Numerical investigation of fluid flow around NACA0012 airfoil using OpenFOAM

Subham K. Das

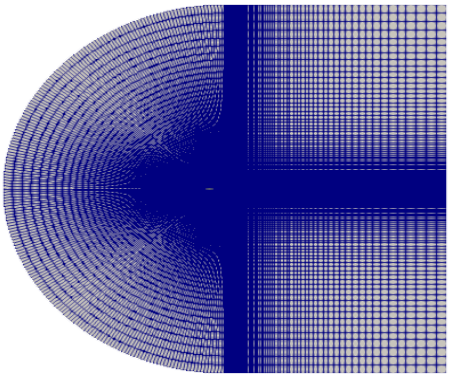
Indian Institute of Technology Jodhpur

Email: das.3@iitj.ac.in

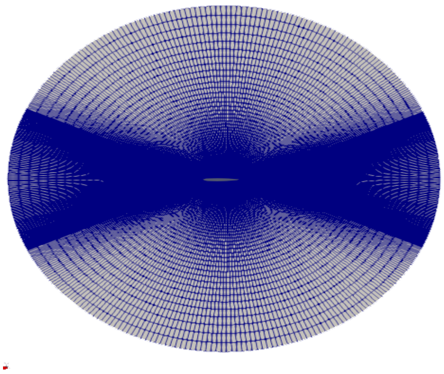
Abstract

Unmanned Aerial Vehicles (UAVs) are increasingly being used for numerous applications such as surveillance, reconnaissance, agriculture, scientific missions and aerial photography. UAVs, compared to conventional air-crafts offer advantages of cost-effectiveness, higher payload capacity, and portability. However, there is an upper limit to the maximum altitude UAVs can achieve as they operate on Reynolds number ranging from 10^3 to the order of 10^5, where a significant amount of lift is not generated. The accurate and efficient determination of the aerodynamic forces on wings due to relative fluid motion are essential for the design and development of UAVs. The wings of the UAVs are derived from airfoils and the aerodynamic forces are often characterized by the lift and drag coefficients of the airfoils. The objective of this study is to numerically examine the effect of meshing strategy, domain extent and the choice of the computational domain on the flow physics around a NACA0012 Airfoil.

Two of the most promising grids for studying the flow around an airfoil are the C and O type grids. For the purpose of this study, comparisons were made between both C and O grids on the basis of the accuracy of the results produced by them and their ability to capture the flow physics. It is necessary to have sufficient length of the domain behind the trailing edge of the airfoil in order to capture the wake. It also helps in proper convergence of the solution. But having a larger domain size increases the computation time significantly. Hence another objective of the study was to come up with a domain size that would be able to predict the lift and drag characteristics of the airfoil and at the same time provide better computational efficiency. Taking these factors into consideration, the two computational domains were modeled using three different values of domain extent. The length of the domain behind the trailing of the airfoil edge investigated are 5c, 10c, and 15c respectively where c is the chord length of the airfoil. The grids generated for both the domains are shown in Fig. 1.



(a) C-domain



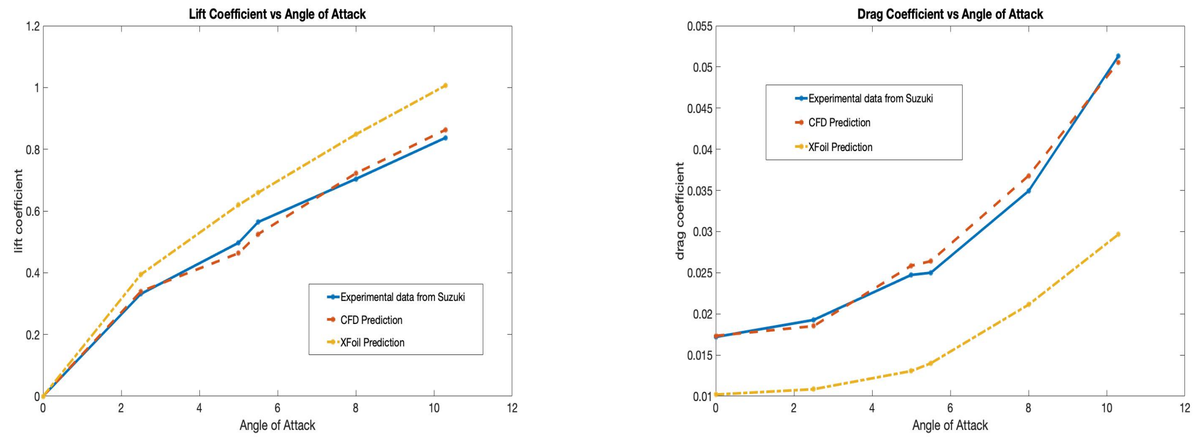
(b) O-domain

Figure 1: Grids generated to investigate the effect of boundary proximity and choice of computational domain on numerical results.

