6th assignment: Evaluation of $C_L \times \alpha$ curve for the NACA0012 airfoil

This assignment is about the flow solution around an airfoil NACA0012. A scheme of the geometry and boundary conditions can be seen below.

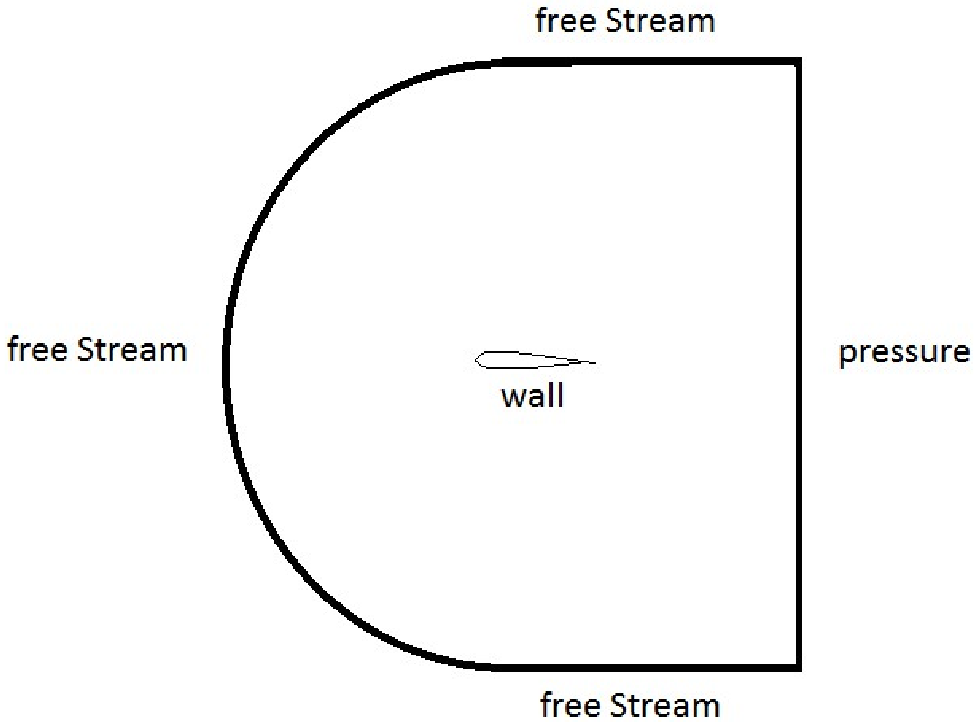

Fig. 1) Flow domain and boundary conditions.

The NACA0012 airfoil has a chord L = 1m and the free stream velocity is U = 1m/s. The numerical results will be compared with experimental results for Re = UL/ν = 2×10^6. A kinematic viscosity $\nu = 5\times10^{-7}$ m^2/s will be used in the simulations.

1) Choose a directory to save the case and the case name.

2) Import the mesh. The mesh used in the assignment is in the GAMBIT format and is named "naca0012.neu".

3) After importing the mesh use the "Mesh" menu to specify the "empty" boundary conditions, so that the flow is in the xy plane. You should specify also the "wall" and "patch" boundary conditions. Notice that a selected boundary acquires a soft red color.

4) In the "Setup" menu you should choose the solver. Select a solver for steady, incompressible and turbulent flow. The simpleFoam solver is going to be used for the simulation.

5) In the "Turbulence" menu select turbulent flow. The k-ω SST and Spalart-Allmaras models will be used.

6) In the "Transport properties" you should set the value of the kinematic viscosity. Set $\nu = 5\times10^{-7}$ m^2/s.

7) In the “Discretization” menu there are four submenus, for time discretization, spatial discretization (Convection), gradient calculation and interpolation. In the time discretization the simulation should be selected as steady. In the spatial discretization the linear upwind scheme should be selected for the velocity instead of the default upwind scheme. The turbulent quantities can be solved with the 1st order upwind scheme. In the other submenus the default options are ok.

8) In the “Solution” menu there are four submenus for solvers, Simple algorithm options, residuals and relaxation. Lower the residuals values to 10^{-5}. If using the consistent SIMPLE algorithm opt for a 1.0 relaxation for pressure and 0.7 relaxation for velocity. Set a 0.1 relaxation for the turbulent quantities. The other options can be left with their default selections.

9) Ignore the “*Operating Conditions*” and “*cell zones*” menus.

