Computational Fluid Dynamics Lab

Samuel Razumovskiy *
University of Colorado at Boulder, Aerospace Engineering Sciences
ASEN 3111, Aerodynamics

I. Introduction

This computational fluid dynamics (CFD) lab was conducted to study accuracy of a CFD simulation. Using the CFD data we try to find the zero lift angle of attack ($\alpha_{L=0}$), the lift slope, the stall angle (α_{stall}), and the Maximum sectional coefficient of lift(C_l). Once these are found they can be compared to thin airfoil theory, vortex panel method, and experimental data. An additional comparison was also performed with the coefficient of pressure (C_p) distribution along the airfoil. The results show that the CFD simulation finds fairly similar performance results as the experimental data and the thin airfoil theory, but a slight difference from the vortex panel method.

A. CFD Simulation Setup

The simulation was set up using ANSYS Fluent, a NACA 0012 airfoil, and a Reynolds number of 6,000,000 using the turbulent RANS model. In order to be able to find the desired values a graph of C_l vs. α needs to be generated. In order to generate the graph the value of C_l at a given α is calculated and then stored, the range of α values used was $0° - 15°$.

B. Analysis

Once the C_l vs. α graph is completed we can find $\alpha_{L=0}$ by finding the x-intercept, the lift slope by finding the slope of the straight portion of the graph, the stall angle by identifying at what α the max C_l occurs, finally the maximum C_l is simply what max value C_l reaches. Fluent was used directly to plot and compare the C_p distribution along the airfoil at $\alpha = 15°$. This was compared to the NASA study performed simply by comparing plots.

