

HOMWORK #10

AE 308

COMPRESSIBLE AERODYNAMICS

SPRING 2018

Evaluate the performance of the NACA 2412 2-D airfoil section over a range of Mach numbers and a fixed angle-of-attack, using Computational Fluid Dynamics (CFD). Students may work alone or in team of two (turn in one HW with both student's names).

1. Using Pointwise grid generator on a campus workstation, produce an appropriate O-grid using the provided lower and upper "database" curves, rotated for $\alpha = 5^\circ$. All steps shown in class. Use 0.01 m (1 cm) resolution along curves with 0.001 (1 mm) resolution at both LE and TE. For the 2-D surface grid (or "domain") extrude using an initial normal-to-wall spacing of 1.0E-04 m (0.1 mm) and growth rate of 1.0 for 20 steps, followed by another 30 steps at a growth rate of 1.25. Choose appropriate boundary condition types within Pointwise (should be no "undefined" boundaries). Must check orientation to ensure surface normal is in +Z direction. Select domain and "Export CAE" to create the Fluent .cas file. (4 pts)

Export or screen capture two images: i) overall domain, ii) close-up of LE, iii) close-up of TE.

2. Read .cas file into Ansys Fluent to solve using "density-based" and "steady" methods. Set Energy equation ON, Viscous model to INVISCID, change air model to include ideal gas law for density variation, and use AUSM solution method and Courant # of 0.5 for robustness. Using monitors for drag and lift monitor the progress of these values until converged, or pseudo-steady (i.e., oscillating in repeatable manner). Set freestream Mach number according to each case given below. Freestream temperature may be left at default of 300 K. Note that case #4 should be run with "pressure-based" method for faster convergence. For each new Mach number, remember to change boundary condition, re-initialize the solution, and update the reference conditions, all before starting to run solution. (4 pts)

Case	Freestream Mach
#1	3.0
#2	1.0
#3	0.8
#4	0.3

Produce table of C_d and C_l and L/D versus Mach number and make corresponding plot for all three values vs Mach on the same plot.

3. Write out .cas (grid + boundary conditions + settings) and .dat (solution) after each case is converged. Read each pair of .cas and .dat files together into Tecplot. (2 pts)

Produce a pressure contour + streamlines plot of flowfield around airfoil for each freestream Mach # case (as shown in class).

HINTS

POINTWISE:

Should complete first part of first tutorial in Tutorial Workbook (accessed via Pointwise) to get practice with using Pointwise to make O-grid, BEFORE class demonstration of Pointwise. Tutorial grid is similar to grid required for this HW.

FLUENT:

Most workstations in labs have 4 cores....can use parallel processing option and select 3 cores for faster results. Should not take more than 1000 iterations per run.

When setting "Monitors" for drag and lift, check boxes for plotting and screen output so that you can monitor these coefficients as they evolve during the computation.