Airfoil Design for Low-Reynolds Number

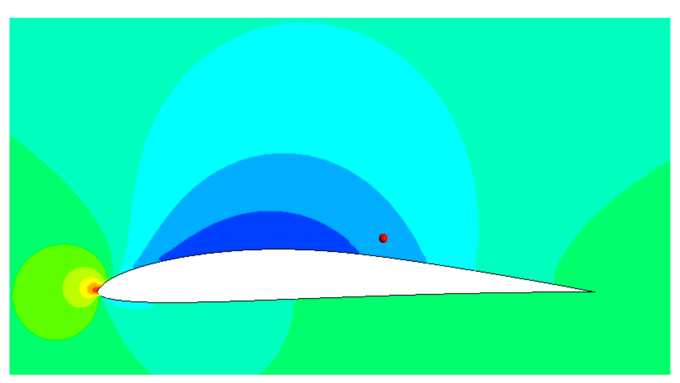
Christopher Ong, Mechanical Engineering, Senior

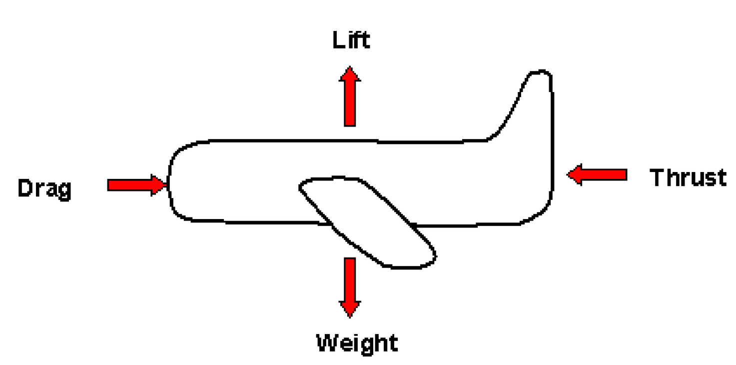
Maria-Isabel Carnasciali, Ph.D., Assistant Professor, Mechanical Engineering

Introduction

The purpose of the research was to get a grasp of the design process involved with selecting and modifying an airfoil. Then begin to incorporate features seen in nature such as raised and dimpled edges on insects, animals and marine life to improve the aerodynamic properties of the airfoil.

