Master de Physique
Faculté de Physique & Ingénierie

Faculté de physique et ingénierie Université de Strasbourg

Hydrodynamics : Shock Tube and Riemann Solvers

D. Aubert, F Thalmann

Season 2022-23

Abstract

'Shock Tubes' designate numerical experiments designed to test solvers for hydrodyamics and Euler equations. Riemann problems in turn designate the description of the time evolution of a physical system made of two hydrodynamical states (defined by a density, velocity and pressure) initially separated by an interface : these two states will interact and mix, and many shock tubes experiments consists of Riemann problems. More generally, being able to solve a Riemann problem is at the core of many modern techniques in computational fluid dynamics. The aim of this project is to design a hydrodynamics solver, in 1D, and reproduce classic shock tubes and Riemann problems.

1 Numerical techniques for the Euler Equation

Conservation equations and conserved quantites

The dynamics of a 1D fluid is described by the Euler equations. These equations can be described in a so-called conservative form on conserved quantities. Let us first define the *primitive quantities*:

$$\mathbf{W} = (\rho, u, P), \tag{1}$$

where ρ is the fluid density, u its velocity and P its pressure. In this project we will assume a polytropic equation of state with exponent γ. We also define the sound speed $a = \sqrt{\gamma P/\rho}$. Let $\mathbf{U}$ be the array of conserved quantities given by

$$\mathbf{U} = (\rho, \rho u, \frac{1}{2}\rho u^2 + \frac{P}{\gamma - 1}) = (U_0, U_1, U_2), \tag{2}$$

where the last expression stands for the total energy of the gas.

In 1D, the Euler equation can be written as :

$$\frac{\partial \mathbf{U}}{\partial t} + \frac{\partial \mathbf{F}(\mathbf{U})}{\partial x} = 0 \tag{3}$$

where the *flux function* $\mathbf{F}(\mathbf{U}) = (\rho u, \rho u^2 + P, u(E + P))$ is given by

$$\mathbf{F}(\mathbf{U}) = (U_1, \frac{1}{2}(3-\gamma)\frac{U_1^2}{U_0} + (\gamma - 1)U_2, \gamma\frac{U_1}{U_0}U_2 - \frac{1}{2}(\gamma - 1)\frac{U_1^3}{U_0^2}). \tag{4}$$

Numerical Update

An explicit conservative formulation for the discretization of this equation is given by ;

$$\frac{\mathbf{U}_i^{p+1} - \mathbf{U}_i^p}{\Delta t} + \frac{F_{i+1/2}^p - F_{i-1/2}^p}{\Delta x} = 0, \tag{5}$$

where Δx and Δt stand respectively for the cell size and the time step of the numerical solver. This relation provides a mean to update the conservative state in cell i at timestep p, $\mathbf{U}_i^p$ to its next estimate $\mathbf{U}_i^{p+1}$:

$$\mathbf{U}_i^{p+1} = \mathbf{U}_i^p + \frac{\Delta t}{\Delta x}(F_{i-1/2}^p - F_{i+1/2}^p). \tag{6}$$

Here $F_{i+1/2}^p$ designates an *intercell flux*, between cells i and $i+1$: it results from the interaction of the two physical states $\mathbf{U}_i$ and $\mathbf{U}_{i+1}$ via the Euler equations. Different flavors of hydrodynamical solvers will use different estimate of this intercell flux, each with their own pros and cons in terms of simplicity, accuracy or complexity.

2 First simple hydro solver

At this stage, we have pretty much everything to code a hydro solver and run test cases. We simply need to specify the intercell flux, a strategy to set the timestep, fix boundary conditions and choose a set of initial conditions. All these concepts are explained or detailed below : from there Eq. 6 can be used to cycle through the timesteps, to model the time evolution of the physical properties of the fluid.

Lax-Friedrich Flux

Probably the simplest but non-trivial intercell flux is known as the Lax-Friedrich flux :

$$F_{i+1/2} = \frac{1}{2}(F(U_i) + F(U_{i+1})) + \frac{1}{2}\frac{\Delta x}{\Delta t}(U_i - U_{i+1}). \tag{7}$$

It is known to be very diffusive but is very easy to implement and as such is often chosen as a basis for a first implementation.

Time step

The formulation chosen here is only conditionally stable and requires the timestep Δt to obey the *Courant* condition :

$$\Delta t < \frac{\Delta x}{S_(max)}.$$

Here Δx is the spatial resolution of the spatial sampling and $S_{\max}$ stand for fastest signal (also known as 'waves') that propagates from intercell interfaces. It can be estimated using

$$S_{\max} = \mathrm{MAX}(|u| + a), \tag{8}$$

where u and a are respectively the velocity and the sound speed and the maximal value is taken over all the cells of the calculation. Therefore at each timestep, $S_{\max}$ is computed from the current state and

$$\Delta t = C\frac{\Delta x}{S_{\max}}, \tag{9}$$

where typically $C \sim 0.9$.

Boundary Conditions

For shock tube tests, one usually assume *transmissive* boundary conditions, i.e. we will assume $\mathbf{U}_{-1} = \mathbf{U}_0$ and $\mathbf{U}_N = \mathbf{U}_{N-1}$, when needed.

Initial Conditions

The most academic shock tube test assumes this set of initial conditions with $\gamma = 1.4$, also known as *the Sod problem*:

$$\begin{aligned} \mathbf{W}(x < 0.5) &= (1.0, 0.0, 1.0) \quad (10) \\ \mathbf{W}(x >= 0.5) &= (0.125, 0.0, 0.1) \quad (11) \end{aligned}$$

where the computational domain assumes $x \in [0, 1]$.

