

## Homework 4

## Classification of Differential Equation and Shock Tube

due March 1, 23:59

Guidelines:

- Use only a **single pdf**. Preferably scan your homework, if you take a photo, make sure it is sharp and bright;
  - **Include all plots in the pdf** in the right place;
  - Do not present plots without commenting them: always write a (short) description of what the plot tells you and what you can conclude from it;
  - You can work in groups up to 3;
  - If you work in groups:
    - write the names of all group members at the beginning of the report;
    - you can work and discuss together but each group member is required to submit an individual report written with his/her own words; **copy-paste reports will not be accepted**
  - Please use the following naming convention: “**surname\_hwX.pdf**” and include all Matlab files as a single **separate** archive: “**surname\_hwX.zip**”. X is the number of the homework.
- 

## Task 1: Classification of partial differential equation

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  $\det(\underline{\underline{B}} - \lambda \underline{\underline{A}}) = 0$  (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,  $\rho$ , is given by (see also figure 1)

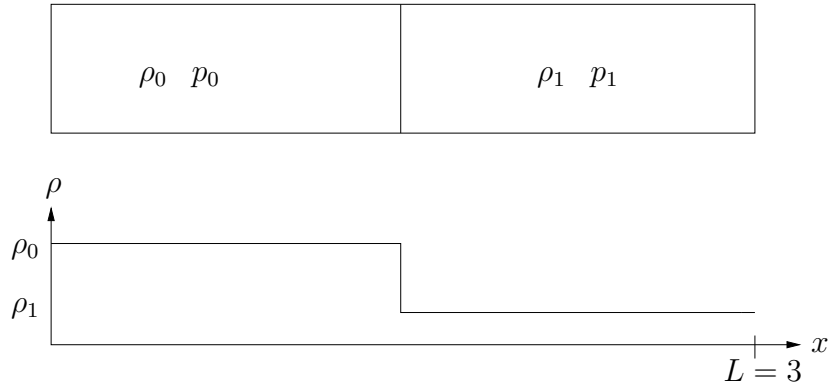


Figure 1: Initial condition.

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

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

The initial jump in figure 1 generates a rarefaction wave moving to the left and a shock wave moving to the right, resulting in both flow regions being separated by a so-called contact discontinuity. The contact discontinuity gives a jump in density while both the velocity and the pressure remain continuous.

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

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

where  $K$  is a constant determined by the initial conditions and the isentropic expansion factor is  $\gamma = 1.4$ . Note that in this simplified case, no contact discontinuity will appear, and only a rarefaction wave and a (weak) shock will be generated. 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:

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

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:

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

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

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

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:

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

where the multiplication by  $\Delta x V_s$  is needed in order to obtain the correct physical unit for the viscous term.  $\Delta x$  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

$$V_s = \max_x (|u + c|, |u - c|),$$

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  $\rho$  is approximated by a second-order central difference and  $\bar{\rho}$  is the mean value of  $\rho$  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 / .py
artificial_visc.m / .py
dx.m / .py
mac_cormack.m / .py
boundary_cond.m / .py
flux_function.m / .py

```

1. The main program `shocktube.m` needs to be completed with the appropriate calculation of the time step. Additionally if any pre/postprocessing 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  $\rho$  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,

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

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

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

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  $\rho_L$  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  $t_f \approx 0.003$  (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	0.4
0.05	0.05
0.45	0.05
0.05	0.25