Assignment 1

Subsonic flow over a thin symmetric airfoil

We are going to study 2D steady compressible (irrotational and isentropic) flow around a thin symmetric airfoil. We are going to use the small disturbance potential equation and also the transonic small disturbance potential equation, and compare the results of these two equations. Recall that for small Mach numbers the results should be roughly the same, but as the Mach number increases the small disturbance potential equation is not a good approximation any more. The first of these equations reads

$$(1 - M_\infty^2)\phi_{xx} + \phi_{yy} = 0, \tag{1}$$

where M_∞ is the free stream Mach number and ϕ represents the small disturbance velocity potential so that the streamwise velocity fluctuation is $u' = \phi_x$, the vertical velocity fluctuation is $v' = \phi_y$ and the streamwise velocity is $u = U_\infty + u'$, where U_∞ is the free stream velocity. The transonic disturbance equation is

$$\left(1 - M_\infty^2 - (\gamma + 1)M_\infty^2 \frac{\phi_x}{U_\infty}\right)\phi_{xx} + \phi_{yy} = 0, \tag{2}$$

where γ is the ratio of specific heats.

Consider a symmetric, airfoil of chord $c = 1$ at zero angle of attack; the upper surface of the airfoil is given by

$$y_p(x) = \varepsilon x(1 - x/c), \quad 0 \le x \le c, \tag{3}$$

where $\varepsilon = 0.18$ is a constant. Since the solution is going to be symmetric we can ignore the lower part of the computational domain and solve only for the upper part. In addition, since the airfoil is thin, we can use thin airfoil theory and impose the boundary condition at $y = 0$ instead of at the airfoil surface. Then we do not need to work with a body fitted mesh. A simple mesh aligned with the Cartesian coordinate system will suffice. The computational domain spans, in the vertical direction, from the chord line at $y = 0$ to the top boundary located at $y = L_y = 25c$. The left and right far boundaries are placed 25 chord lengths away from the leading and trailing edge respectively. Along the lower boundary the boundary condition is

$$\begin{aligned} \phi_y &= U_\infty \frac{\mathrm{d}y_p}{\mathrm{d}x} \quad 0 \le x \le c, \\ \phi_y &= 0 \quad \text{otherwise,} \end{aligned}$$

where $y_p(x)$ is the airfoil profile. Along the far field boundaries ϕ is held constant. Since ϕ is the disturbance potential and not the full potential we can use $\phi = 1$ or equal to any other constant at these boundaries.

Since the domain is very large we need to use a non-uniform grid. Let's use 30 mesh points equally spaced to span the chord line $0 \le x \le c$, and then stretch the mesh (as explained below for the vertical direction) from the airfoil leading edge, starting with the same grid spacing as in the uniform part, to the left far field boundary and also from the trailing edge to the downstream right far field boundary, using additional 40 grid points in each direction. Concerning the vertical direction, we are going to use a exponential mesh stretching starting at $y_1 = 0$, ending at $y = L_y$ and using $N_y = 61$ grid points, we also want the mesh point to start with $\Delta y_{min} = \varepsilon/10$, for that we need to determine the stretching factor κ. For the j grid point the exponential law gives

$$y_j = y_1 + L_y \frac{\exp\left(\kappa \dfrac{j-1}{N_y - 1}\right) - 1}{\exp(\kappa) - 1}. \tag{4}$$

The parameter κ is determined by Newton's method for finding the value for which $f(\kappa) = \Delta y_{min} - (y_2 - y_1) = 0$, where y_2 is obtained by substituting $j = 2$ in equation (4). Note that, since the grid is stretched, the formulas you employ for the first and second derivative need to take this fact into account. Please use central differences for all derivatives.

Assume we normalize pressure and density so that $U_\infty = 1$, $\rho_\infty = 1$, and $\gamma = 1.4$. Then, since the free stream sound speed is $a_\infty = U_\infty/M_\infty$, then the free-stream pressure is

$$p_\infty = \frac{1}{\gamma}\rho_\infty a_\infty^2 = \frac{1}{\gamma M_\infty^2}.$$

In order to solve equation (1) the same techniques we employed in class to solve the Laplace equation can be used. Equation (2) is more complicated since it is non-linear. To solve this equation you should employ an iterative procedure using line relaxation. You should start the iteration process with $\phi = 1$ everywhere or with the solution obtained for equation (1). This is the solution at iteration n and we want to obtain the next iteration $n+1$. Assume we already solved for the first $i-1$ lines and we are solving now line i, simultaneously. For this we need to discretize the equation (2). The derivative ϕ_x we discretize using the values of ϕ from the previous iteration n, so we know them. The derivative ϕ_{yy} we discretize using the values at the current iteration $n+1$ since we are going to solve for all of them simultaneously. The derivative ϕ_{xx} we discretize it using for $i-1$ the values at the current iteration $n+1$ that we already computed, for i also the value at the current iteration $n+1$ that we are going to obtain now and for $i+1$ the values from the previous iteration n since we do not know the new ones yet. After doing this we obtain a tri-diagonal system for the unknown values $\phi(i,1)^{n+1}, \phi(i,2)^{n+1}, \ldots, \phi(i,N_y)^{n+1}$. Once we solve this system we can proceed to the next line $i+1$. The line solution is swept through the flow field starting at $i=2$. The sweep through the mesh, from left to right, is then repeated until the solution converges the absolute value of the residual to an acceptable tolerance.

- For $M_\infty = 0.65$, obtain a numerical solution of equations (1) and (2) using the mesh defined above.
- Evaluate the pressure coefficient, $c_p(x)$ on the surface of the airfoil (in the interval $-0.5c \le x \le 1.5c$) and the value of the Mach number in the region near the airfoil $M(x,y)$.
- Evaluate how $c_p(x)$ and $M(x,y)$ change with increasing M_∞ and in particular what are the differences using equations (1) and (2).
- For equation (2), what is the maximum M_∞ for which your solution converges? Do you have an idea why this is the case?
- Solve also the equations varying the parameter ε in the range $\varepsilon \in [0.1, 0.3]$ at a fixed M_∞ of your choice (but not too small) and show the changes in $c_p(x)$.
- **Deliverables:** the source code and a short report summarizing your results (maximum 8 pages including figures). In the report, in addition to the presentation of your results, you should explain the decisions you have taken like for example: what discretization you employed, how did you impose the boundary conditions, etc. It is important that the report is concise, not being just a copy of the theory. You should provide key explanations, key results, and only those plots that support your narrative. You should not include plots just to fill pages. I find the following approach useful: You should imagine that you are working in a company, and your boss who is a busy person, needs to understand in a few pages, what you did, how you did it, what results you got and what we learnt from those results. If you want to get feedback on your report, you should deliver a printed copy of the report in my pigeonhole (Benet building). If you only submit a pdf file, you will only get a grade.