Homework 4

Classification and Shock Tube

due February 16, 2015

Task 1 : Classification of partial differential equations

The non-dimensionalised set of equations describing a two-dimensional, stationary, frictionless, incompressible flow (incompressible Euler equations) is

$$\begin{aligned} \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} &= 0\,, \\ u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + \frac{\partial p}{\partial x} &= 0\,, \\ u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + \frac{\partial p}{\partial y} &= 0\,, \end{aligned} \tag{1}$$

where u and v are the velocity components in the x and y directions and p is the non-dimensional pressure. We can define the state vector as

$$\underline{u} = \begin{bmatrix} u \\ v \\ p \end{bmatrix}.$$

a) Write the equations (1) in the so-called *quasi-linear* matrix form

$$\underline{\underline{A}}\frac{\partial \underline{u}}{\partial x} + \underline{\underline{B}}\frac{\partial \underline{u}}{\partial y} = 0\,.$$

b) Classify the system of equations (1) by using the characteristic equation $\det\left(\underline{\underline{B}} - \lambda\underline{\underline{A}}\right) = 0$ (where $\underline{\underline{A}}$ and $\underline{\underline{B}}$ are the matrices obtained in part a)). What is the expected behaviour of the solution?

Task 2 : Shock tube

In this task we consider the flow inside a shock tube. A shock tube is a pipe, closed at both ends, with a diaphragm in the middle separating a region with high-pressure gas and a region with low-pressure gas. The initial condition for the density, ρ, is given by (see also figure 1)

$$\rho(x,0) = \begin{cases} \rho_0, & \text{where} \quad x \le L/2\,, \\ \rho_1, & \text{where} \quad x > L/2\,, \end{cases}$$

and the fluid is at rest, *i.e.* $u(x,0) = 0$. L is the length of the tube.
The motion of a barotropic gas (pressure is only a function of the density) in the shock tube can be described by the 1D Euler equations

$$\begin{aligned} \rho_t + (\rho u)_x &= 0\,, \\ (\rho u)_t + (\rho u^2 + p)_x &= 0\,, \\ p &= K\rho^\gamma\,, \end{aligned} \tag{2}$$

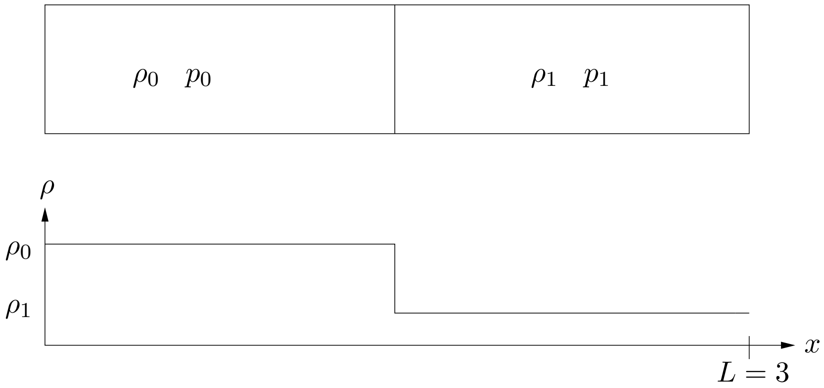

Figure 1: Initial condition.

where K is a constant determined by the initial conditions and the isentropic expansion factor is $\gamma = 1.4$. The system of equations (2) can be written in conservative form as

$$U_t + F(U)_x = 0\,, \tag{3}$$

where $U = (\rho, \rho u)$, and $F(U) = (\rho u, \rho u^2 + p)$ is the flux vector. This equation can be solved numerically using the MacCormack scheme:

$$U_j^* = U_j^n - \lambda[F(U_{j+1}^n) - F(U_j^n)] \qquad \text{Predictor step,} \tag{4}$$
$$U_j^{n+1} = \frac{1}{2}(U_j^n + U_j^*) - \frac{\lambda}{2}[F(U_j^*) - F(U_{j-1}^*)] \qquad \text{Corrector step,}$$

where $\lambda = \Delta t/\Delta x$.
Usually, the numerical solution of these equations will show unphysical oscillations (so-called *wiggles*). In order to damp these oscillations, an artificial-viscosity term can be added to the right-hand side of the equations. System (3) is thus modified as follows:

$$U_t + F(U)_x = (\nu_{\text{num}} U_x)_x\,, \tag{5}$$

where $(\nu_{\text{num}} U_x)_x$ is the artificial-viscosity term. The artificial viscosity ν_{num} should be a small value of the order of the grid spacing Δx. The system (5) can be written in conservative form as

$$U_t + [\underbrace{F(U) - \nu_{\text{num}} U_x}_{\tilde{F}}]_x = 0\,, \tag{6}$$

in which we use the modified flux function $\tilde{F}$.
The term U_x in the modified flux function has to be evaluated numerically. When computing U^* in the predictor step, U_x should be approximated by a backward difference; when computing U^{n+1} in the corrector step, U_x should be approximated by a forward difference.
Since we want to add numerical viscosity in regions with high gradients, we use a density switch model in which the density is used to localise the shock:

$$\nu_{\text{num}} = \Delta x V_{\text{s}}(C_2 sw(\rho) + C_0)\,,$$

where the multiplication by $\Delta x V_{\text{s}}$ is needed in order to obtain the correct physical unit for the viscous term. Δx is given by the mesh size and the parameter V_{s} is related to the convection speed (characteristic speed) of the solution and is chosen as

$$V_{\text{s}} = \max(|u + c|, |u - c|)\,,$$

where c is the speed of sound.
The density switch $sw(\rho)$ is computed as follows

$$sw(\rho) = \left|\frac{\partial^2 \rho}{\partial x^2}\right| \frac{\Delta x^2}{\bar{\rho}} \approx 2\frac{|\rho_{j-1} - 2\rho_j + \rho_{j+1}|}{|\rho_{j-1} + 2\rho_j + \rho_{j+1}|}\,.$$

The second-order derivative of ρ is approximated by a second-order central difference and $\bar{\rho}$ is the mean value of ρ computed at the j-th grid point as $\bar{\rho}_j = (\rho_{j+1} + 2\rho_j + \rho_{j-1})/4$. Where large gradients are present in the density field $sw(\rho)$ will be of order one, and, on the contrary, where the density field is smooth $sw(\rho)$ will be approximately zero.
C_2 is a parameter and should be chosen in order to obtain sufficient viscosity to damp the oscillations. C_0 is a "background" diffusion parameter and should also be chosen. Both C_0 and C_2 should be of order one or less. The optimal values of C_0 and C_2 are usually determined after some experimentation with different values, see task c) below.

Your task is to complete a MATLAB code which solves equation (6) using the MacCormack formulation (4).

The following files can be downloaded from the course home page:

`shocktube.m`

`artificial_visc.m`

`dx.m`

`mac_cormack.m`

`boundary_cond.m`

`flux_function.m`

1. The main program `shocktube.m` needs to be completed with the appropriate calculation of the time step. Additionally if any pre/postproccesing operations are needed they should be added here. Note the use of global variables.

2. The file `flux_function.m` defines the flux function. Here the flux function for the system of equations (3) must be coded. Also, two lines in `mac_cormack.m` need to be completed.

3. `boundary_cond.m` sets the boundary conditions, *i.e.* $(\rho)_1$, $(\rho)_n$, $(\rho u)_1$ and $(\rho u)_n$, (n is the number of grid points).
   To set the boundary conditions we use the physical condition that the tube is closed at both ends, so homogeneous Dirichlet conditions $u(0,t) = u(L,t) = 0$ are imposed. The other condition corresponds to a numerical boundary condition imposed by setting the value of ρ at the boundaries as a zeroth order extrapolation from the value of the density inside the tube. This is a simplification of the concept of Riemann invariants,

$$\rho_1 = \rho_2\,, \qquad \rho_n = \rho_{n-1}\,.$$

4. Finally, the artificial-viscosity model is implemented in `artificial_visc.m`. In this file you need to add the definition of the speed of sound and V_s.

The length of the tube is set to $L = 3$. Note that the pressure p needs to be updated at every time step with the equation $p = K\rho^\gamma$.

The following points should be addressed in your report (including the completed MATLAB codes and the plots of your results):

a) To better understand this scheme, write the MacCormack scheme for the advection equation

$$u_t + f_x = 0\,, \qquad f = au\,.$$

You will notice that it coincides with another scheme. What is the name of this scheme (see lectures)?

b) The CFL stability condition is guaranteed by choosing $\Delta t = C_N \Delta x / u_{max}$ with the Courant number $C_N < 1$. Here, u_{max} is the maximum absolute value of the characteristic speeds. From the quasi-linear form of the equations,

$$\begin{pmatrix}\rho \\ u\end{pmatrix}_t + \begin{pmatrix} u & \rho \\ K\gamma\rho^{\gamma-2} & u\end{pmatrix}\begin{pmatrix}\rho \\ u\end{pmatrix}_x = 0\,,$$

show that the characteristic speeds are $u \pm c$ with $c^2 = K\gamma\rho^{\gamma-1} = \gamma p/\rho$ where c is the speed of sound. Use this to set the time step in the main program `shocktube.m`.

c) Having completed the MATLAB files, run the code. The initial jump breaks up into a rarefaction wave moving left and a shock moving right. Measure the shock speed s (this can be done by considering the shock location at different times). Check its correctness by computing the analytical shock speed s from the Rankine-Hugoniot condition

$$s(\rho_L - \rho_R) = \rho_L u_L - \rho_R u_R\,,$$

where ρ_L and u_L are the computed states at the left-hand side of the shock, ρ_R and u_R are measured on the right-hand side of the shock.

d) Run the code up to final time $t_f \approx 0.003$ (round it with the number of time steps) with different Courant numbers C_N and for the values of the artificial viscosity parameters C_2 and C_0 given below and discuss the solution. For the last case (optimal parameters) run the code up to $t_f \approx 0.01$, after the shock is reflected.

C_0	C_2
0.05	0.45
0.05	0.05
0.4	0.05
0.05	0.25