1. Notes on the software
2. Assigned exercise (submission via Blackboard; deadline: Thursday 8 March, 11 pm)

---

1. NOTES ON THE SOFTWARE

1.1 Accessing and Running the Program

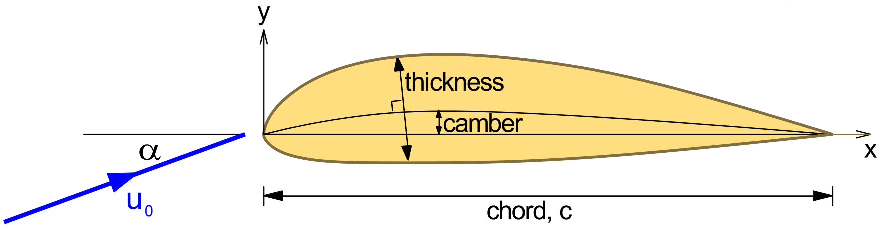
The following files must be downloaded from the CFD web pages:

`aerofoil.exe`	(graphical user interface)
`streamaero.exe`	(CFD solver)
`gridaero.exe`	(grid-generator)
`coordinates.dat`	(sample aerofoil coordinates)

Double-click `aerofoil.exe` to start. All files will be saved in the same folder.

1.2 Flow Considered

The program simulates 2-d, incompressible flow around an aerofoil at angle of attack α.



The aerofoil section may be *either* an analytical profile for the NACA 4-digit series:

NACA *mpth*

where

m = maximum camber in percentage of chord;
p = position of maximum camber in tenths of chord;
th = maximum thickness in percentage of chord;

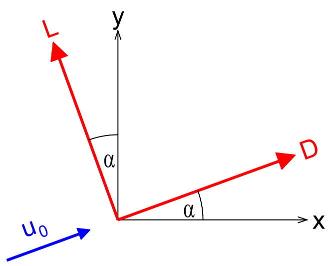
or any other aerofoil for which you can provide coordinates (in file `coordinates.dat`). The default profile is the symmetric NACA 0009 aerofoil.

All variables are non-dimensionalised using the uniform approach-flow velocity U_0 and the aerofoil chord c; i.e. $X = x/c$ and $U = u/U_0$. The Reynolds number is defined as $\mathrm{Re} = U_0 c/\nu$.

The program outputs to screen the equation residuals and, at both backup and completion:

- any separation or reattachment points (including the upstream stagnation point);
- minimum and maximum y^+ values;
- drag and lift coefficients, based on force components (per unit span) parallel and perpendicular to the approach flow:

$$c_D = \frac{drag}{\frac{1}{2}\rho u_0^2 c}, \qquad c_L = \frac{lift}{\frac{1}{2}\rho u_0^2 c}$$



1.3 Program Operation

For a particular case you should:

- set aerofoil and grid parameters;
- generate a grid;
- set the angle of attack (a “case” parameter);
- solve;
- analyse/plot.

As all transfer between components is done by reading and writing files, you can exit the graphical user interface at any stage and restart from the same point.

1.4 Main Buttons

Grid

[Set up (laminar)]	set aerofoil and grid parameters for the default laminar flow
[Set up (turbulent)]	set aerofoil and grid parameters for the default turbulent flow
[Edit]	edit grid and aerofoil parameters
[Run]	run the grid-generator

Case

[Set up]	set default flow, transition and flow-specific plot parameters
[Edit]	edit case parameters (including the angle of attack)

Solver

[Set up (laminar)]	set parameters for a default laminar-flow calculation ($Re = 10^3$)
[Set up (turbulent)]	set parameters for a default turbulent-flow calculation ($Re = 10^6$)
[Edit]	edit solver parameters
[Run]	run the CFD solver

Plots

[Set up (near)]	set parameters for a default plot focused on the aerofoil
[Set up (far)]	set parameters for a default plot showing the whole domain
[Edit]	edit general plot parameters
[Grid]	plot the grid
[Blocks]	plot the domain decomposition into blocks
[Profiles]	plot streamwise-mean-velocity profiles along the aerofoil **(**)**
[Vectors (all)]	plot mean-velocity vectors at all nodes
[Vectors (reg)]	plot interpolated mean-velocity vectors on a regular grid **(**)**
[Streamlines]	plot streamlines
[Pressure]	plot pressure contours
[TKE]	plot turbulent-kinetic-energy contours

(**) Regular-grid velocities and profiles are only output at the end of a 2-d flow calculation.

Graphs

[Cp graph]	plot a graph of pressure coefficient (parameters in Case - Edit)
[Cf graph]	plot a graph of skin-friction coefficient (parameters in Case - Edit)

Hard copy

[File]	save the current plot as a picture file (type `.png`)

2. ASSIGNED EXERCISES

Laminar Flow

(L1) Grid Generation

Set up and generate the default laminar-flow grid. Include suitable grid plots in your report.

