C3.1 Turbulent Flow over a 2D Multi-Element Airfoil

Questions: Marco Ceze, *mceze@umich.edu*

If you intend to run this case, please contact the case administrator to facilitate communication.

1 Overview

This problem is aimed at testing high-order methods for a two-dimensional turbulent flow with a complex configuration. It has been investigated previously with low order methods as part of a NASA Langley workshop. The target quantity of interest is the lift and drag coefficients at one free-stream condition, as described below.

2 Governing Equations and Flow Conditions

The flow is to be modeled by the 2D Reynolds-averaged Navier-Stokes equations. The properties of air to be used are $\gamma = 1.4$ (ratio of specific heats) and $Pr = 0.71$ (Prandtl number). In the 2^{nd} HOW we had a mix of constant and variable dynamic viscosity results and we found that the drag and lift values do not vary significantly between the two cases. However, we found the nonlinear problem to be easier to converge using Sutherland's law for viscosity with $T_{\text{ref}} = 288K$. The participant can choose to run either case.

The choice of turbulence model is left up to the participants; recommended suggestions are: 1) the Spalart Allmaras model, and 2) the Wilcox k-omega model. Please report any modifications made to the original published models. Also, we encourage you to verify your turbulence model implementation with the results provided by NASA at: `http://turbmodels.larc.nasa.gov`.

The flow conditions are: free stream Mach number $M_\infty = 0.2$, angle of attack $\alpha = 16°$, and Reynolds number $Re = 9 \times 10^6$ based on the reference chord of $c_{\text{ref}} = 0.5588\ m$.

3 Geometry and Meshes

The participants are welcome to generate their own meshes, however, **we do encourage the use of the geometry file (airfoil and farfield boundary) provided in the link below as this will facilitate the comparison of all the results**.

- `https://www.dropbox.com/s/q2yh0xiin95v7n2/c31_geo.igs?dl=0`

We are also providing a quartic mesh in two versions, one composed by quadrilaterals and one by triangles. They can be downloaded in GMSH format from the following links:

- `https://www.dropbox.com/s/fp34q225lnhx6g7/c31_mesh_quad_q4.msh?dl=0`
- `https://www.dropbox.com/s/l1yq56woj7guv09/c31_mesh_tri_q4.msh?dl=0`

4 Boundary Conditions

Adiabatic no-slip wall on the airfoil surface, free-stream (full state + Riemann solver) at the farfield. Boundary conditions may be strongly or weakly enforced, please report the details in your writeup.

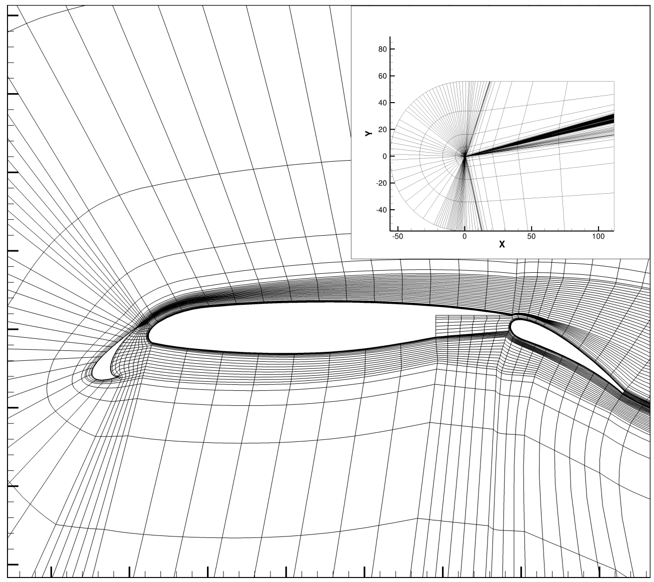

Figure 1: Provided mesh in the quadrilateral version (4070 elements).

5 Requirements

1. Perform a convergence study of drag and lift coefficients (based on c_{ref}), c_l and c_d, using one or more of the following three techniques:
   - Uniform mesh refinement of the coarsest mesh.
   - Quasi-uniform refinement of the coarsest mesh, in which the meshes are not necessarily nested but in which the relative grid density throughout the domain is constant.
   - Adaptive refinement using an error indicator (e.g. output-based).
2. Record the degrees of freedom and work units for each data point, where the initial CPU time ($t = 0$) corresponds to initialization with free-stream conditions on the coarsest mesh. If using parallel processing, please report the number of processors used and the TauBench value for the number of CPUs used in addition to the single CPU TauBench value.
3. Submit your c_d and c_l data versus DOF and work units using the following TecPlot form (use multiple zones for more than one data set):
   `https://www.dropbox.com/s/wypg7ko1ubl9keb/CL_CD_work_form.dat?dl=0`.
4. Submit your C_p and C_f values versus x using the TecPlot form below. **Sign convention:** use the sign of the $x-$component of the shear stress as the sign for C_f.
   `https://www.dropbox.com/s/4yqnhnx5hhehcar/Cp_Cf_form.dat?dl=0`

6 Resources

Reference results and slides can be found on the link below. Keep checking for updates.
`https://www.dropbox.com/sh/jpa3t1t7c7gtwbk/AADsEr3R9Q95ra2qwRwO4NnFa?dl=0`