

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 **16 November 2022**.

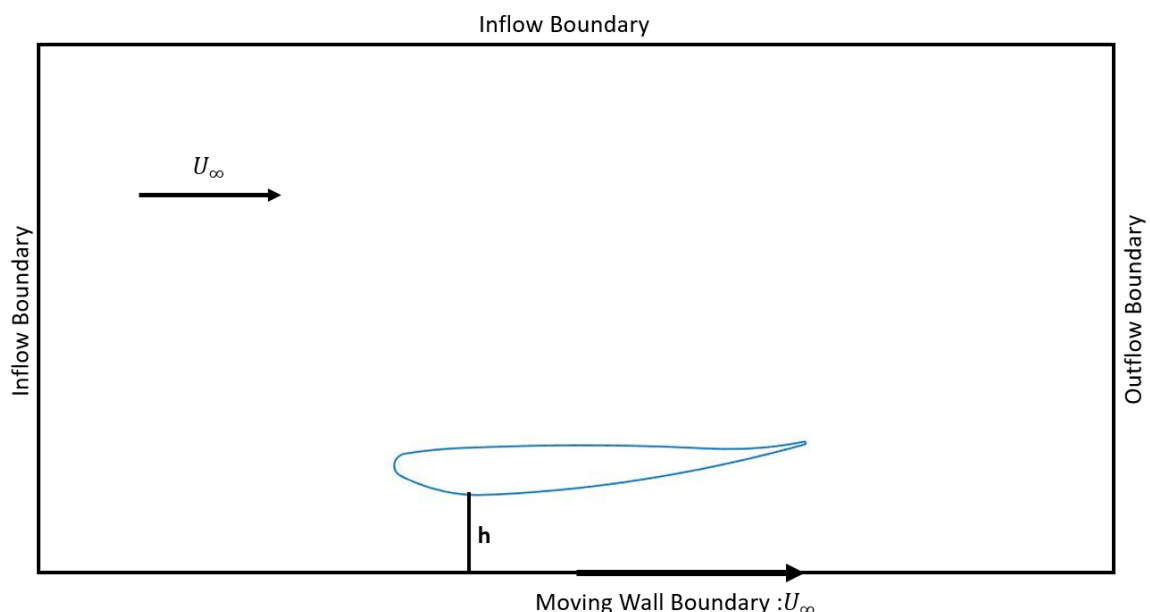
If you submit the report late, one point will be subtracted from the grade; a second point will be subtracted if you submit more than one week past the due date.

Problem description

Your task is to perform 2D steady-state RANS simulations on a Tyrrell 026 F1 car front wing at two different wing heights of $h/c = 3.36$ (freestream region) and $h/c = 0.134$ (ground effect region), with h being the height from the ground and c being the chord length of the airfoil. The free stream Mach number is $M_\infty = 0.0874$ and the chord-length Reynolds number 462000.

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 and boundary conditions are sketched below, coordinates of the airfoil are provided in a separate file. The inflow boundary condition with turbulence intensity 0.2% should be imposed far upstream of the airfoils leading edge, and at the upper boundary. The bottom wall should be a moving boundary with velocity equal to the free-stream value. Chose an appropriate domain size; the sketch is not to scale!



Tasks

1. Load the coordinates into the mesh generator of your choice. Create a suitable computational domain positioning the wing at a distance $h/c = 3.36$ from the ground.
2. Mesh the computational domain with a coarse, structured multi-block grid with about 20.000-25000 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, and an adiabatic moving wall for the ground. Impose 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 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 airfoil surface together with the provided reference data. Extract the non-dimensional lift and drag coefficients and summarize them in a table together with the reference lift coefficient.
6. Compute and plot the y^+ value of *grid1* along the airfoil. 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. Shift the wing to a distance from the ground of $h/c = 0.134$ and generate a fine grid, with resolution similar to grid 2. Plot the solver residuals and repeat the same post processing and visualizations as done before. Compare these results with the case at $h/c = 3.36$ and with the provided reference data. Discuss your observations.
9. In order to quantify the effect of numerical errors, repeat the simulation at $h/c = 0.134$ with a first-order upwind scheme. In order to quantify the effect of the turbulence model, repeat the same simulation 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.