Airfoil Design at Low-Reynolds Numbers

Christopher Ong
Tagliatela College of Engineering – Mechanical Engineering
Dr. Maria-Isabel Carnasciali – Assistant Professor, Mechanical Engineering

Abstract

Airfoils were typically expected to be used for large craft like what is seen in commercial or private airplanes for transportation of peoples and goods across the globe. In the last ten years, there has been an increased look into how airfoils can be designed for lower ranges of speed. Currently small, low-speed aircraft are being used to study large areas due to relative cost compared to other methods. The purpose of this research was to see if incorporating aspects of bioinspired design with airfoils would improve the overall flight performance. Computational Fluid Dynamics (CFD) simulations were used to verify existing airfoil design with experimental data currently available. Various geometries (bumps, dimples, triangles, etc.) were added to the airfoil designs at the leading or trailing edge and simulated to get an understanding of how fluids interacted with those shapes.

Introduction

An airfoil is a streamlined geometry that interacts with a moving fluid which produces aerodynamic forces around the body. The fluid of interest in this case is air. There are two particular forces of interest for determining the effectiveness of an airfoil: lift and drag forces. The lift force acts perpendicular to the body of the airfoil body creating an upward motion. The drag forces act in parallel against the airfoil body hindering its performance.

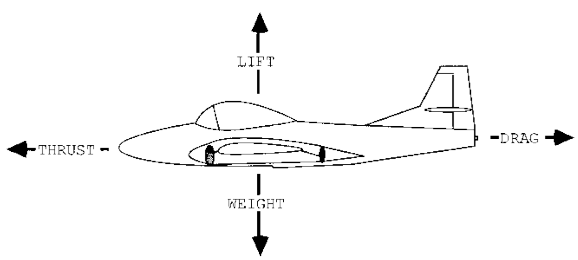


Figure 1 – Aerodynamic Forces

For drag there are two contributing forces knowns as pressure and skin friction drag forces. The pressure drag forces are the forces that result from the pressure force distribution around the airfoil body, while the skin friction is the viscous effects caused by the fluid moving against the rest of the airfoil body [1]. These forces are non-dimensionalized by a dynamic pressure force determined by the size of the airfoil and the speed of the air flowing by. The resulting lift coefficient (C_L) and drag coefficient (C_d) are given by equations 1 & 2.

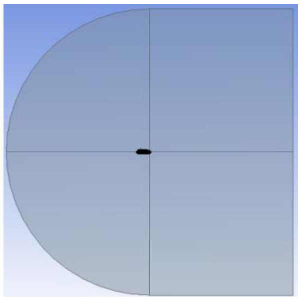
$$C_L = \frac{2F_L}{\rho A v^2} \, (1) \qquad C_d = \frac{2F_d}{\rho A v^2} \, (2)$$

The Reynolds number is an important dimensionless number that helps describe the fluid flow that is being studied. Reynolds number is a ratio between the inertial and viscous forces of a fluid, which is density of fluid, velocity of the fluid, reference length over the dynamic viscosity of the fluid [1]. The airfoil study is at Reynolds number of 6E04, 1E05 and 2E05, which using several variables can be used to determine the velocities that were to be used. With the use of airflow close to sea level and an airfoil length of one meter, the velocity range is within 1 to 3 m/s, or 2.2 to 7 mph.

Simulation Verification

To verify the accuracy of the simulation setup there was a comparison of published test data and the simulation results of the airfoil. For test data, the University of Illinois has published hundreds of wind tunnel test results on their website that can be used by anyone who is interested in airfoil selection or design[4][5]. Due to hundreds of different airfoils for various purposes, only a handful of airfoils were selected. The selection criteria for the airfoils were geometries that were designed for smaller aircraft such as R/C planes. The airfoils that were selected had their results compared, and it was determined that E193 airfoil had the highest lift coefficient. This higher lift coefficient can be attributed to the higher curved shape that the airfoil has.

The simulation software that was used was FLUENT, which is part of the ANSYS® package. ANSYS® has several components that allow for geometry and meshing generation. A 2D E193 airfoil was imported into ANSYS® Design Modeler. The research involved the study of fluid interacting with the airfoil. The E193 is a solid body, so a control volume had to be created surrounding the airfoil in order to simulate the airflow around the body. Shown in **Figure 2**, the control volume can be seen with a rectangular shape that is 25 meters tall and 12.5 meters back. The rest of the control volume has a circular area that has a radius of 12.5 meters. A surface was then created for the control volume with an airfoil shape cut out in the center from that surface.



