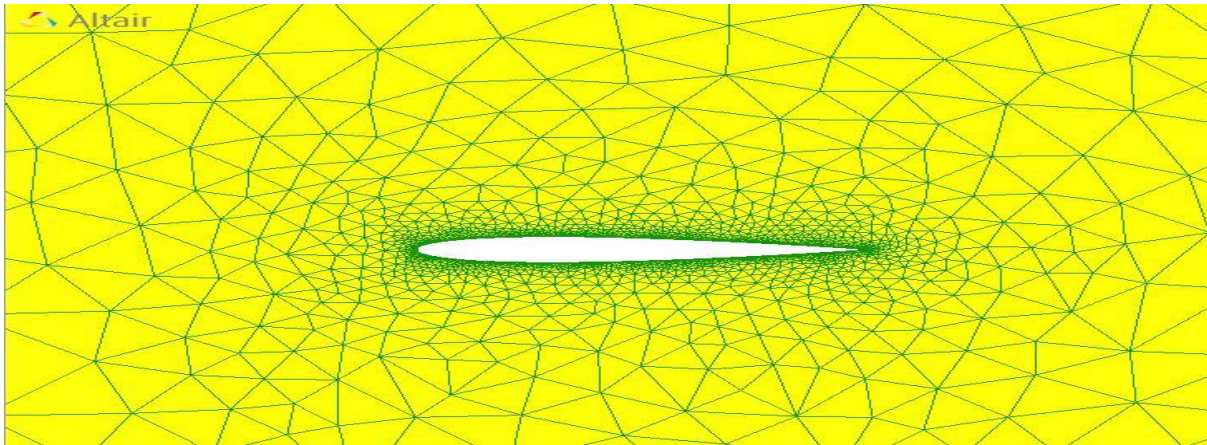
The CFD software performs the simulation operation in three steps. Firstly, the Pre processor where the problem statement, mathematical model and initial and boundary conditions for the problem are established. The design is also meshed here. Secondly, the Solver stage where it solves the problem for the established parameters for the specified number of iterations. Finally, the Post Processor stage where the solved problem and its parameters are visualized and analyzed.

### **II. Design**

An aerofoil is the shape of a wing or blade as seen in the cross section. It is a two-dimensional object, the shape of the cross section of the wing at right angles to the wing span, with the function of producing a controllable aerodynamic force by its motion through the atmosphere. An airfoil shaped body moving through a fluid medium produces an aerodynamic force. The design of the aerofoil was done using CATIA V5. The aerofoil is ensconced in a certain defined domain which makes analysis using AcuSolve simpler and more effective.

### **III. Meshing**

The meshing of the aerofoil was carried out using AcuSolve software. Zone meshing technique was used for this aerofoil. By zone meshing, accuracy of simulation results was improved. A very small grid size was assigned for the immediate outer layer of the aerofoil and a relatively larger grid size was assigned for the rest of the computational domain so as to avoid computational wastage. The generated mesh is shown in figure 1.

