Effects of Varying Porous Structures of an Airfoil

Sonal Sukhatme - sonal.sukhatme@gmail.com
La Canada High School, Class of 2024
USC Viterbi Department of Aerospace and Mechanical Engineering, SHINE 2022

SHINE
Summer High School Intensive in Next-Generation Engineering

Introduction

The Luhar lab focuses on the interactions of fluids and structures. This summer, I focused on how porous structures affect the lift and the drag of an airfoil. Reducing drag and increasing lift produces more efficient wings, which in turn decreases the amount of fuel needed to fly.

Objectives & Parameter Design

To reduce drag and increase lift in order to produce more efficient wings, we varied two parameters: the inter-hole spacing and the angle of the holes. For spacing, we modified the gap *s*, between holes as a function of the length of the airfoil:

s = 10%, 15%, and 20%

For the angles, we drilled holes that would appear horizontal when the airfoil was at the following angles of attack:

AoA = 3°, 6°, 9°, and 12°

We tested on a NACA 0012 airfoil that was *10 cm* long. To apply our findings to realistic airfoils, and to be consistent with previous work, we experimented at Reynold's Number, $Re = 5\times10^4$. The Reynold's Number is a nondimensional quantity that characterizes a system in a way that allows its properties to be compared at various scales. The (nondimensional) lift and drag of an airfoil are given by:

$$C_d = \frac{D}{0.5\rho U^2 c}$$

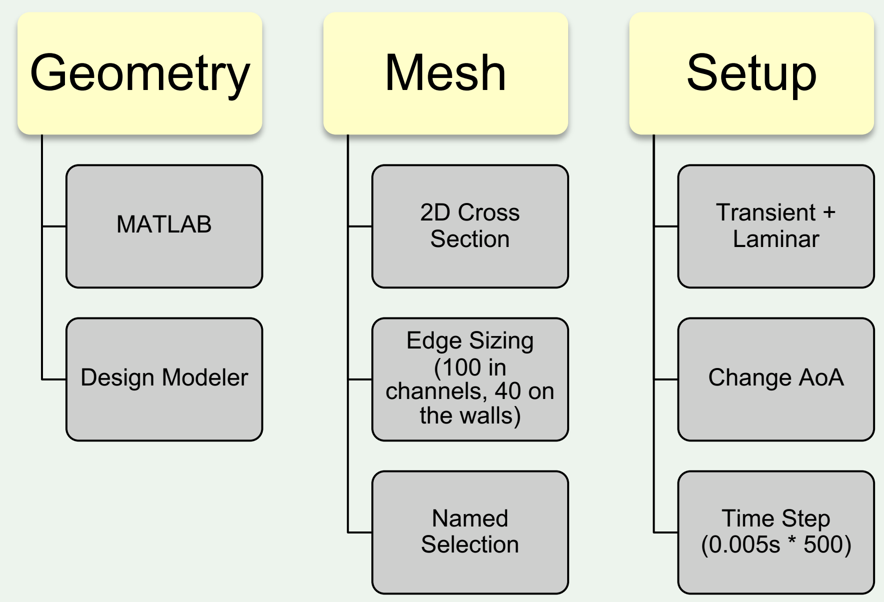
$$C_l = \frac{L}{0.5\rho U^2 c}$$

Our airfoil is symmetric and must be positioned at a nonzero *AoA* to generate lift. By adding holes through the airfoil, air is able to move from below to above the airfoil, reducing the so-called 'separation bubbles' by preserving laminar flow – a flow where the layers of the fluid move smoothly past each other without mixing.

Methods and Results

Simulations

Method:



We ran our simulations with Ansys, which runs in the order above. First, we imported the geometry of the airfoil that we generated in MATLAB. We used a 2D model instead of 3D so we could get higher resolution results with less processing power and processing time. The software tests certain data points across the field. In order to get the fine resolution that we want near the airfoil and in its channels, we have to specify the grid lines, called the mesh. We also named different parts of the model "inlet", "outlet", "wall", or "channel" so that we could tell the software where the fluid was entering and exiting from, and what parts of the model were solid walls. Then, in setup, we set the simulation to a time dependent model (*time step = 0.005 secs; 500 timesteps*). Additionally, we varied the angle of attack, by specifying the velocity of the fluid using *x* and *y* components, which depend on the sine and cosine of the angle of attack.