10) In the “Boundary Conditions” menu set the velocity and pressure at the “patch” boudaries. In the free stream, stipulate the velocity vector components and a zero gradient Neumann condition for the pressure (zeroGradient). Here we have to deal with the issue of angle of attack. According to the figure below, the velocity in the *x* direction will be given by $U = U_\infty \cos\alpha$ and the velocity in the *y* direction will be given by $V = U_\infty \operatorname{sen}\alpha$.

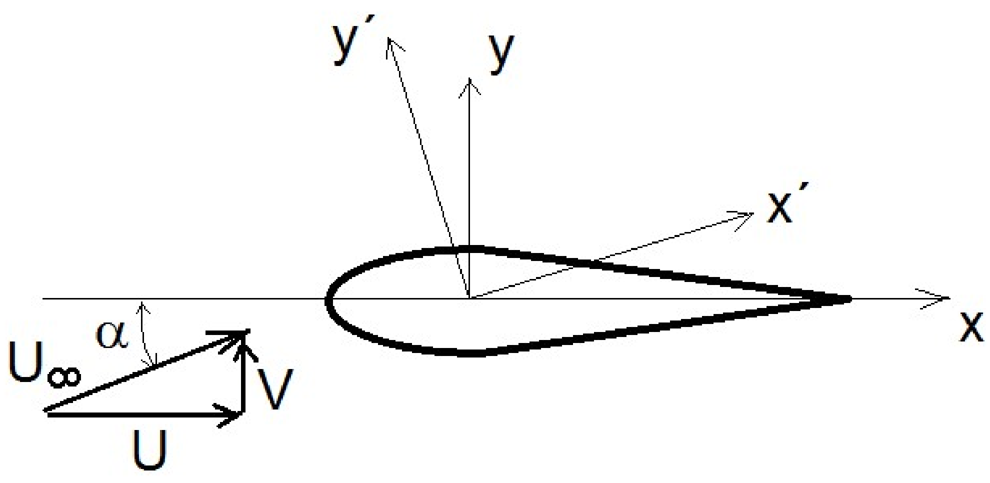

Fig. 2) Velocity components and angle of attack.

Thus, for an angle of attack α = 12º, we will have the data in the figure below:

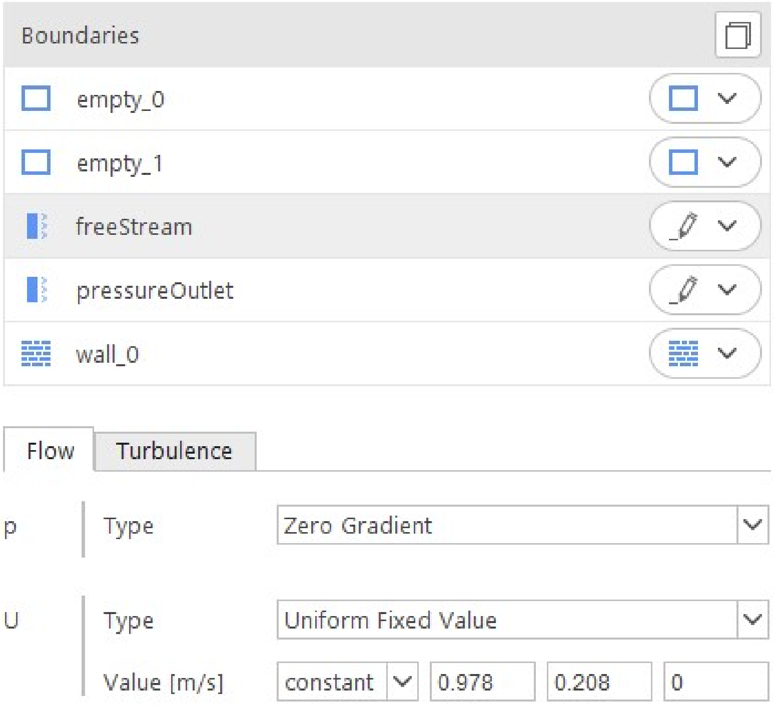

Fig. 3) Boundary condition for the free stream velocity and $\alpha = 12°$.

