Simulating Fluid Flow past a Hydrofoil (Project 2)

MECH 479/587 - Computational Fluid Dynamics
Winter Term 1

1   Problem Setup

In this project you will explore the CFD modeling of turbulent flow around a two-dimensional wing geometry, commonly used in airplanes, wind turbines, and marine structures. To save time, the geometry and mesh files are provided for your simulations.

Each student is given a different mesh around an airfoil; hence the CFD results and reports will not be similar. The differences between the airfoils are, however, marginal—some are designed to increase the lift coefficient, some to maximize the lift to drag coefficient, and some to meet other engineering requirements. In case you know how to mesh a domain and would like to create your own grid, the x-y coordinates of each airfoil are provided in a separate folder. The following link is one of the many resources you can use to learn how to create a mesh around an airfoil: ANSYS Fluent tutorials by Cornell

Simulation Guidelines

Here are some guidelines for setting up the simulation—generally. We will discuss some of these aspects in further detail in Tutorial 7.

- Use a density-based solver.

- Use the ideal gas model for air (as your fluid). Do not change other properties.

- Use the k-omega SST Reynolds-Averaged Navier-Stokes (RANS) turbulence model.

- In steady-state cases, use a Courant Number of 50 for faster convergence. Feel free to explore other options as well.

- Set the operating pressure to the value specified in the Tutorial 7 (P= 59607.1 Pa), unless you have experimental data for a different operating pressure.

- Use the 'pressure-far-field' boundary type for 'velocity.inlet', 'bottom.wall', and 'top.wall' boundaries. Use the 'pressure-outlet' boundary type for the outlet boundary. Set the boundary conditions as follows:

    - Inlet boundary (also top and bottom walls):
        * T = 273 K, Gauge Pressure = 0 atm
        * Turbulent Viscosity Ratio and Turbulent Intensity: default values
        * Angle of Attack and Mach Number - see section 2

- Outlet boundary:
    * T = 273 K, Gauge Pressure = 0
    * Backflow Turbulent Viscosity Ratio and Backflow Turbulent Intensity : default values.
  - Airfoil boundary:
    * T = 300 K, No-slip boundary condition for velocity.
    * Set the reference values based on inlet boundary data. Use reference area as $1m^2$.
    * Use the drag or lift coefficient (C_D or C_L) to monitor the convergence history. For the steady-state cases, run the solver for atleast 4000 iterations. If you observe convergence (no variation) in C_D and C_L, you can terminate the calculation.

2 Report Requirements

Please note that the quality of the report and the discussion of the results makes up part of your grade.

- Introduction: State the objectives of the project.

- Material and methods: Provide enough details to allow the work to be reproduced.

- Results and Discussion: Run the following cases and report the results. Your presentation of the results should be clear and concise. Your discussion of the results should be comprehensive.

  1. For Mach number = 0.3, run the steady solver with Angles of Attack (AoA) = $\{2, 4, 8\}°$. In each case report the values of C_D, C_L, and $\frac{C_L}{C_D}$. Plot C_L and C_D as a function of the angle of attack (in radians). Determine the slope of C_L v/s AoA and compare it with the analytical slope calculated based on the inviscid thin-airfoil theory.

  2. Present pressure, velocity or vorticity field contours for two of these cases and link your flow-field observations with the lift-drag coefficient graphs.

  3. Run the steady case with Mach number = 0.9 and AoA = 4°. Report CL and CD values, and also pressure and velocity contours. Discuss these results. What is the main difference between this case and the cases in the previous step? Do you think a finer mesh is needed to solve this problem more accurately?

  4. For Mach number = 0.3, run the transient solver with AoA = 15°. Choose a proper time-step and report your observations (specially the vorticity contour) at some specific end time. What is the main difference between this case and the previous cases (with smaller AoA)?

- Conclusions: Summarize the main conclusions of the study.

Note 1: In any of the cases above, if you do not get the results that you expect, please provide your CD or CL convergence history (that is, the plot you use to monitor

convergence) and discuss the possible causes of the abnormal behavior. Does using a smaller angle of attack give you more reasonable results? What about a smaller Mach number? Do you think the mesh plays a role in this?