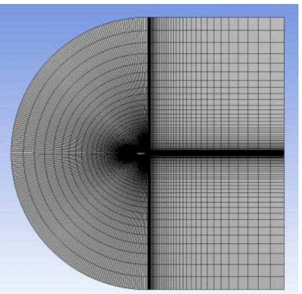


Figure 2 – 2D Control Volume   Figure 3 – Meshed 2D Volume

The control volume was then meshed as seen in **Figure 3**, using mesh controls that allowed for a mesh biased focusing more on the surroundings of the airfoil. The cell density can be seen in **Figure 3**, as the darker part of the mesh. The logic behind this is that the fluid interacting with the airfoil is the important focus to capture the most accurate lift and drag forces acting on the body. It is important to capture the pressure field in the surroundings of the airfoil for best results when looking at either the leading or trailing edge of the airfoil.

The simulation setup used a density based solver, with boundary conditions being the component velocities at the inlet, pressure outlets for the surroundings and using the Spalart-Allmaras viscous turbulence model. In FLUENT, there are three turbulence models, Spalart-Allmaras, K-Epsilon and an inviscid model. All the models produced similar results for the lift and drag forces, but the Spalart-Allmaras was chosen as the model due to the shorter simulation times. This allowed for a wider range of simulations to be ran within a shorter amount of time. FLUENT has several monitors that allow the determination of lift coefficient, drag coefficient and residual monitors. For a simulation to be determined as useful the residual monitors has to converge under a certain criteria and the criteria that was selected was 1E-04. The residual monitors report the continuity equation, the select model values and the velocity components.

The simulations were ran with the velocity boundary conditions of 0.88, 1.46 and 2.92 m/s, which correspond with the Reynolds numbers of 6E04, 1E05 and 2E05, respectfully. Each velocity were broken up into component velocities within the angle range 0 – 12 degrees in increments of 2. This changing angle is referred to as the angle of attack, which is the angle of the airfoil acting against the moving fluid. This was done to compare the results of the simulation and the data that was published by the University of Illinois. In **Figure 4**, the simulation values can be seen at the corresponding Reynolds numbers and angles of attack. In **Figure 5**, the wind tunnel data produced by the University of Illinois can be compared to the simulation data.

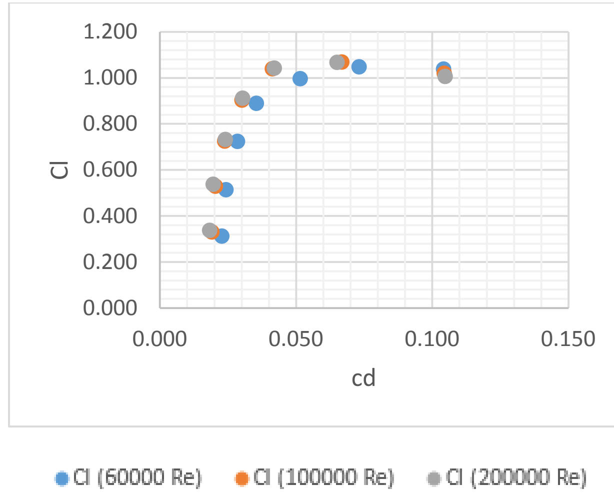


Figure 4 – Simulation Data

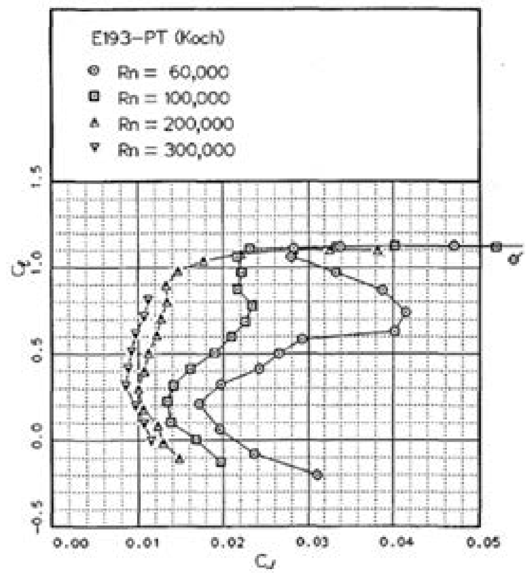


Figure 5 – Wind Tunnel Data from University of Illinois

The data published by the University of Illinois did not come with any uncertainty values, so it cannot be determined what the exact values are. The published data at the higher Reynolds number can be compared to the simulation data with similar results. For Reynolds number of 6E04 and 1E05 the lower lift to drag coefficient values do not correspond with the simulation results. This can be due to the inaccuracy of the tools used in the wind tunnel trials or noise caused by other factors that could hinder the accuracy of the test. There seems to be enough correlation between the wind tunnel data and the simulation data to give an idea of the accuracy of the simulation setup. At higher values there is an uncertainty of closer to +/- %5.

