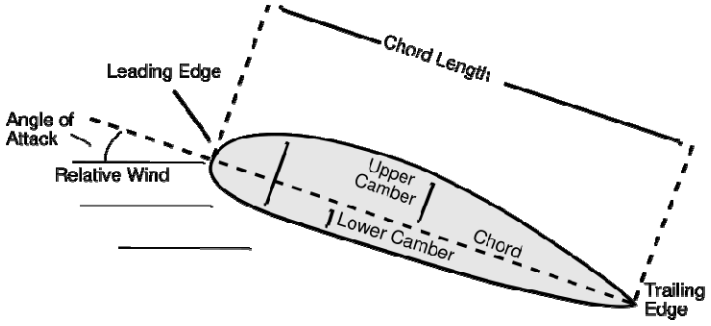
UNIVERSITY OF NEW SOUTH WALES

MECH3540, AERO3630, NAVL3620

CFD Assignment

An aerofoil is the shape of a wing or blade (of a propeller, rotor or turbine) or sail as seen in cross-section.



Any object with an angle of attack in a moving fluid, such as a flat plate, a building, or the deck of a bridge, will generate an aerodynamic force (called lift) perpendicular to the flow. Aerofoils are more efficient lifting shapes, able to generate more lift (up to a point), and to generate lift with less drag.

In this assignment you will investigate the flow patterns around the aerofoil for different angles of attack and Reynolds number, the pressure distributions and the resulting lift and drag.

Steps involved are listed below:

1. Create the geometry of an aerofoil with a cord length of 1 m using DesignModeler.

   Use a NACA0006 aerofoil if the last digit of your student number is even and a NACA0009 aerofoil if the last digit of your student number is odd (see data.pdf)

   If the sum of your student number is even, use angles of attack, α=2, 4 and 6°. If the sum of your student number is odd, use angles of attack, α=1, 3 and 5°.

   Instructions for creation of the geometry can be found in geom_instructions. pdf.

2. Generate a mesh using UMI in Workbench project. To set the problem as two dimensional, you will need to put one mesh element in the extruded direction and set symmetry boundary conditions.

3. Define a problem in CFX-Pre by setting the inlet velocity corresponding to Re=3x10^6, free slip at the top and bottom of computational domain and for the outlet set the average static pressure 0.

4. Run the case and obtain flow patterns (velocity vectors, streamlines), pressure distribution and lift and drag coefficients for your particular case.

5. Validate your results by comparison with experimental data provided in the file data.pdf.

6. Discuss your results.