(L2) Symmetric Aerofoil

Set up the default case and solver parameters for laminar flow. Solve, and include plots of streamlines, shaded pressure, velocity profiles (not vectors) and c_p graph in your report.

(L3) Aerofoil Drag

For the calculation above, record the drag coefficient c_D. Repeat grid and solver calculations with double the number of grid cells in each direction. (Double `NChord`, `NWake` and `NRadius` and re-run the grid generator and flow solver.) Does your solution give a satisfactorily grid-independent value for drag? Compare your results for drag coefficient with Blasius' theory (look it up) for a *flat plate* of similar length:

$$c_D = \frac{1.33}{\sqrt{\text{Re}}} \quad (\textit{per side})$$

Suggest reasons for any differences.

(L4) Effect of Angle of Attack

Regenerate the default laminar grid. With the default laminar-flow solver parameters (Re = 1000) compute the flow at angles of attack 3° and 6°. In each case:

- record drag and lift coefficients (as reported by the program);
- record any separation point if there is one.

For the case $\alpha = 6^{\circ}$ only:

- plot streamlines, shaded pressure, velocity profiles, c_p and c_f graphs.

Note that the program reports all points of near-surface flow-direction reversal. You must exclude any corresponding to stagnation or reattachment points (using ... common sense).

Explain how flow separation can be identified from the skin-friction graph and lift can be *estimated* from the pressure-coefficient graph. Give two reasons why this is only an estimate.

(L5) Effects of Camber and Thickness.

Generate the grid with default cell parameters but cambered aerofoil (NACA 2309). Compute flow with Re = 1000 and angle of attack 3°. Repeat for a thicker aerofoil (NACA 2312). Record drag and lift coefficients and separation point (if any) in each case. Comment on the effects of camber and thickness.

Turbulent Flow

(T1) Lift vs Angle of Attack

Set the default grid parameters for a *turbulent* flow; (the default symmetric aerofoil is NACA 0009). Run the grid generator. Set the default solver parameters for turbulent flow ($Re = 10^6$; Launder-Sharma turbulence model).

Compute the flow for angles of attack $\alpha = 0°, 3°, 6°, 9°, 12°$. Record (in tables and graphs) lift and drag coefficients against angle of attack. Also included in the table and in the lift graph should be the thin-aerofoil theory (White, Chapter 8) for lift coefficient:

$$c_L \approx 2\pi \sin\alpha$$

(T2) Effect of Camber on Lift

Repeat the calculations and graphs in T1 above for the cambered NACA 2309 aerofoil. This time compare the computed lift coefficient with

$$c_L \approx 2\pi \sin(\alpha + \frac{2h}{c})$$

h = maximum camber. (Angle α must obviously be in radians to do the addition here.)

(T3) Calculation of Lift and Drag

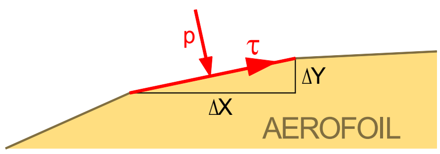
Set default grid and solver parameters for turbulent flow. In the grid parameters, change “`Analytic profile?`” to “`No`”. The grid generator will then read the aerofoil definition from `coordinates.dat` (which corresponds to the Göttingen 804 (EA8) aerofoil section, downloaded from the UIUC airfoil coordinates database). Calculate flow at angle of attack 6º.

In your Report, include a plot of the grid, and the resulting drag and lift coefficients.

The code produces a data file `surface.dat` containing columns with the following data:

$$X_{mid} \quad Y_{mid} \quad c_p \quad c_f \quad \Delta X \quad \Delta Y$$

where X_{mid} and Y_{mid} are non-dimensional cell-face-centre coordinates on the surface of the aerofoil, and $(\Delta X, \Delta Y)$ is the corresponding surface line segment. c_p and c_f are pressure and skin-friction coefficients defined by:



$$c_p = \frac{p}{\frac{1}{2}\rho u_0^2}, \qquad c_f = \frac{\tau}{\frac{1}{2}\rho u_0^2}$$

τ is the shear stress acting tangentially to the surface in the direction of $\Delta\mathbf{X}$; the sense of both correspond to a *clockwise* traversal of the aerofoil (see diagram).

Show how to use this data to produce drag and lift coefficients for the aerofoil, separating values for pressure, viscous and total forces. Confirm that the total drag and lift coefficients yield the values reported by the program and comment on the relative size of the pressure and friction contributions to lift and drag.