Results and Discussion

Various geometries were simulated to get a basic understanding on how the fluid interacts with the basic shapes. There were dimpling and recessed surfaces simulated that dug into a flat plate alongside raised surfaces that showed the contrast. It was determined that surfaces digging into a body would create eddies, that would apply a larger pressure load lowering the aerodynamic properties of the body [2]. The raised surfaces were chosen to be incorporated into the airfoil to see how these shapes affected the aerodynamic properties of the airfoil.

In **Figure 6**, it can be seen that the airfoil at a 0-angle of attack has a higher pressure field compared to that of the rest of the body. This can be considered the case for other angles in which the airfoil's leading edge is facing the moving fluid. Due to the higher pressure field at all simulated angles of attack, this area was chosen to incorporate other geometries to understand how the fluid flow would be affected, ultimately affecting the airfoil's aerodynamic properties.

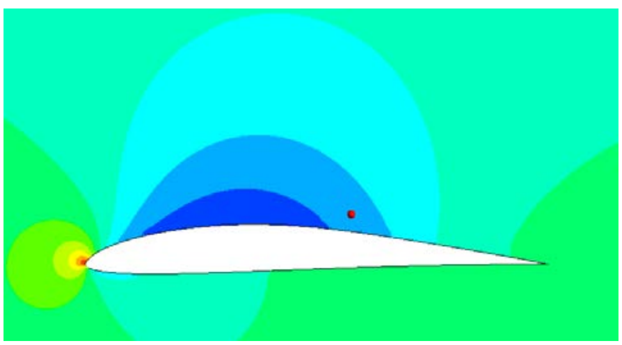


Figure 6 – Pressure Gradient of Airfoil at 0 Angle of Attack
(Red – Highest Pressure / Blue - Lowest Pressure)

The two basic geometries that were incorporated were right triangles and half circles at the leading edge. The half circles were approximately 0.1 meters in diameter or about one-tenth of the total airfoil length. The triangles were about 0.1 meters in length with varying heights of 0.02, 0.025 and 0.03 meters in height over the surface. Two modified airfoils can be seen in **Figure 7**; these airfoils are at 0-angle of attack.

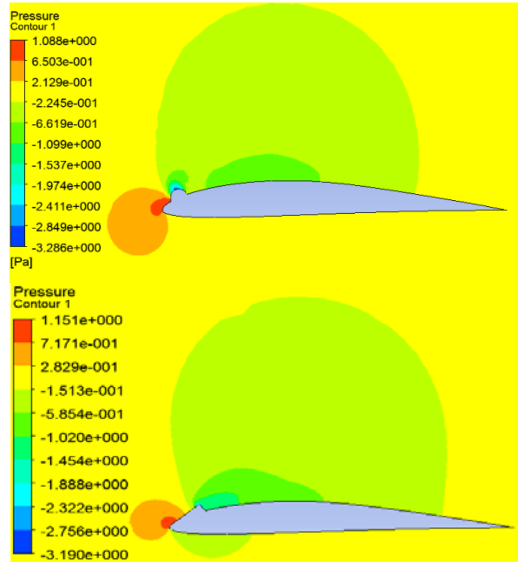
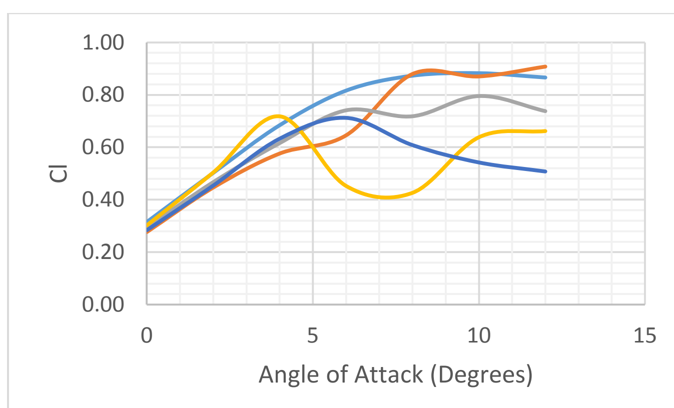


Figure 7 – Modified Airfoils at 0 Angle of Attack

The five simulation trial results are compared in **Figure 8**, which shows how the lift coefficient compared to the drag coefficient. These simulations were ran at a 1E05 Reynolds number with a corresponding velocity of 1.46 m/s. The residual criteria for the monitors were 1E-03; far from an ideal case of 1E-06. This was done due to the lower computational times for the results.