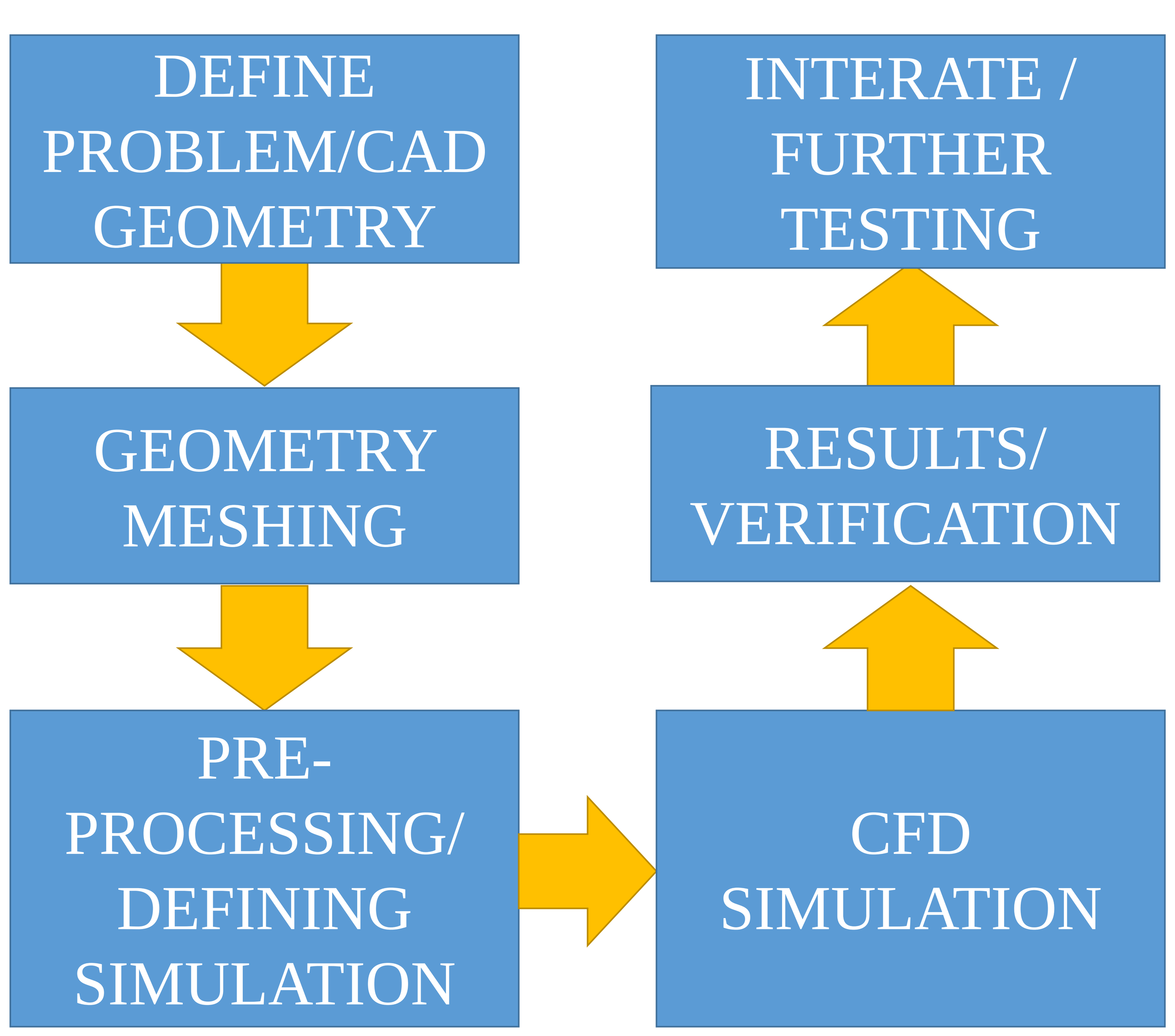
An airfoil is a streamlined shape that is designed to flow through a fluid. The moving fluid will cause several forces to act onto the body. One force is the lift force, which acts underneath the shape causing a pressure difference. This causes the airfoil to move upward. Then there is the drag forces, which are pressure and skin friction drag that act against the movement of the airfoil.





Methodology

To have aerodynamic testing of a shape can be costly and timely due to time constraints and needed fabrication. That is why the use of Computation Fluid Dynamic (CFD) tools become cost effective and useful to test simple geometries such as an airfoil with any major time constraints. CFD follows these steps:



Case Verification

The airfoil that was selected for the case study was the E193 "sailfoil". This airfoil was designed and tested at velocity range of 1 – 10 m/s by the Aerospace Department at Illinois University. What differentiates a sailfoil from an airfoil is the intended purpose. The E193 was designed for small fixed-winged gliders or Micro Air-Vehicles (MAVs). The CFD tool ANSYS FLUENT ran several simulations at varying Reynolds numbers (60000, 100000 and 200000). For this purpose each Reynolds number indicates a low fluid velocity. Figure 1 and 3 are the CFD simulation results for and figure 2 and 4 are the wind tunnel data from Illinois University.