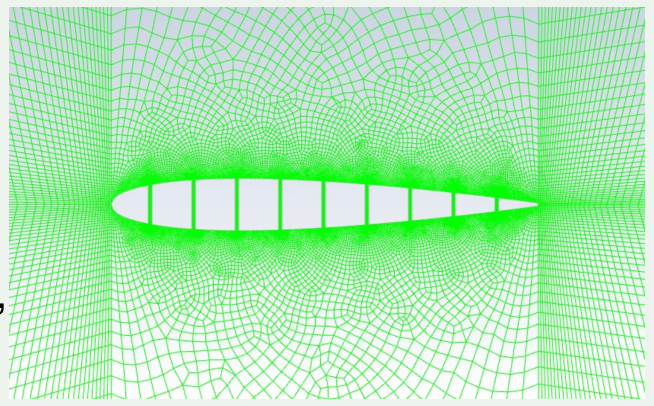


Figure 1. Mesh for s = 10%

Results:

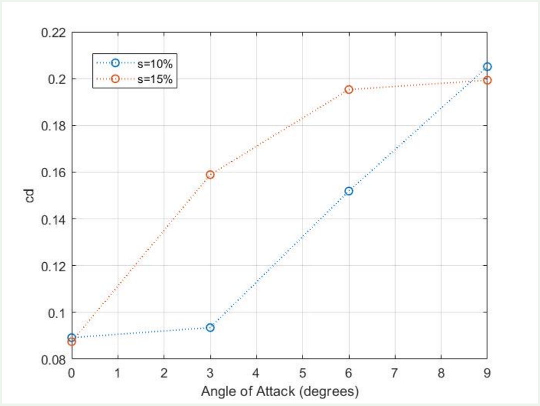


Figure 2. Graphs of Cd vs. AoA at spacing = 15%, 20%

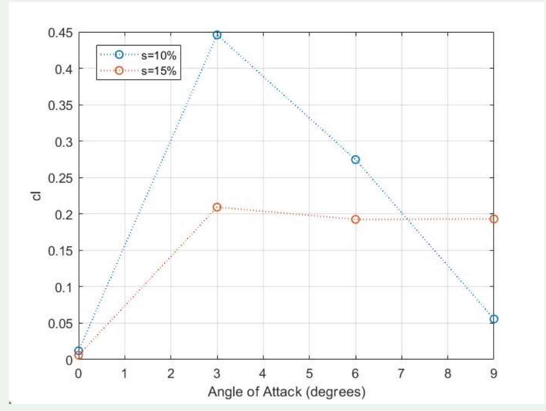


Figure 3. Graphs of Cl vs. AoA at spacing = 15%, 20%

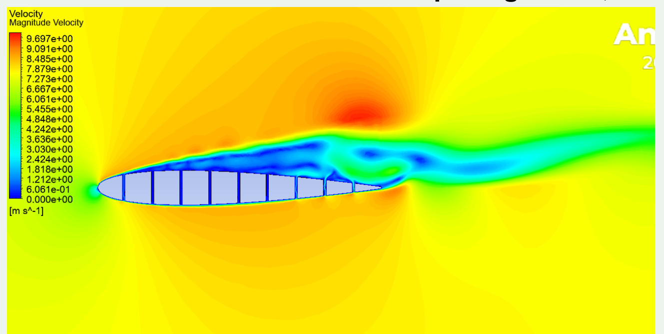


Figure 4. Velocity Magnitude Contour for s = 10%, AoA = 6°

Experiments

Method:

For our experiment, we 3D printed airfoils with different parameters. First, we used SolidWorks, a CAD program, to design two models - one with a set spacing (s = *15%)* and one with a set angle of *0°*. This way, we could make copies of each and vary each parameter separately. Due to the limitations of the 3D printer, we kept the size of the holes to *1 mm* in diameter, to avoid clogging the holes with the resin. The printer also had a size limitation, so we printed the airfoils in two parts, and added a *9 mm* hole through the airfoil so that the parts could be assembled with a rod. After printing, we sanded the airfoils down and glued the parts together.