*ID:109343605

II. Results

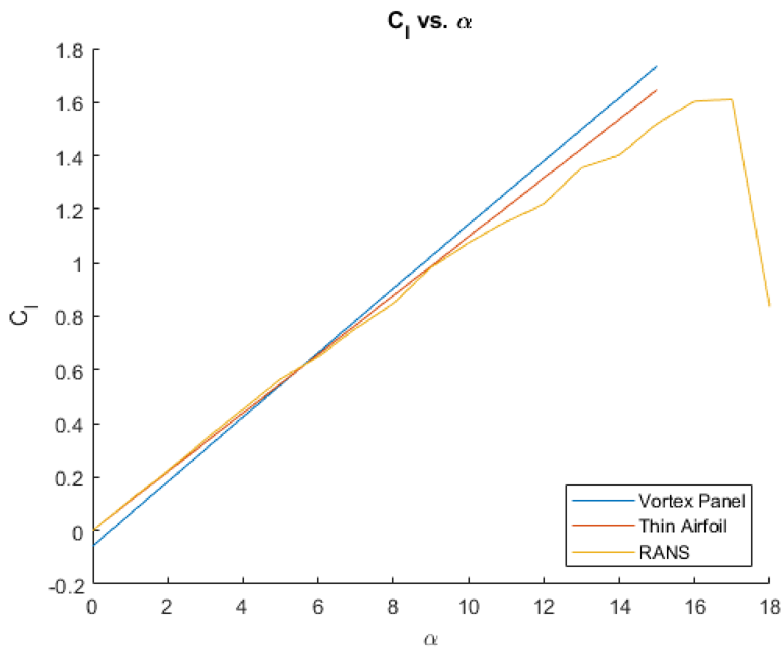

Figure 1: C_l vs. α CFD RANS

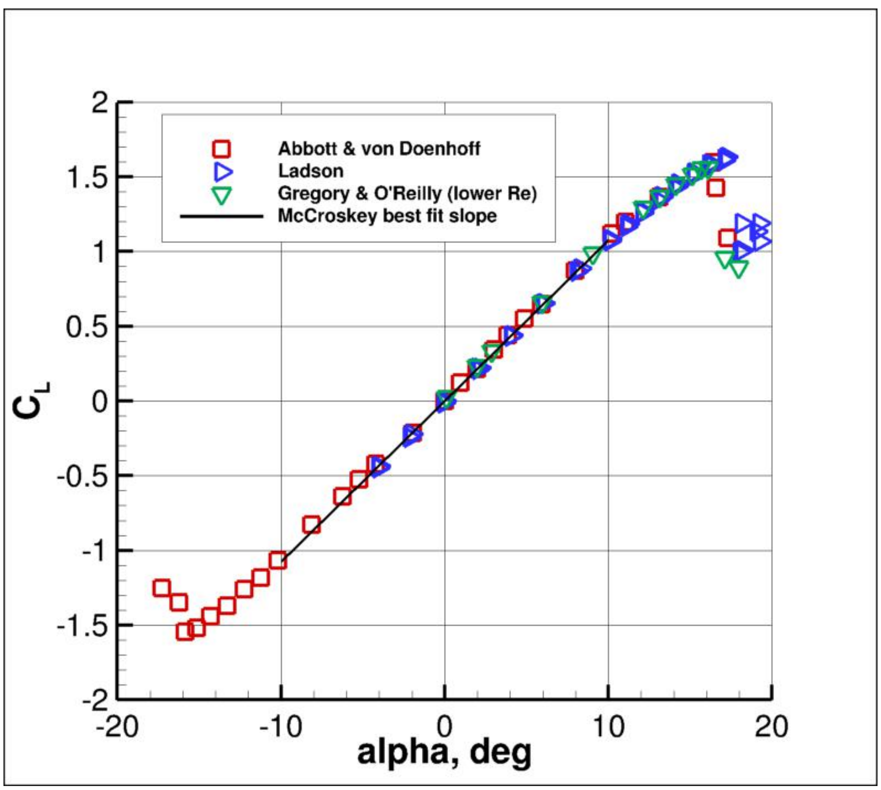

Figure 2: C_l vs. α experimental NASA [3]

