Group 5: Evaluating Drag and Downforce Model Validity through Comparison of Physical Testing and Numerical Analysis

Group 5: Elliot Hong, Samuel Mankoff, Ming Chin, Lucero Jones-Ortiz, Stephen Salazar, Colby Winters

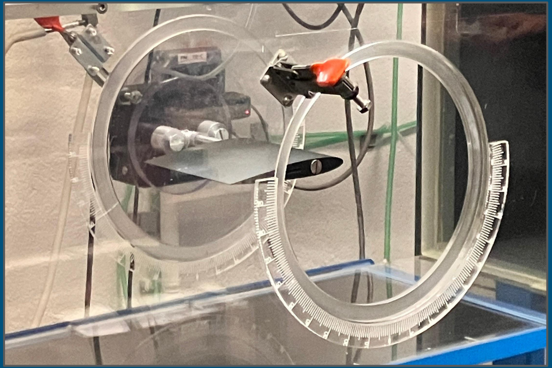
Introduction

- **Overview:** Analyze validity of numerical Computational Fluid Dynamics (CFD) model for drag and downforce of an airfoil against physical testing with wind-tunnel.

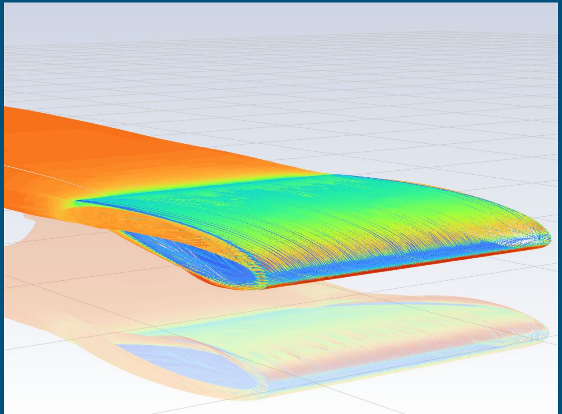
- **Motivation:** Physical testing is expensive and time consuming, so engineers commonly use numerical methods. We want to understand how accurate and useful they can be.

- **Hypothesis:** Numerical model drag and downforce values will remain within ±10% physical testing data

Physical Model



Versus



Numerical Model

Theory

- We are comparing numerical results from a CFD model within Ansys and recreating physical testing data from the wind-tunnel to compare the results
- CFD model solves the Navier Stokes equations in order to obtain drag and downforce values

Equation for Drag/Downforce

- Note the coefficient (C_d or C_l) in front changes depending on whether calculating drag or lift

CFD Solver

- K-Omega SST solver. Blends K-Omega for the walls & K-epsilon for the nodes in free space
- 0.1mm mesh size

$$\frac{m}{s} = \sqrt{\frac{2 \cdot p \cdot 132.3}{1.2}}$$

Velocity Equation using Pitot Tube Readings

$$\nabla . \underline{u} = 0$$

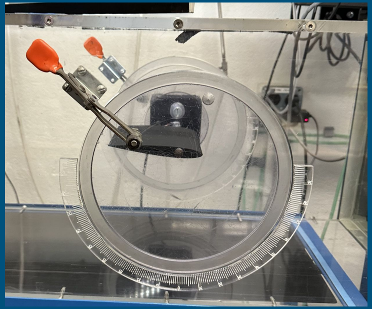
$$\rho \frac{D\underline{u}}{Dt} = -\nabla p + \mu \nabla^2 \underline{u} + \rho F$$

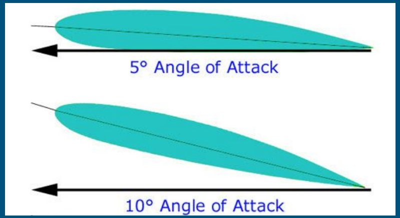
The Navier Stokes equations

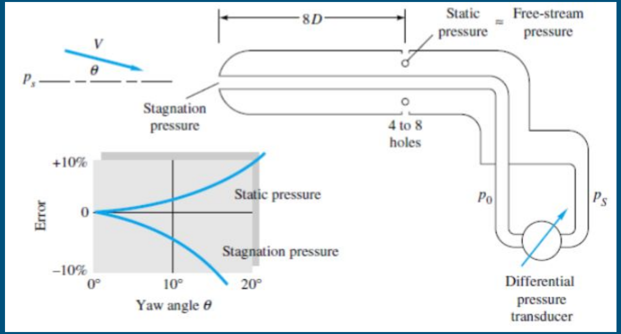
$$F_D = C_D A \frac{\rho V^2}{2}$$

Experimental Setup/Methods

- Setup (Wind-tunnel)
  - Mounted Airfoil to strain gauge within wind-tunnel, changed direction of strain gauge depending on whether measuring drag or downforce
- Procedures
  - Changed fan speed from 0 to 1700 revolutions per minute (RPM) at increments of 100 RPM for angles of attack ranging from 0° to 10° at increments of 5°
- Sensors:
  - Strain gauge - measures displacement
  - Pitot Tubes - measures fluid velocity,