Figure 1

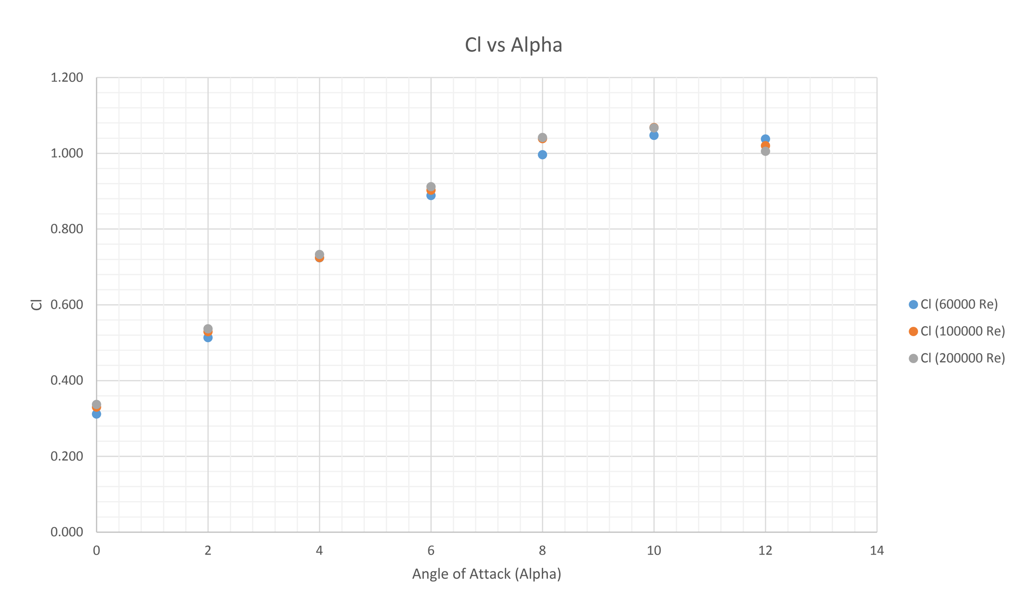


Figure 2

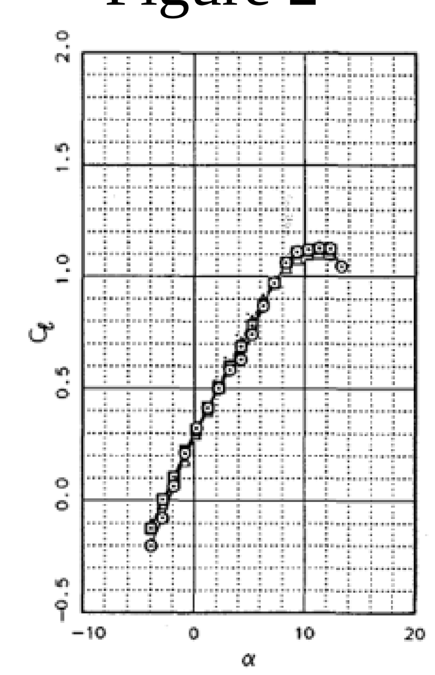
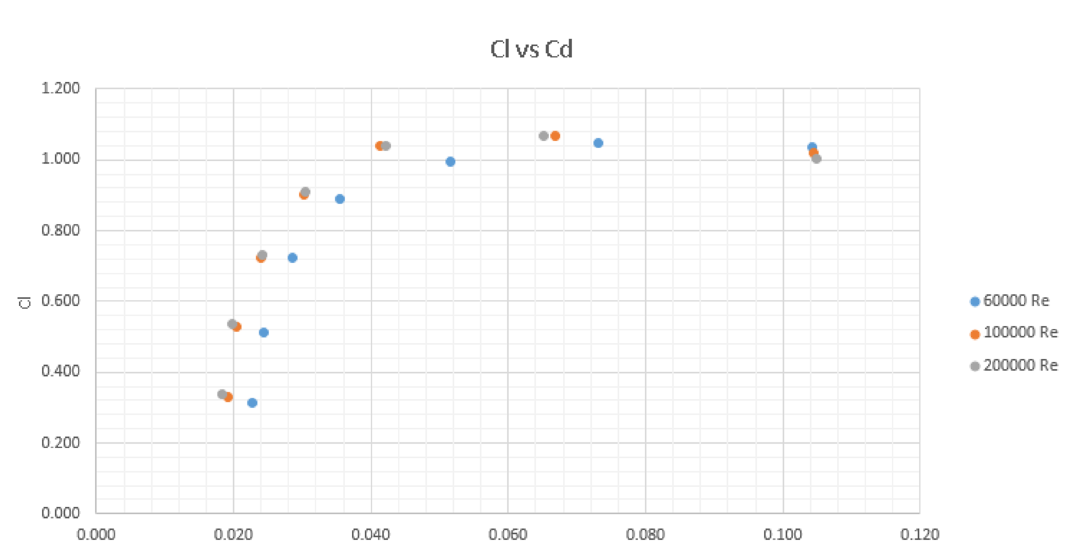