Still in the free stream, the values of the kinetic energy k and the specific dissipation ω must be specified. Use $k = 0.00015\ m^2/s^2$ and $\omega = 300\ s^{-1}$, according to the figure below:

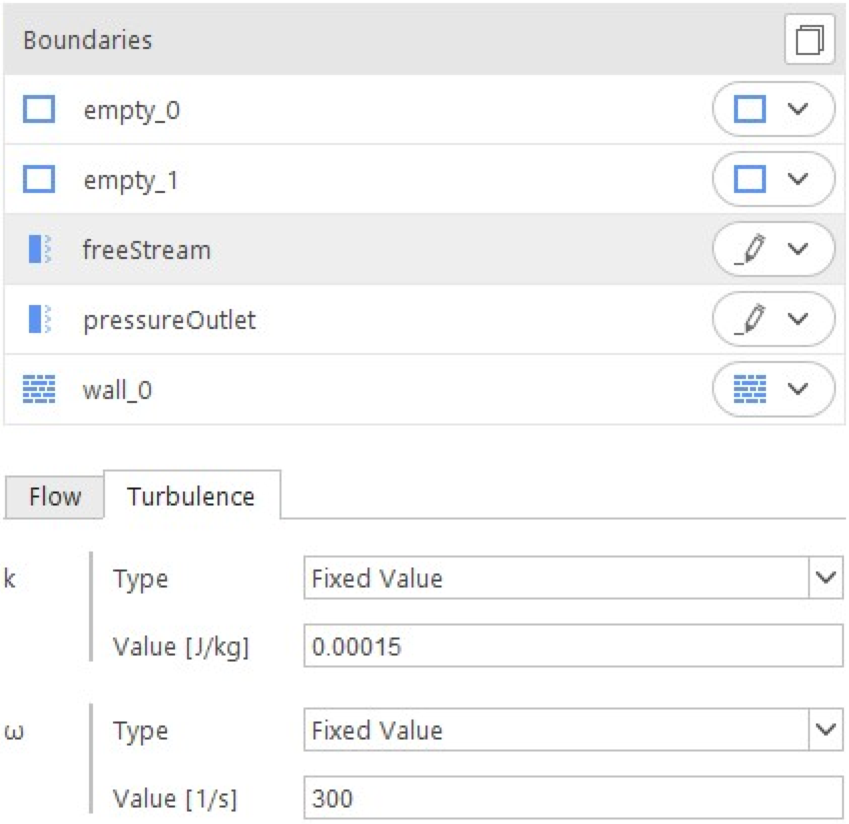

Fig. 4) Free stream boundary conditions for k and ω.

When repeating the simulation using the Spalart-Allmaras model, use in the free stream a boundary condition $\tilde{\nu} = 3\nu$, or $\tilde{\nu} = 1.5\times10^{-6}$ m^2/s.

In the outflow (*pressure*) set an arbitrary uniform pressure (typically 0 m^2/s^2 ; the unit is due to the fact that the pressure is divided by the density), and a zero gradient for velocities and turbulent quantities.

In the “wall” boundary a zero velocity is specified using the “fixedValue” boundary condition and a “zeroGradient” condition is used for pressure. Do not use openFOAM's no-slip condition for velocity (although in theory that's right, it does not work well). Set $k = 0$, and use the standard wall function for ω. When using the Spalart-Allmaras model, set $\tilde{\nu} = 0$ m^2/s on the wall.

11) In the “Initial Conditions” menu, set the initial estimate for the quantities to be solved. Use the data in the figure below:

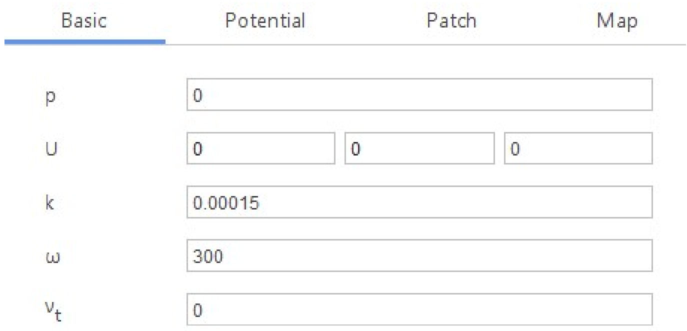

Fig. 5) Initial conditions.

When repeating the simulation using the Spalart-Allmaras model, use $\tilde{\nu} = 1.5\times10^{-6}$ m^2/s as initial value for the transported eddy viscosity (equal to the free stream boundary condition).