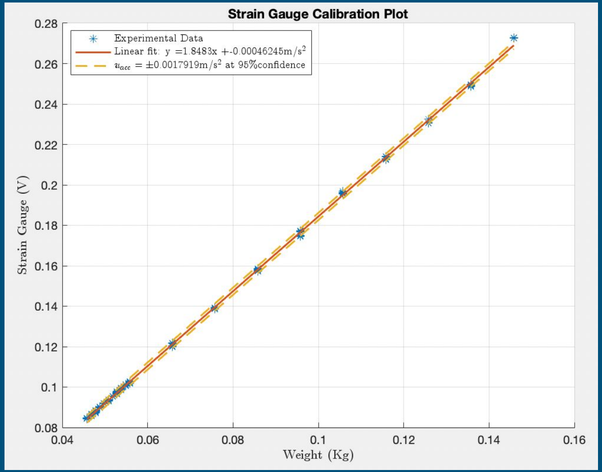


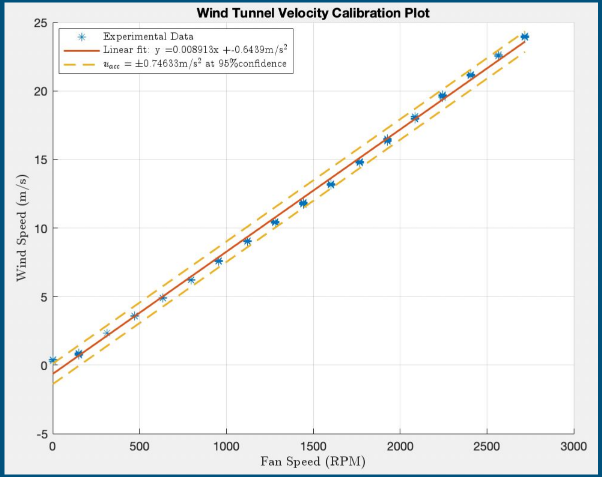




Calibration

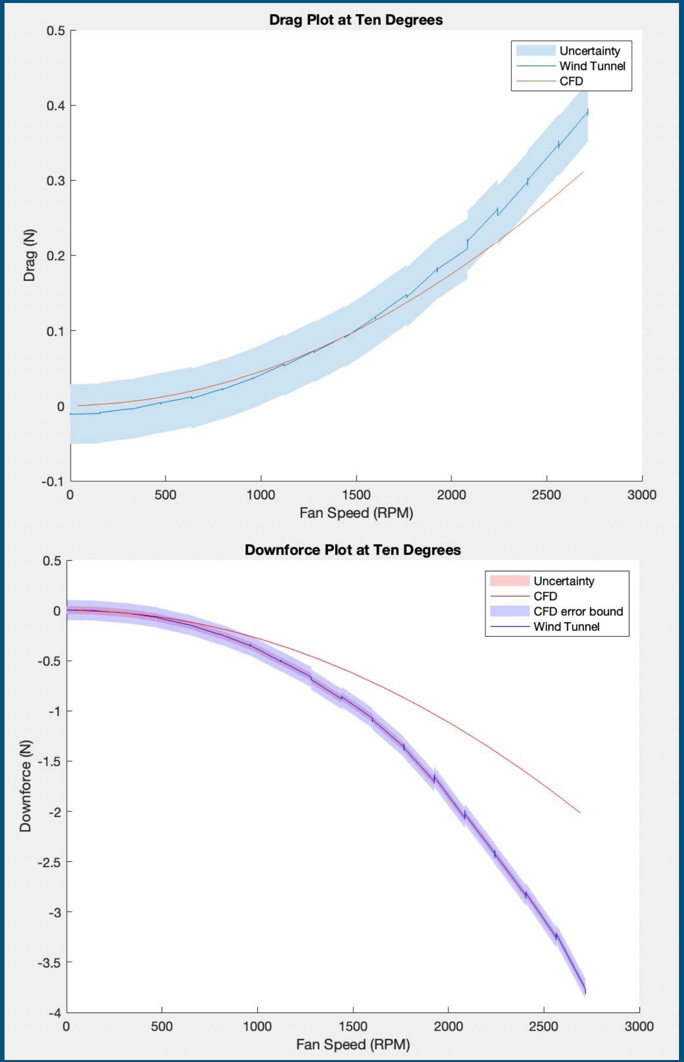
- Calibration
  - Performed calibration of strain gauge adding weights to strain gauge and measuring voltage at each weight
  - Velocity calibration was completed by running the wind tunnel at increments of 100 RPM for the motor and then converting the pressure readings from the pitot tube into velocities
- Conversion Factors (Slope):
  - 0.541 N/V
  - 0.00891 m/s / RPM





Results/Error Analysis

- Drag measured in CFD and wind tunnel fell within the uncertainty of the strain gauge, calculated using propagation of uncertainty, for all angles of attack (10° can be seen in the figure)
- Downforce measured in CFD and wind tunnel do not fall within ±10% error bound or within the uncertainty of the strain gauge, meaning ANSYS CFD is not a valid tool in simulating downforce



Conclusion

Sources of Error:

- Setup
  - Adjusted Angle of Attack using Protractor
- Sensor Noise and Uncertainty

What We Would Change?

- Getting more data points
  - Running multiple trials with consistent setup
- Using several sensors to measure the same variable allowing us to validate the datas accuracy

Results

- Ansys was not within a reasonable amount of error

References

[1] Endresen, Per Christian, and Heidi Moe Føre. "Numerical modelling of drag and lift forces on aquaculture nets: Comparing new numerical load model with physical model test results." International Conference on Offshore Mechanics and Arctic Engineering. Vol. 85888. American Society of Mechanical Engineers, 2022.

[2] Munson, Bruce Roy. *Fundamentals of Fluid Mechanics*. John Wiley & Sons, Inc., 2014.

[3] Hesse Staff. Wind Tunnel Datasheets. University of California, Berkeley., 2024.

Questions?

Discussion/Conclusion

Simulation

- Accurately predicts wind tunnel drag
- Does not accurately predict downforce

CFD

- CFD results for downforce falls outside the reasonable ±10% error bounds
- The model used for CFD is a good predictor of drag forces
  - Model needs to be adjusted to be a good predictor of downforce by potentially optimizing the meshing