Figure 3



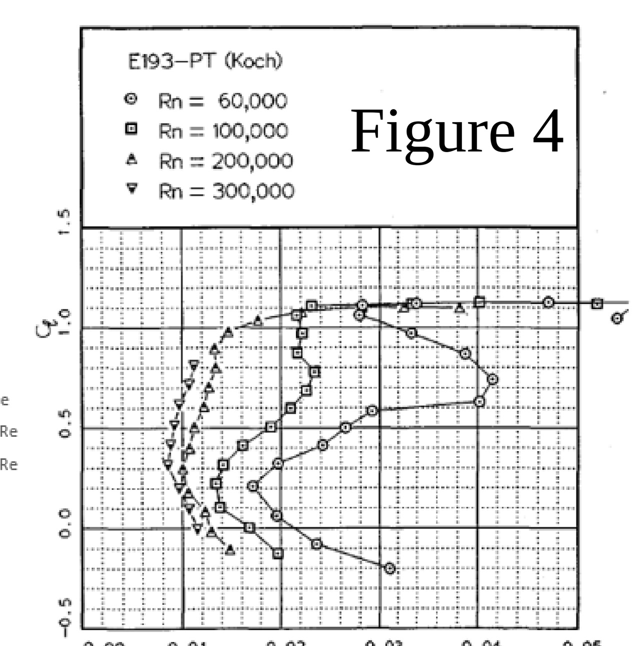
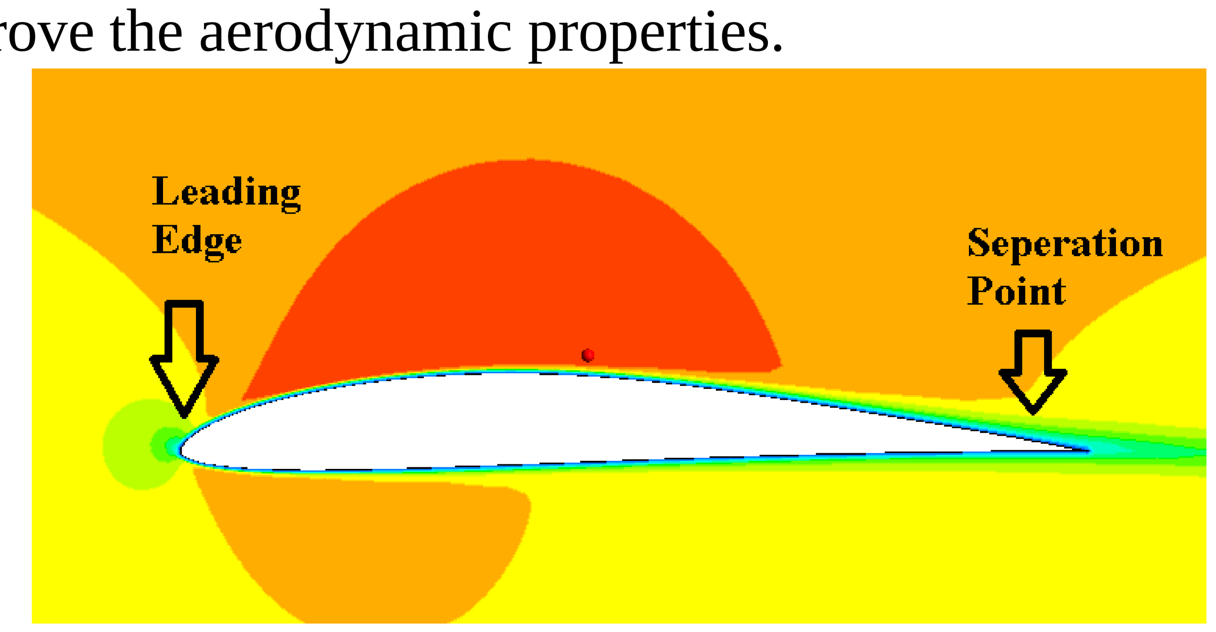