Lissaman[1] presented a comparison of the performance of airfoils at different bands of the Reynolds number. He concluded that airfoils fail to operate at a Reynolds number below 100 due to the dominant viscous effect and demonstrate a relatively weaker performance up to a Reynolds number of 10^5. At this range, the flow behavior is quite complex due to the transition from laminar to turbulent regime. Also, the fluid particle is unable to sustain the adverse pressure gradient which ultimately leads to boundary layer separation and reattachment. So, first computations are done using OpenFOAM for Reynolds Number of 6×10^6 for both types of grids at different values of domain extents and then compared with the standard NACA0012 validation data available at NASA's website[2]. It was observed that in case of a C-Domain, the distance of the far-field region had a significant effect on the computed solutions. However, this effect became less significant after a domain size of $10c$. There was very little effect of boundary proximity on the computed solutions in case of an O domain. Although there was a significant amount of error in lift coefficient (C_L), the C Mesh was able to predict the drag coefficient (C_D) with reasonable accuracy. The computational efficiency of C mesh was much lesser than the O mesh with the same domain extension. As expected, the computation time increased significantly with an increase in the domain length since the number of cells increased significantly.

In case of an O grid, the flow near the wall was resolved well when compared to a C grid which could not resolve it properly. This was further supported by the values of lift and drag coefficient at an angle of Attack (AoA) of 15 degrees. According to the experimental data available at NASA's website, the lift coefficient at 15 degrees AoA is 1.5. C grid resulted in a value of 0.95 which was too less. However, the lift coefficient obtained with the O grid was 1.46 which proved it a better choice. Hence the O domain with a domain size of five times the chord length of the airfoil was chosen for further study.

In order to further test the reliability of the O domain, numerical simulation was performed on a conventional NACA0012 airfoil at a Reynolds Number of 2×10^5 and at different AoA. The pressure and velocity contours were analyzed and the lift and drag coefficients were compared with the available experimental data from Suzuki[3] and they were in good agreement. These data were then compared with the results obtained from XFoil[4]. The comparison between the experimental data, CFD results, and XFoil[4] prediction are shown in Fig. 2. It was observed that XFoil over predicted the lift coefficient and underestimated the drag coefficient by a significant amount and the error in estimation increased with an increase in AoA. This observation showed that XFoil was unable to predict the flow characteristics at a Reynolds Number of 2×10^5.



(a) **(b)**

Figure 2: (a) Comparison of experimental and numerical lift coefficient with Xfoil Prediction, and (b) Comparison of experimental and numerical drag coefficient with Xfoil Prediction at a $Re = 2 \times 10^5$.

One of the drawbacks of 2D simulation using Computational Fluid Dynamics is its inability to predict the flow characteristics around an airfoil at higher AoA. There has been a lot of research going on in order to predict the flow characteristics near stall using sophisticated turbulence modeling that accounts for the transition between laminar to the turbulent regime. As observed during experiments, the stall of the airfoil under study was at $11°$ degrees. The CFD simulation gave accurate results until $10.5°$ degrees but failed to predict the onset of stall. To investigate this further, the experimental results near the stall were compared with the data available at the XFoil database for the same case. It was observed that the data available in the Xfoil database deviated significantly from the experiments. This clearly implied that both CFD and XFoil could not provide reliable results near the stall. Hence, further research is required for better prediction of the onset of transition and include the effect of the same thus covering a broad range of AoA.

References

[1] PBS Lissaman, Low Reynolds Number Airfoils.

[2] 2D NACA0012 Airfoil Validation Case, turbmodels.larc.nasa.gov.

[3] Yasumasa Suzuki, Aerodynamics and Acoustical Characteristics of Two dimensional Airfoil Flows.

[4] Mark Drela, XFoil User Primer, MIT Aero Astro Harold Youngren, Aerocraft, Inc.