CE 133 / ME 180, Lab project #12

Ahmed Bakhaty, Miquel Crusells-Girona

One of the most important applications of Computational Fluid Dynamics (CFD) is the study of airfoils' aerodynamics.

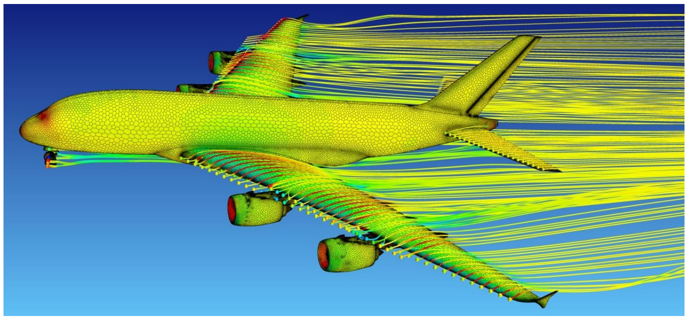

Figure 1: CFD model of the Airbus A380.

In this lab, you are going to analyze the behavior of a Cessna 172 idealized wing. Assume a stationary laminar flow for the analysis.

To begin with, create the region of study: a rectangle of width 10 m and height 3 m, centered at (3,0.05) m. Construct the airfoil as a parametrized curve $\mathbf{x}(s)$, defined by:

$$\mathbf{x}(s) = \begin{cases} (s, 0.4\sqrt{s}(1-s)) & \text{if } y \geq 0 \\ (s, -0.2\sqrt{s}(1-s)) & \text{if } y < 0 \end{cases} \tag{1}$$

In order to generate the area enclosed by this curve, make sure to 1) generate the union of both parametrized branches, and 2) convert the closed curve into an area. Both options can be found in the *Geometry* tab. Finally, generate the difference between the rectangle and the airfoil, and mesh the domain with a *normal* mesh. Figure 2 depicts the meshed geometry. Choose the fluid to be air.

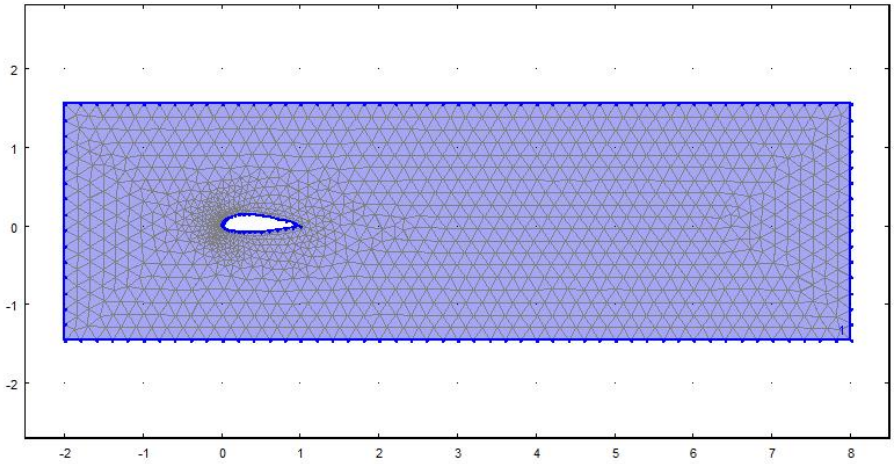

Figure 2: Meshed geometry of the problem.

Create flow boundary conditions: an inlet on the left side and an outlet on the right side, both prescribing a normal velocity of 40 m/s, a typical take-off airspeed for a Cessna 172. Also, use a slip wall for the top and bottom sides. Answer the following questions.

1. The equation that you are trying to solve is the Navier-Stokes equation, which includes an advective term. Write down the equation and identify the advective term. Is this term linear or nonlinear? Why?

2. Because of this reason, the problem has to be solved iteratively. Try to compute the solution with Comsol's default solver. What happens? Now, modify the relative tolerance to 0.3 and allow a maximum number of iterations of 150. Also, turn on pseudo-time stepping on the *Laminar flow* tab. Can you solve the problem now? The tolerance that one can achieve on this type of problems is very dependent on how sophisticated the solver is. We see that Comsol's solver is not very robust.

3. You assumed laminar flow to solve the problem. This assumption is represented in fluid mechanics by Reynold's number, Re, by stating that Re $< 10^7$ for exterior flows. Verify your hypothesis by estimating the Reynold's number of the problem,

$$\mathrm{Re} = \frac{v\rho L}{\mu}$$

where L is the airfoil chord length, v is the free stream velocity, ρ is the density and μ is the viscosity.

4. An airplane flies thanks to the difference in pressure that the flow creates around the wing. This pressure difference is due to the fact that the airfoil is not symmetric with respect to the x axis. With an air velocity of 40 m/s, compute the lift per unit wing length and relate it to the pressure distribution. The main parameter that pilots have in order to increase the lift (during take-off) is the angle of attack.

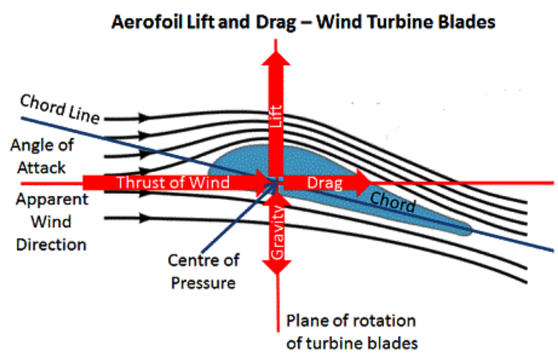

Figure 3: Drag, lift and angle of attack.

Make three copies of your Comsol model and rotate the airfoil with positive angles of attack, in degrees, of $\alpha_1 = 10$, $\alpha_2 = 20$ and $\alpha_3 = 30$. Plot the lift as a function of the angle of attack for an air velocity of 40 m/s. You may need to run two steps in order to solve the last case, i.e. run first $v = 20$ m/s and then $v = 40$ m/s starting from the previous solution as your initial guess. What do you observe? Also, include a streamline plot for each angle of attack.

There is a critical angle of attack, for a given airspeed, at which the airplane stalls. This means that the lift is no longer greater than the weight of the aircraft, and the aircraft starts to fall. Click here to observe what happens when an aircraft stalls. This is one of the very first conditions that pilots learn to mitigate. What do you think a pilot can do to mitigate this situation?