6 CFD for Steady and Unsteady Flows

Tutorial: PEs hands-on setup for Edge

Exercises and Projects – software here: RANS eg SU2

Review questions

1. Explain how the 3D finite volume solver can work by considering only 1D flows a) between cells, b) at free-stream boundaries. Consult also the sections on boundary conditions in Ch 4.

2. Explain the differences between cell-center and vertex-center based solver schemes.

Airfoil computations

3. Fig. 6.9 – 6.12 shows shocks and boundary layers on the RAE104 airfoil from *M* 0.7825 to 0.9125 at α 0.91 deg. computed by RANS. A few of the cases are slow to converge and indicate low flow stability with incipient separation and unsteadiness. Repeat the exercise at α 1.01 deg and 0.81 deg. Do all cases converge well? Transonic flow is known to be sensitive.

   The data – if the cases all converge - now allows estimates of $c_{l,\alpha}$ and $c_{m,\alpha}$. Use the data to plot the travel with *M* of AC and CP. Pitch-up or Tuck-under?

4. At *M* = 1.5 an airfoil with a rounded nose gives a blunt body flow around the nose. The shock standoff distance is related to *M* and the nose radius. Make grids for naca00xx foils, xx = 12, 10, 8 and 6 with high resolution in the nose regions and run SU2 in Euler mode to illustrate how the subsonic region varies with nose radius. How does the nose radius scale with thickness ratio?

5. Run the RAE100 airfoil through the "capricious" transonic speed regime, and compare to the results in Figure 6.6 for the RAE104.