Homework 5

Classification and Shock Tube

due February 25, 2016

Task 1 : Classification of partial differential equations

A simplified form of the one dimensional shallow water equations is considered. These partial differential equations are expressed in terms of velocity at the water column $u(x,t)$ and potential energy $\phi(x,t)$:

$$\begin{cases} u_t + \left(\frac{u^2}{2} + \phi\right)_x &= 0 \\ \phi_t + (u\phi)_x &= 0 \end{cases} \qquad (x,t) \in \mathbb{R} \times [0,\infty). \tag{1}$$

The shallow water system can be written in conservative form as follows

$$U_t + F(U)_x = 0, \tag{2}$$

with the state variables and fluxes

$$U := \begin{bmatrix} u \\ \phi \end{bmatrix}, \quad F(U) := \begin{bmatrix} u^2/2 + \phi \\ u\phi \end{bmatrix}. \tag{3}$$

a) Rewrite Eq. (**??**) in the so called quasi-linear matrix form

$$U_t + AU_x = 0, \qquad A(u,\phi) \in \mathbb{R}^{2\times 2}. \tag{4}$$

b) Calculate the eigenvalues of A and classify the type of the PDE system.

c) Now, let

$$A = \begin{bmatrix} 1 & 1 \\ 4 & 1 \end{bmatrix}. \tag{5}$$

This matrix has a complete set of eigenvalues and eignvectors. Diagonalize the matrix and find a characteristic form

$$W_t + \Lambda W_x = 0, \tag{6}$$

where W is a state vector including the characteristic variables and Λ a diagonal matrix containing the eigenvalues of the system. This equation has an exact solution. Find the solution analytically using the intial conditions

$$u = u_0, \quad \phi = \phi_0, \quad (x,t) \in \mathbb{R} \times \{0\}. \tag{7}$$

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 **??**)

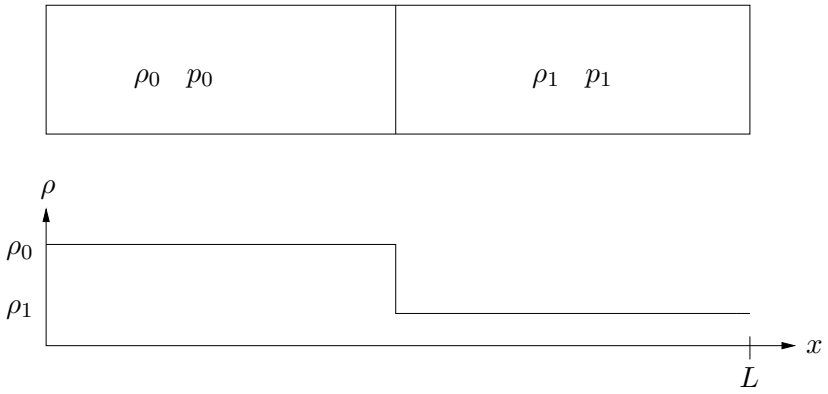

Figure 1: Initial condition.

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

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

$$\begin{aligned} \rho_t + (\rho u)_x &= 0\,, \\ (\rho u)_t + (\rho u^2 + p)_x &= 0\,, \\ p &= K\rho^\gamma\,, \end{aligned} \tag{8}$$

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

$$U_t + F(U)_x = 0\,, \tag{9}$$

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:

$$\begin{aligned} U_j^* &= U_j^n - \lambda[F(U_{j+1}^n) - F(U_j^n)] \qquad \text{Predictor step,} \\ U_j^{n+1} &= \frac{1}{2}(U_j^n + U_j^*) - \frac{\lambda}{2}[F(U_j^*) - F(U_{j-1}^*)] \qquad \text{Corrector step,} \end{aligned} \tag{10}$$

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 (**??**) is thus modified as follows:

$$U_t + \tilde{F}(U)_x = 0, \quad \tilde{F}(U)_x := F(U)_x - (\hat{\nu} U_x)_x\,, \tag{11}$$

where $(\hat{\nu} U_x)_x$ is the artificial-viscosity term, and where $\tilde{F}$ is called the "modified flux function". The artificial viscosity $\hat{\nu}$ should be a small value of the order of the grid spacing Δx. 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:

$$\hat{\nu} = \Delta x V_\mathrm{s}[C_2 sw(\rho) + C_0]. \tag{12}$$

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

$$V_{\rm s} = \max_x \{|u+c|, |u-c|\}\,,$$

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

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

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. The parameter C_2 should be chosen in order to obtain sufficient viscosity to damp the oscillations, and 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 (**??**) using the MacCormack formulation (**??**).

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 (**??**) 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,

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

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 $\rho_{_L}$ and $u_{_L}$ are the computed states at the left-hand side of the shock, and $\rho_{_R}$ and $u_{_R}$ are the values on the right-hand side of the shock and thus given by the initial conditions.

d) Run the code up to final time $t_\mathrm{f} \approx 0.003$ (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 $t_\mathrm{f} \approx 0.01$, 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