When comparing the data in **Figure 8** to **Figure 4**, it shows that the trials have overall smaller lift coefficients and higher drag coefficients. The drag coefficient drastic change can be attributed to the increased surface at the leading edge causing a higher pressure buildup. This leads to a higher coefficient drag and can lead to a lower coefficient of lift. It appears a streamedlined body performs better than a geometry that has was modified with bumps, dimples or raised surfaces.



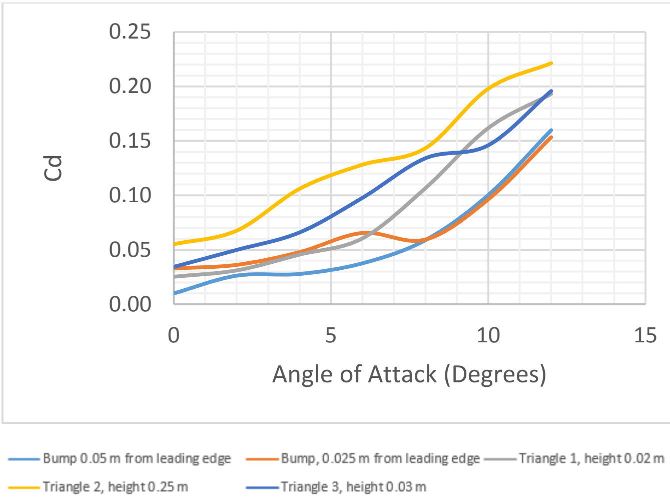


Figure 8 – Modified Airfoil Results

Conclusion and Further Consideration

The focus of the study was the fluid flow over a 2D airfoil and considering changing aspects of the geometry to influence the aerodynamic forces acting on. By adding elements to the geometry, the lift and drag forces were influenced by the changes. The values from the modified geometries are lower than that of the original airfoil. This can be attributed to the changing of the thickness of the body at points. With the airfoil being 2D, changing the chord length or the thickness of the airfoil will add the surface of which the aerodynamics forces are acting on. This will change the amount of force acting on the airfoil.

The airfoil is a simplified 2D case neglecting any 3D considerations of the fluid flow. This would be the next step for further studies. By bringing the airfoil to a 3D geometry, there has to be consideration on the total wing geometry, angle of the wing and tips at the ends that can affect the fluid flow in other ways. From here locating strategic points on the airfoil to influence the fluid flow would be the next step. By adding geomtries to these strategic locations will assist with keeping a stable flow around the airfoil.

References

[1] Anderson, J. D. (2001). *Fundamentals of aerodynamics*. Boston: McGraw-Hill.

[2] Vento, J. (n.d.). Analysis of Surface Augmentation of Airfoil Sections via Flow Visualization Techniques. Retrieved August 12, 2015.

[3] Mullen, B. (2014, February 9). FLUENT - Flow Over an Airfoil. Retrieved June 4, 2015.

[4]http://m-selig.ae.illinois.edu/ads/coord_database.html

[5] http://m-selig.ae.illinois.edu/uiuc_lsat.html

[6]http://www.globalsecurity.org/military/systems/aircraft/intro-performance.htm

Acknowledgement

I like to thank my faculty mentor, Dr. Carnasciali, for assisting me on this project. She guided me and worked with me through the numerous questions I had during the course of the project. I look forward to continuing the research.

I would also like to thank Summer Undergraduate Research Fellowship (SURF) of the University of New Haven for providing me this great opportunity In particular, I like to thank Carol Withers, Janice Sanderson and Mr. and Mrs. Carrubba for their continued support of the SURF program.

Biography

Christopher Ong, from Patterson, New York, is currently a senior studying Mechanical Engineering at the University of New Haven. His current interests are in thermodynamic and fluid mechanic studies. He plans to further his education by pursuing a masters in aerospace engineering with a focus in computational physics. A career working on rocket propulsion systems would be a dream come true!