Assignment for AE4202 – CFD for Aerospace Engineers

Introduction

The assignment should be performed in groups of two students. You are free to choose your partner. Both team members should equally contribute to the work and submit a joint report. Though not recommended, you could also do the work alone or submit individual reports.

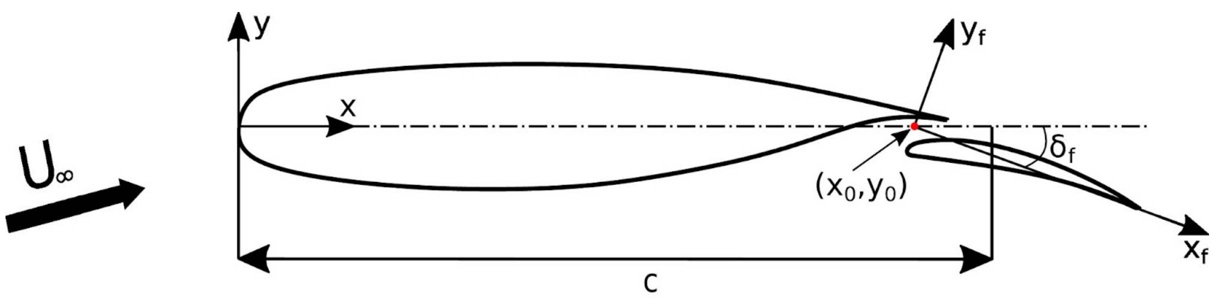
The deliverable is a short report with about **10 pages**, excluding title page and list of references. The report should be submitted through Brightspace by **15 November 2023**.
If you submit the report late, one point will be subtracted from the grade; a second point will be subtracted if you submit more than one week past the due date.

Problem description

Your task is to perform 2D steady-state RANS simulations of the NLR7301 two-element airfoil, at angles of attack alpha = 6 and 13.1 degrees, chord-length Reynolds number 2.51 million and Mach number 0.185.

You can consider ambient flow conditions with static temperature 293K and static pressure 101325 Pa. Model air as a compressible ideal gas with a dynamic viscosity given by Sutherland's law.

The geometry is sketched below and coordinates of the airfoil are provided in separate files. The wing and flap coordinates are normalized by the reference chord length, which is based on the wing chord with the flap retracted. The inflow boundary condition with turbulence intensity 1% should be imposed far upstream of the airfoils leading edge. The reference experimental data of the pressure coefficient are reported in different reference systems for the wing (x,y) and the flap (x_f,y_f). The origin of the flap coordinate system in wing coordinates is (x_0/c,y_0/c)=(0.90067,0.00260) and the angle of the flap is $\boldsymbol{\delta}_f$=20°.



Tasks

1. Load the coordinates into the mesh generator of your choice. Create a suitable computational domain that will allow you to rotate the geometry easily.

2. Mesh the computational domain with a coarse, structured multi-block grid with about 20.000 cells (*grid1*). Pay attention to orthogonality, smoothness and resolution of the boundary layers. Visualize and describe blocking and mesh in the report and discuss the mesh quality.

3. As boundary conditions, impose an adiabatic wall at the airfoil, flow direction, total temperature, total pressure and turbulence intensity at the inflow and static pressure at the outflow. Use a 'high-resolution' scheme for the spatial discretization and the k-omega turbulence model. Report the simulation setup and discuss your choices.

4. Perform a steady state simulation at angle of attack 6 degrees with the solver of your choice. Plot the solver residuals to verify convergence of the simulation for this and all subsequent simulations.

5. Visualize the flow field: produce contour plots for the static pressure and velocity. Plot the non-dimensional pressure coefficient C_p along the wing and the flap surfaces together with the provided reference data. Extract the non-dimensional lift and drag coefficients and summarize them in a table together with the reference coefficients.

6. Compute and plot the y+ value of *grid1* along the wing and the flap. Refine the grid such that y+ < 1 for the refined grid (*grid2*). Visualize the mesh, pay attention to orthogonality and smoothness. Perform the simulation and plot y+ for *grid2* to verify that the target resolution was achieved. Critically discuss the mesh quality.

7. Perform the same post processing and visualizations as done for *grid1*. Include the new C_p curve in the same figure and the lift and drag coefficients in the same table as the results for grid 1. Describe and evaluate the results.

8. Now rotate the fine mesh *grid 2* and simulate the other angle of attack (13.1 degrees). Plot the solver residuals and repeat the same post processing and visualizations as done before, compare with the provided reference data and discuss your observations.

9. In order to quantify the effect of numerical errors, repeat the simulations on the rotated *grid2* with a first-order upwind scheme. In order to quantify the effect of the turbulence model, repeat the simulations on the rotated *grid2* with a turbulence model that can better account for anisotropic turbulence. Visualize and analyze the effect on the results.

10. Summarize your results, discuss them critically and suggest possible improvements.