12) In the “Monitors” menu you should specify the monitoring of the coefficients of the forces on the airfoil wall. Recall that drag (Drag) is computed in the free stream direction, and lift (Lift) is computed in the direction orthogonal to the free stream direction. Thus, the drag direction is the x' direction in figure (2), and the lift direction is the y' direction. It turns out that we have $\vec{e}_{x'} = \cos\alpha\,\vec{e}_x + \operatorname{sen}\alpha\,\vec{e}_y$ and $\vec{e}_{y'} = -\operatorname{sen}\alpha\,\vec{e}_x + \cos\alpha\,\vec{e}_y$. The reference area is 1m^2, because the chord is 1 m. In the figure below the data for $\alpha = 12$º is shown:

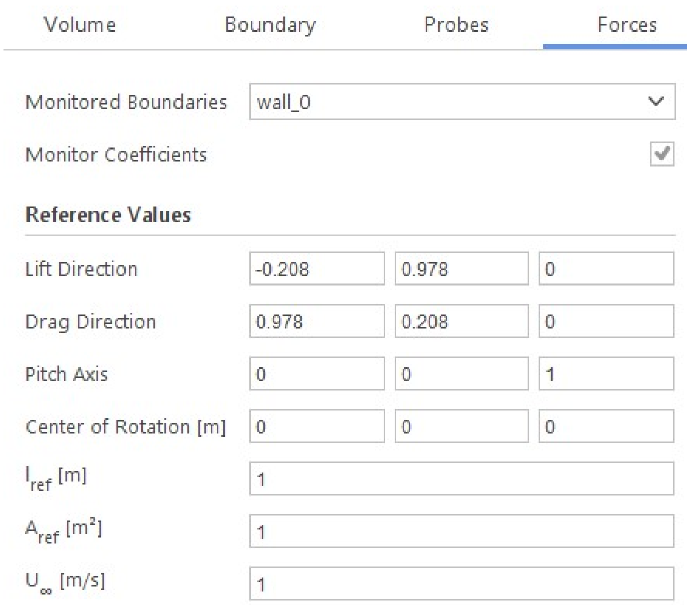

Fig. 6) Data for calculating drag and lift coefficients when $\alpha = 12^o$.

13) In the "Run" menu you should specify the maximum number of iterations (around 10000) in the "Time Control" submenu, and the frequency to save files of the solution fields in the "Output" submenu.

15) Save the case using the floppy disk button and run.

16) After convergence, the C_D and C_L values from the last iteration are the valid values for the corresponding angle of attack. Change the input data of the velocity components at the free stream boundary and the directions of the axes of the force coefficients for each value of angle of attack α. Simulate the angles of attack α = 5º, 10º, 12º, 13º, 14º and 15º. Compare your curve with the experimental curve below. Get ready for long simulations, with up to 5000-6000 iterations.

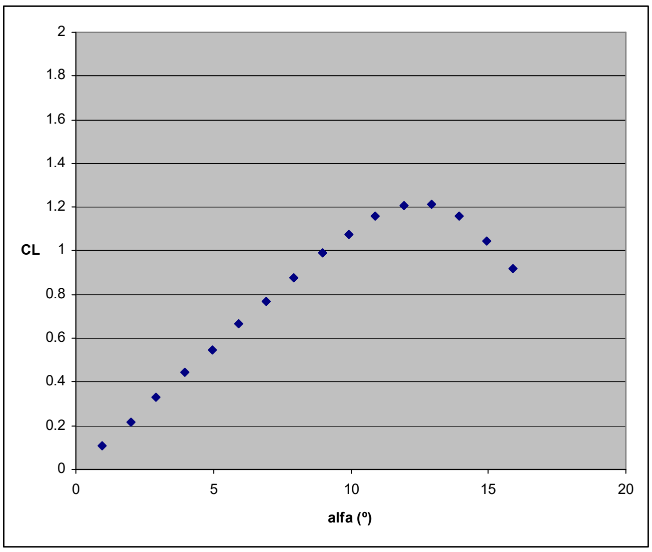

Fig. 7) NACA0012 airfoil $C_L \times \alpha$ curve. Experimental values for Re = $2{\times}10^6$.

α (º)	C_L
1	0.11
2	0.22
3	0.33
4	0.44
5	0.55
6	0.66
7	0.77
8	0.88
9	0.99
10	1.07
11	1.16
12	1.21
13	1.22
14	1.16
15	1.04
16	0.92