AOE 6145 CFD
Spring 2025
Instructor: Dr. Chris Roy
Project: (Officially) Due Wednesday 5/7/2025 at 11pm
Note: You have until Monday 5/12/2025 1:00pm to turn it in without penalty

Starting with your quasi-1D nozzle Euler equation code from HW#4, write a 2D finite volume code to solve the Euler equations on curvilinear structured meshes. Implement both the van Leer and the Roe upwind schemes. Use MUSCL extrapolation and flux limiters to obtain second-order accuracy. Use a Runge-Kutta explicit time integration scheme (either 2 or 4 stages) to march the solutions to steady state. Monitor L_2 norms of the steady-state iterative residuals to ensure sufficient iterative convergence levels are reached. Use the practical method we discussed in class to relate the iterative residuals to the iterative errors in the global quantities of interest (i.e., discretization error norms for MMS, total pressure loss for the inlet, and lift and drag coefficients for the airfoil).

Perform a code verification study using the Method of Manufactured Solutions (MMS). Examine both supersonic and subsonic cases following Roy et al (2002 and 2004) which are posted on Canvas. Fortran functions for computing the exact solution and the source terms for these two cases are available on Canvas. Mathematica notebooks are also available if you want to output the exact solutions and source terms in another programming language. Compute the order of accuracy using the norms (L_1 and L_2) of the solution variables (i.e., the ones you solve for in the code) at the cell centers. Note: you may need to turn off the flux limiters to achieve the formal order of accuracy of the upwind methods. Systematically-refined curvilinear grids are available on Canvas. For solution plotting purposes, you can use interpolation to get your cell-centered solution to the grid nodes or plot them as cell-averaged data in Tecplot. Note: for debugging, you may want to initially run your code on Cartesian grids where it is very easy to establish the correct total fluxes at each *x*- and *y*-face.

Compute two additional cases with your code. The first case is the supersonic flow through a simplified 2D 30 degree inlet. The inflow conditions are given below and the <u>freestream flow is in the *x*-direction</u>.

Mach	4.0
pressure	12,270 Pa
temperature	217 K

Multiple systematically-refined grids are available on Canvas for this case. Evaluate the averaged total pressure at the outlet to predict the total pressure losses.

The second case is the flow over a NACA64A series airfoil with a thickness-to-chord ratio of 6%. You will need to modify your code to handle the periodic boundary condition across the trailing edge cut. The freestream conditions for two angles of attack are:

Mach	0.84	0.75
static pressure	65,855.8 Pa	67,243.5 Pa
static temperature	300 K	300 K
angle of attack, α	0 deg.	8 deg.

Multiple grids are also available for this case. Compare your CFD predictions for the pressure coefficient to the experimental data available on Canvas for the zero angle of attack case: use the freestream static pressure for the reference pressure. Compare the lift coefficient (integrated force on the airfoil normal to the freestream velocity vector) and drag coefficient (integrated force on the airfoil tangent to the freestream velocity vector) for the first case to the experimental results. You can run the angle of attack case by simply changing the direction of the freestream velocity vector: $u_\infty = V_\infty \cos(\alpha)$ and $v_\infty = V_\infty \sin(\alpha)$.

Do a formal write-up on this project as you would for a conference paper. I recommend the AIAA Paper format (given on the Canvas site), but any reasonable formal report format is acceptable. Some useful definitions are given below.

Total pressure: $p_0 = p\left[1+\frac{\gamma-1}{2}M^2\right]^{\left(\frac{\gamma}{\gamma-1}\right)}$

Pressure coefficient: $c_p = \frac{p-p_\infty}{\frac{1}{2}\rho_\infty V_\infty^2}$

Drag coefficient: $c_d = \frac{D'}{\frac{1}{2}\rho_\infty V_\infty^2 c}$

Lift coefficient: $c_l = \frac{L'}{\frac{1}{2}\rho_\infty V_\infty^2 c}$

In the above definitions, c is the airfoil chord, D' is the drag force per unit span (width), and L' is the lift force per unit span (width).

Note: In order to get cell-averaged quantities to a boundary (airfoil surface or ramp outflow plane), you will need to perform an extrapolation (e.g., you could use a MUSCL extrapolation or something more sophisticated). In order to integrate the forces over all faces on a boundary, you will need to perform some kind of numerical integration (maybe midpoint rule if you have the values at the face center?). The functional of interest for the inlet case is the total pressure loss, which can be computed as:

$$Loss = \frac{1}{H}\int_{y\min}^{y\max} \left(p_{0,\infty} - p_{0,exit}\right) dy$$