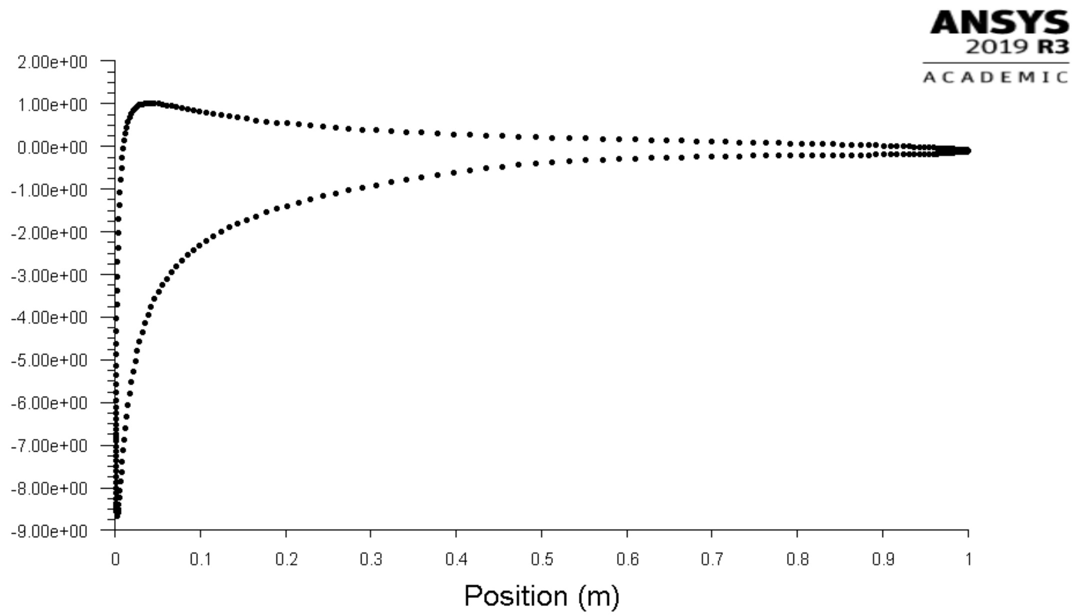

Figure 3: C_p vs. x $\alpha = 15°$CFD

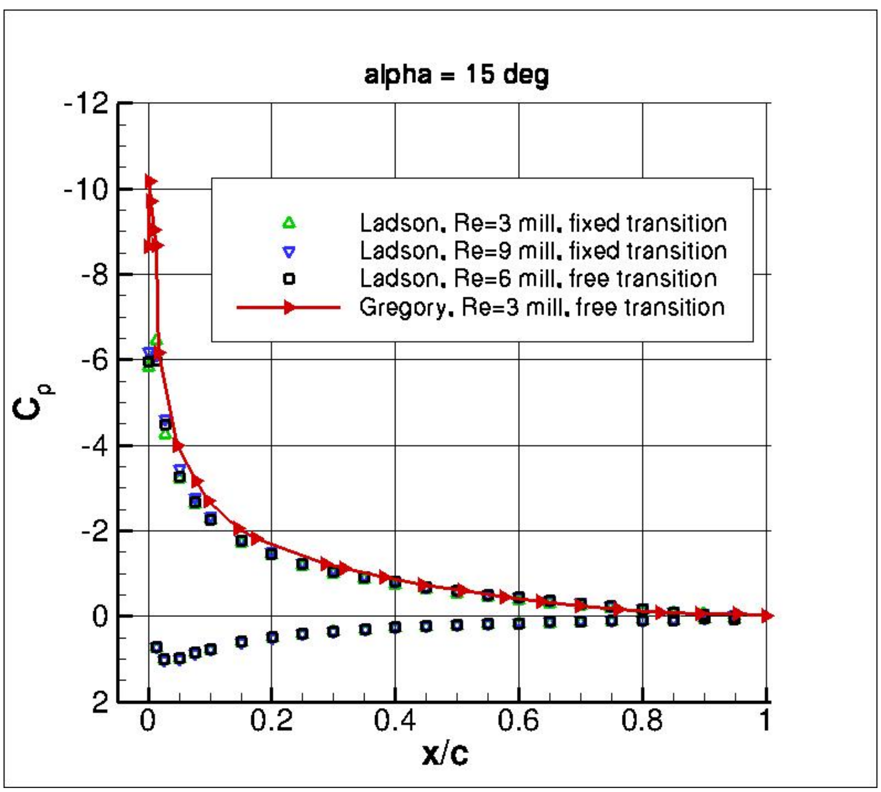

Figure 4: C_p vs. x/c $\alpha = 15°$ experimental NASA [3]

III. Conclusion

According to the CFD analysis $\alpha_{L=0} = 0$, the lift curve slope $a = 5.91\frac{1}{rad}$, the stall angle $\alpha_{stall} = 17°$, and finally the max $C_l = 1.61$. Looking at figure 1. the simulation data follows the thin airfoil theory approximation better than the vortex panel method. Which makes sense since the vortex panel method ignores drag and has an increases in lift compared to thin airfoil theory. The CFD simulation also closely follows the experimental data in Figure 2. Where the maximum C_l is around 1.6-1.7 and $\alpha_{L=0}$ is around 17-18 degrees. Finally, the coefficient of pressure plotted against position along the airfoil shows that the CFD analysis has a similar profile as the experimental NASA study where the y-axis is inverted. The maximum and minimum C_p are close to the experimental data as well with a difference of around 2 for the max, and no observable difference in the minimum. Overall the CFD simulation seems to follow the experimental results fairly closely.

References

[1] Farnsworth (ASEN 3111 CFD Lab) University of Colorado Boulder, 2019. Web.

[2] John D. Anderson, Jr. Fundamentals of Aerodynamics, 6th Edition, McGraw-Hill Aerospace Engineering Series 2011.

[3] Rumsey, C., "2D NACA 0012 Airfoil Validation," NASA Available: http://turbmodels.larc.nasa.gov/naca0012_val.html.