Homework 4

Classification and Shock Tube

due 17/02-2014

Task 1 : Classification of partial differential equations

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

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

Here, u and v are the velocities in the x and y directions, respectively; p is the non-dimensional pressure. Thus, we define the state vector as

$$\underline{u} = \begin{bmatrix} u \\ v \\ p \end{bmatrix} . \tag{2}$$

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

$$\underline{\underline{A}}\frac{\partial \underline{u}}{\partial x} + \underline{\underline{B}}\frac{\partial \underline{u}}{\partial y} = 0\,. \tag{3}$$

b) Determine the type of the system of partial differential equations (1) by using the characteristic equation $\det\left(\underline{\underline{B}} - \lambda\underline{\underline{A}}\right) = 0$ based on $\underline{\underline{A}}$ and $\underline{\underline{B}}$ obtained in part a). What is the expected behaviour of the solution based on the type?

Task 2 : Shock tube

In this task we consider the flow inside a shock tube. A shock tube is a tube, closed at both ends, with a diaphragm separating a region with high-pressure gas and a region with low-pressure gas.

The initial condition for the density ρ is (see figure 1)

$$\rho(x,0) = \begin{cases} \rho_0 & \text{if} \quad x \le L/2 \\ \rho_1 & \text{if} \quad x > L/2\ , \end{cases} \tag{4}$$

and the velocity is $u(x,0) = 0$, i.e. the fluid is at rest. Here, L is the length of the tube.

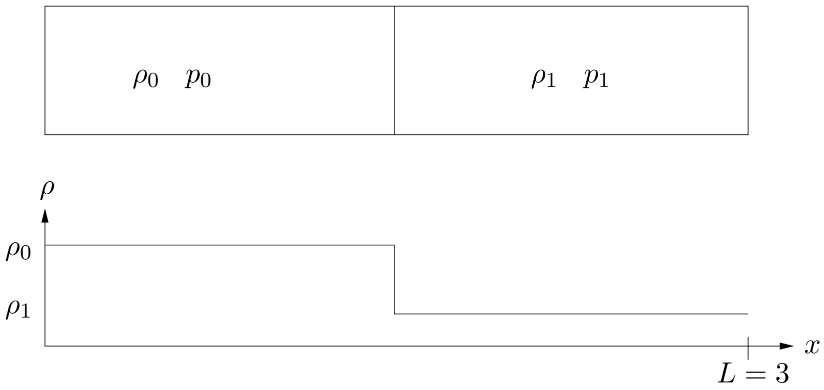

Figure 1: Initial conditions

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

$$\begin{aligned} \rho_t + (\rho u)_x &= 0 \ , \\ (\rho u)_t + (\rho u^2 + p)_x &= 0 \ , \\ p &= K\rho^\gamma \ , \end{aligned} \tag{5}$$

where K is a constant determined by the initial conditions and the isentropic expansion factor $\gamma = 1.4$. The equations above can be written in compact form as

$$U_t + F(U)_x = 0 \ , \tag{6}$$

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

$$\begin{aligned} U_j^* &= U_j^n - \lambda[F(U_{j+1}^n) - F(U_j^n)] && \text{Predictor step} \\ U_j^{n+1} &= \frac{1}{2}(U_j^n + U_j^*) - \frac{\lambda}{2}[F(U_j^*) - F(U_{j-1}^*)] && \text{Corrector step} \ , \end{aligned} \tag{7}$$

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, we can add an artificial-viscosity term to the right-hand side of the equations. Thus, instead of the system (6), one solves

$$U_t + F(U)_x = (\mu_{num} U_x)_x \ , \tag{8}$$

where $(\mu_{num} U_x)_x$ is the artificial-viscosity term. The artificial viscosity μ_{num} should be a small value of the order of the grid spacing Δx. The full system (8) can be written in conservative form as

$$U_t + [\underbrace{F(U) - \mu_{num} U_x}_{\tilde{F}}]_x = 0 \tag{9}$$

where $\tilde{F}$ is a modified flux function.

The term U_x in the modified flux function has to be evaluated numerically. When U^* is computed by the predictor step, U_x should be approximated by a backward difference and when U^{n+1} is computed by the corrector step, U_x should be approximated by a forward difference. This is necessary for a correct implementation of the artificial viscosity term.

Since we only want to add additional viscosity in the region with high gradients, we use a density switch model in which the density is used to localise the shock,

$$\mu_{num} = \Delta x V_{scal}(C_2 sw(\rho) + C_0)\ , \tag{10}$$

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

$$V_{scal} = \max(|u+c|, |u-c|) \tag{11}$$

where c is the speed of sound. The density switch, $sw(\rho)$ is computed in the following way

$$sw(\rho) = \left|\frac{\partial^2 \rho}{\partial x^2}\right| / \overline{\rho} \tag{12}$$

The second-order derivative of ρ is approximated by a second-order central difference and $\overline{\rho}$ is a mean value of ρ computed in grid point j as $\overline{\rho_j} = \rho_{j+1} + 2\rho_j + \rho_{j-1}$.

When there are large gradients in ρ, $sw(\rho)$ will be of order 1, and when ρ is smooth $sw(\rho)$ will be approximately zero. C_2 is a parameter and should be chosen 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 1 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 the equations (9) using the MacCormack formulation (7).

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`

The files that need to be completed are:

1. The main program `shocktube.m` needs to be completed with the appropriate calculation of the time step. Additionally if any pre/postproccesing computations 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 (6) 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 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,

$$\rho_1 = \rho_2 \qquad \rho_n = \rho_{n-1}\,. \tag{13}$$

4. Finally, the artificial-viscosity model is implemented in `artificial_visc.m`. In That file, add the definition of the speed of sound and V_{scal}.

The length of the tube is set to $L = 3$. Note that the pressure p needs to be updated in every time step based on 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) The CFL stability condition is guaranteed by taking $\Delta t = CN\Delta x/u_{max}$ with the Courant number $CN < 1$. Here, u_{max} is the maximal absolute value of the characteristic speeds. From the quasi-linear form of the equations,

$$\begin{pmatrix} \rho \\ u \end{pmatrix}_t + \begin{pmatrix} u & \rho \\ K\gamma\rho^{\gamma-2} & u \end{pmatrix} \begin{pmatrix} \rho \\ u \end{pmatrix}_x = 0 \tag{14}$$

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`.

b) Run the problem as set up in point a). 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 that it is correct by computing the analytical shock speed s from the jump relation (Rankine-Hugoniot condition)

$$s(\rho_l - \rho_r) = \rho_l u_l - \rho_r u_r \;, \tag{15}$$

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 and thus given by the initial conditions.

c) Run the code with different Courant numbers CN and different values of the artificial viscosity parameters C_2 and C_0 and comment on the solution. Try to find the optimal choice that is no oscillations and a good resolution of the shock. Note that C_0 should be smaller than C_2.