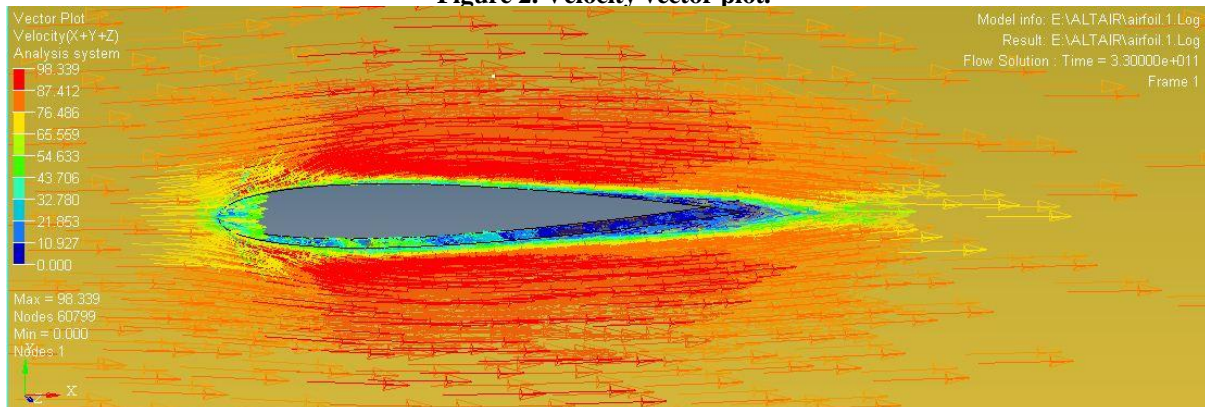


**Figure 1. Generated Aerofoil Mesh**

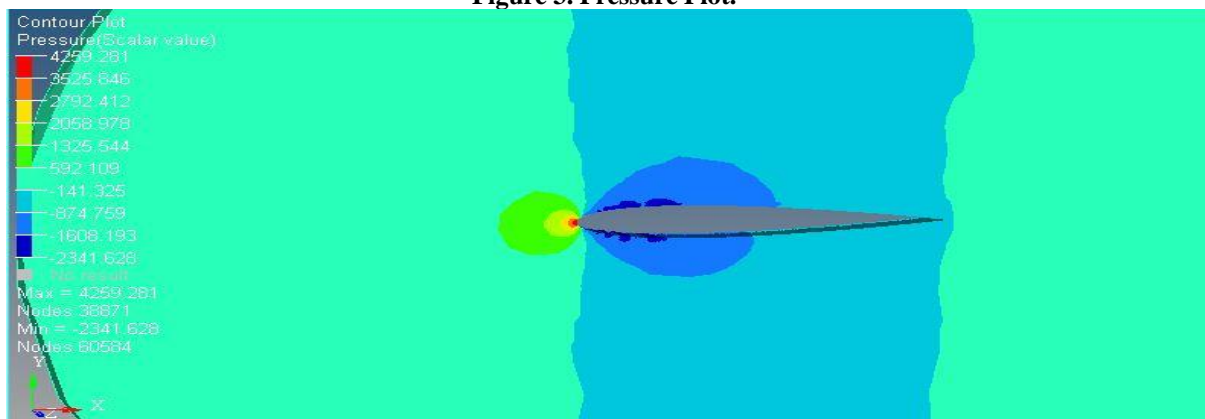
#### IV. Simulation

The zone meshed aerofoil was imported into HyperView software and simulated. The meshing was checked for errors. The solver used is a steady state solver. Turbulence equation was used for the simulation. The type of turbulence equation used was Spalart Allmaras. The flow equation used for the simulation was Navier-Stokes equation. The Temperature equation was turned off as the simulation was being conducted for velocity parameters. The inflow velocity type was Cartesian. The turbulence input was direct. Eddy velocities were kept at zero. The simulation was carried out for various iterations and various velocities and entry angles. The output values obtained were consistent with the theoretical values. The velocity vector plot, and the pressure contour plot of the simulation conducted at X velocity = 83 m/sec and Y velocity = 50 m/sec are shown in Figure 2, and Figure 3 respectively.

**Figure 2. Velocity vector plot.**



**Figure 3. Pressure Plot.**



As seen from the simulation results, velocity at the entry points is high and increases to reach maximum velocity at the sides. The pressure plot suggests that pressure is minimum at the sides and this validates the data shown by the velocity vector plot. The stagnation point is obtained from the plots. Legends displayed on the plots aid in analysis of the measurement values. The data obtained from the plots show that this aerofoil can be used at subsonic speeds.

.

## VI. References

- [1]. Douvi C. Eleni, Tsavalos I. Athanasios and Margaris P. Dionissios, Evaluation of the turbulence models for the simulation of the flow over a National Advisory Committee for Aeronautics (NACA) 0012 airfoil, Journal of Mechanical Engineering Research Vol. 4(3), pp. 100-111, March 2012.
- [2]. Robert J. McGhee and William D. Beasley, Low-Speed Aerodynamic Characteristics of a 17 -Percent Thick Airfoil Section Designed for General Aviation Applications, National Technical Information Service, 1973.
- [3]. Dr.ir.Bert Blocken, MOOC, Sports and Building Aerodynamics, Eindhoven University of Technology, April-June, 2014.
- [4]. J. D. Jacob, University of Kentucky, On the Fluid Dynamics of Adaptive Airfoils, 1998 ASME International Mechanical Engineering Congress and Exposition, November 15-20, 1998, Anaheim, CA, USA.