Note 2: In case you do not know the typical behavior of the lift and drag coefficients for a typical airfoil, you could search for it online and make sure that your results are reasonable. Two figures are attached here with some typical plots for airfoil simulation results.
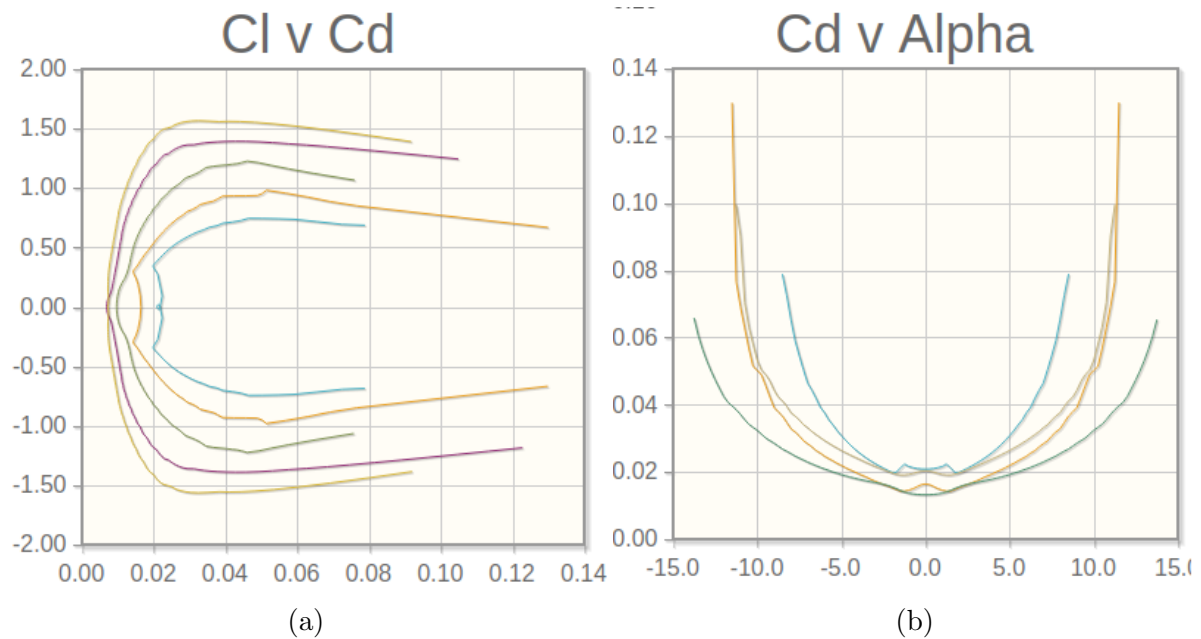

(a)   (b)

Figure 1: (a) C_L vs C_D and (b) C_D vs AoA for NACA 0012 airfoil section for different Re. (Source : airfoiltools.com

Report Guidelines

- The report should not exceed 8 pages.

- Please be professional in producing the figures.

- Restrict your figures and schematics to less than 1/3 of a page.

- Maintain readability of your report, i.e. relevant plots should accompany the discussion of a particular question in the report.

Extra Step: Validation (not mandatory)

- Unfortunately, we do not have experimental data for all the airfoils to ask you to make a comparison between your simulations and experimental results. But the name of each airfoil is the actual name, and you can search for some experimental data related to that specific or similar airfoils in the literature. You can search for experimental results in the following links—but you may try other resources as well:

  https://turbmodels.larc.nasa.gov/

  https://m-selig.ae.illinois.edu/ads/coord_database.html

- Comparing your results with experimental data is **not mandatory**; yet it would be helpful for you to validate your numerical results. Remember, if you find some experimental data for the airfoil, set the boundary conditions as given in the experiment so as to compare the results in the end.

- In case you do not know the typical behavior of the lift and drag coefficients for a typical airfoil, you could search for it online and make sure that your results are reasonable. Two figures are attached here with some typical plots for airfoil simulation results.
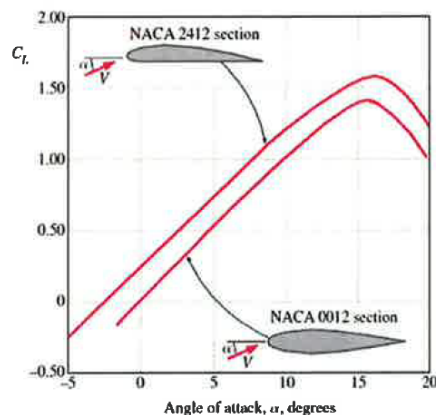
FIGURE 11–50
The variation of the lift coefficient with the angle of attack for a symmetrical and a nonsymmetrical airfoil.
From Abbott (1932).
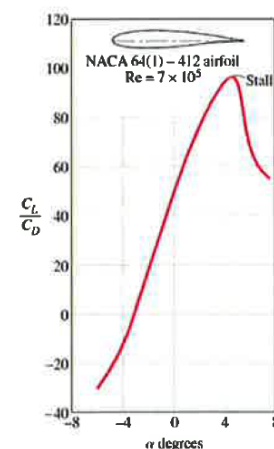
FIGURE 11–43
The variation of the lift-to-drag ratio with angle of attack for a two-dimensional airfoil.
From Abbott, von Doenhoff, and Stivers (1945).

Airfoil No.	Student ID	Airfoil
1	72258213	NACA_M8
2	64086168	FX72_LS_160
3	21582465	EPPLER 420
4	92986264	WORTMANN FX72-MS-150B
5	74835083	Ornithopter
6	30344204	E423
7	10294437	RONCZ-1046-VOYACER-CANARD
8	30356547	AH-93-K-13015
9	44303519	DEFIANT-CANARD-BL20
10	98107899	CLARK-YM-18
11	36914398	AG37
12	10279157	ONERA-OA206
13	25642190	FX72_LS_160
14	40112666	BOEING-VERTOL-VR-9
15	12705299	Ornithopter
16	30459895	EPPLER-397
17	39005590	ONERA-NACA-CAMBRE
18	55693972	NPL-9660
19	71746572	HQ-1012
20	46287371	DEFIANT-CANARD-BL20
21	77761005	PSU-90-125WL
22	61062287	MH-200-1297
23	58974353	EPPLER-331
24	98542558	RONCZ-1046-VOYACER-CANARD
25	93675585	AH-93-K-13015
26	27596832	NACA_M8
27	10701167	EPPLER 420