Figure 4

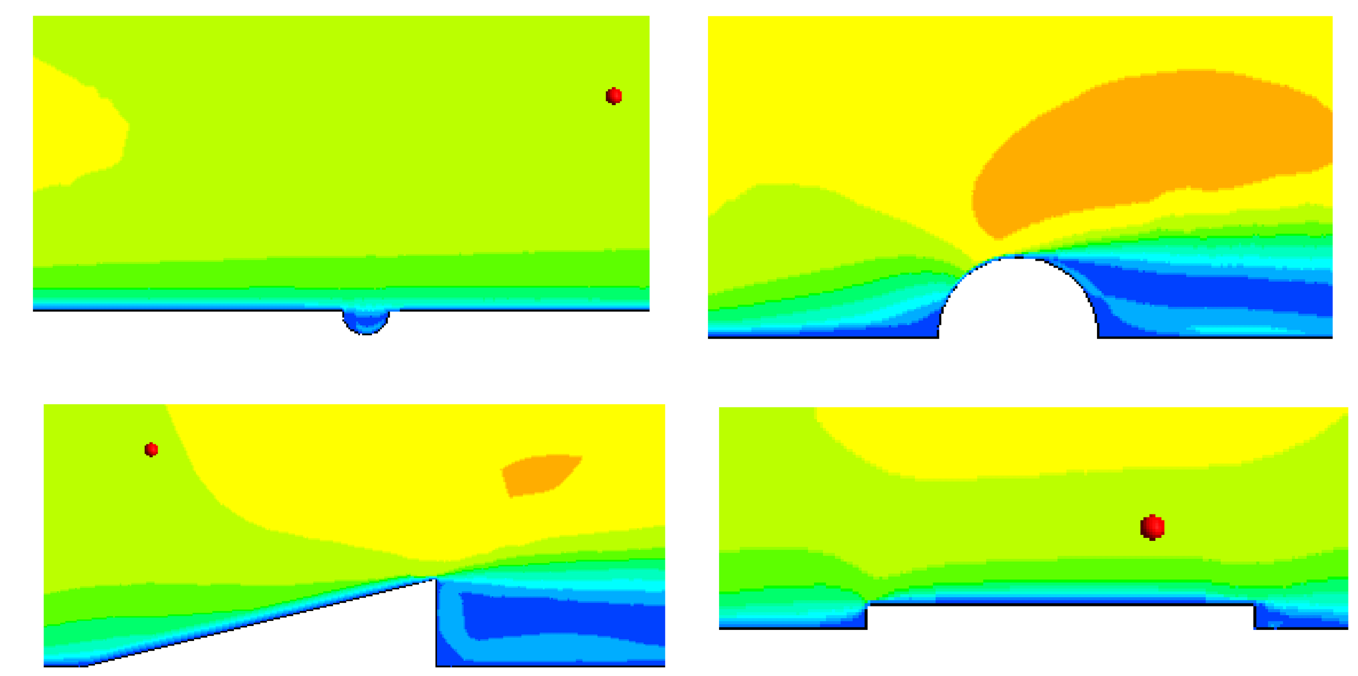
Areas of Interest

Two areas of interest to add design features to the airfoil are at the leading edge and the separation point. These two areas will have features that will affect the fluid behavior around the airfoil to improve the aerodynamic properties.



Further Testing

As of right now the behavior of air has been simulated over basic geometric shapes to see how the velocity and pressure profiles would interact with the shapes. Basic shapes that have been tested so far are dimples, bumps, wedges and raised surfaces at different sizes and lengths.



These are only 2D cases , which will be implemented on the airfoil to see how the forces acting on it will behave. Once the 2D case study is finished, it will be expanded to 3D to see if there are specific areas of interest that can be improved with strategic placing of fins, dimples or wedges.

Citations

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

[2] Vento, John Luis. "Analysis of Surface Augmentation of Airfoil Sections via Flow Visualization Techniques." *Http://digitalcommons.calpoly.edu/cgi/viewcontent.cgi?article=1047&context=aerosp*. Calpoly.edu. Web. 12 Aug. 2015.

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

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

[5]http://web.mit.edu/2.972/www/reports/airfoil/airfoil.html

Acknowledgement

I would like to thank the SURF program for giving me the opportunity to take part in this program over the summer. I also like to thank Dr. Carnasciali for helping me throughout this project.