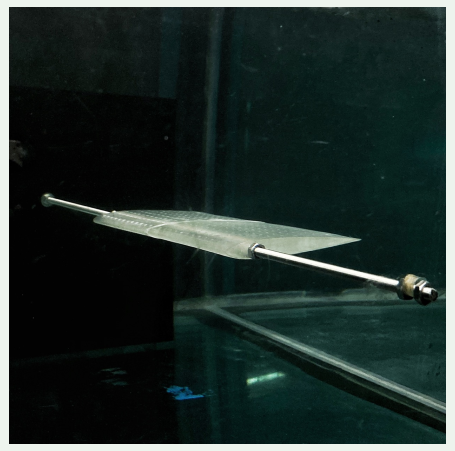


Figure 5. Airfoil fixture within water channel

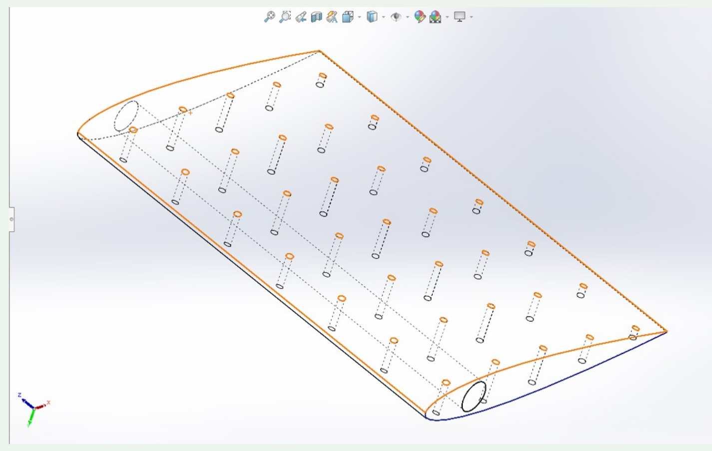
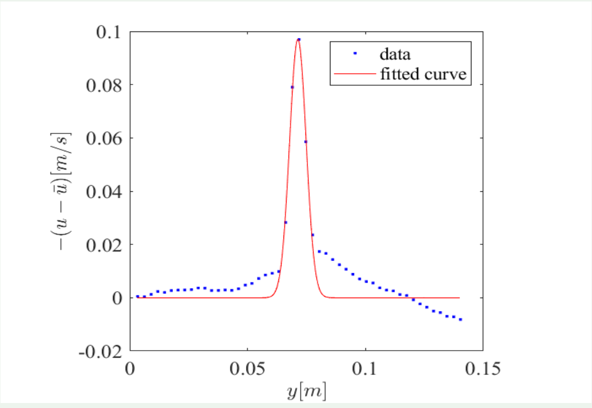


Figure 6. Porous Airfoil 3D CAD Model in SolidWorks

Our experiment works by illuminating a cross section of the airfoil using a laser. To track the flow of the water, small particles are dispersed in the water and their motion is calculated through images taken by a high-speed camera (500 Hz; 0.002 s timestep). Analyzing the velocity of these particles allows us to calculate the lift and drag generated by the airfoil. After calibrating the water channel to correctly line up the laser and focus the camera, we set the water channel to a speed that creates the same Re as our simulations (which used air).

Results:



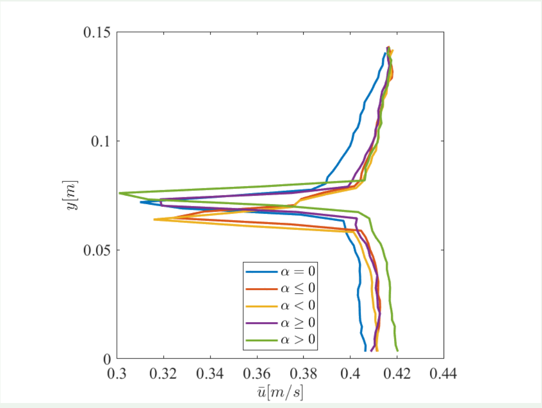


Figure 7. Horizontal velocity profile at trailing edge in y direction

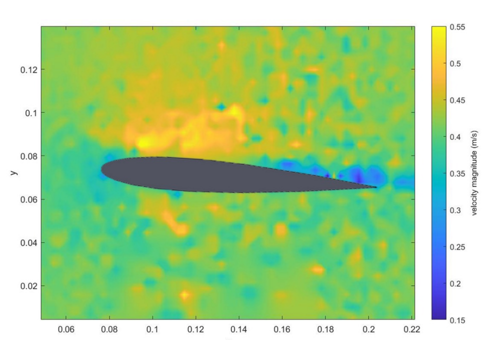


Figure 8. Velocity magnitude contour at s = 10%, positive AoA

Skills Learned

Ansys

SOLIDWORKS

MATLAB

Conclusion

Our simulations showed the following:

1) For small inter-hole separation, the lift coefficient first increases with angle of attack and then decreases.
2) For larger inter-hole separation, the lift coefficient levels off at an angle of attack of 3°.
3) For even larger inter-hole separation, the data are too unreliable to draw a firm conclusion.

Experiments in the water channel were limited to a single inter-hole spacing. We experimented with *AoA = 0°,* and both small and large positive/negative *AoAs*. Interpreting the results is ongoing.

SHINE has been a great experience this summer and has furthered my interest in Aerospace and Mechanical Engineering as well as research in general. I hope to continue learning about these subjects throughout my junior and senior years of high school, as well as in college.

Acknowledgements

I want to thank Professor Luhar for this amazing opportunity, as well as my PhD Mentors Idan Eizenberg and JP Chu for guiding us through our research this summer. I also want to thank Dr. Mills and the entire SHINE team for making this program possible. Lastly, I want to thank my family for their constant support.