3 Results and Shock Tube

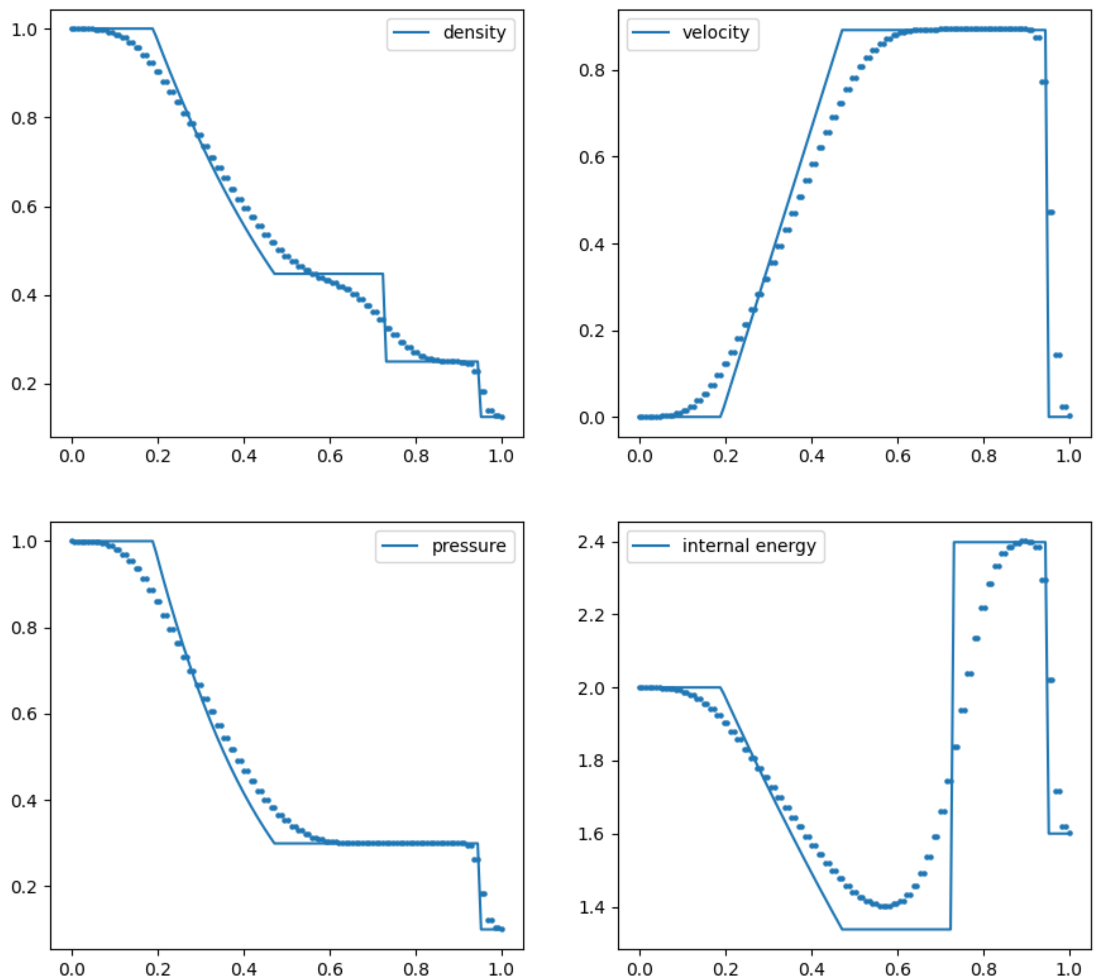

Figure 1: Shock tube using the LF flux at t=0.25, with the spatial profile of the density, velocity, pressure and internal energy. The solid lines stand for the exact solution of the shock tube. Note how the numerical diffusivity can affect the different profiles.

Display the physical state of the primitive quantities at $t = 0.25$ and it should correspond to Fig. 1. From there, try to estimate the impact of spatial or time resolution. Plot also the internal energy profile given by $P/((\gamma - 1)\rho)$.

At this stage you might also want to try other initial conditions such as the *123 problem* :

$$\begin{aligned} \mathbf{W}(x < 0.5) &= (1.0, -2., 0.4) && (12) \\ \mathbf{W}(x >= 0.5) &= (1, 2.0, 0.4) && (13) \end{aligned}$$

the *left Woodward and Collela*

$$\begin{aligned} \mathbf{W}(x < 0.5) &= (1.0, 0, 1000) && (14) \\ \mathbf{W}(x >= 0.5) &= (1, 0, 0.01) && (15) \end{aligned}$$

the *right Woodward and Collela*

$$\begin{aligned} \mathbf{W}(x < 0.5) &= (1.0, 0, 0.01) && (16) \\ \mathbf{W}(x >= 0.5) &= (1, 0, 100) && (17) \end{aligned}$$

4 And after ?

Once you are able to run and validate your first 1D Hydrodynamical solver, several routes can be investigated to expand your project. At this stage you are strongly encouraged to read references such as Toro (2009) and use it to explore the following points

- Other fluxes exists. Simple ones such as the Lax-Wendroff flux are not much more complex than LF to implement. You might want to compare the solutions.
- Fig. 1 display the theoretical profiles of the shock tube. It turns out that the exact solution to these initial conditions can be computed, using *an exact Riemann Solver*. Toro (2009) contains the full procedure to compute it : it's tedious but largely doable. You might want to implement it, to be able to have an absolute reference for your calculations.
- the basic 'serious' flux solver is known as the *Godunov flux*. It uses the exact solution of *all* the Riemann problems of all the interfaces between cells of your model. If you are able to compute the analytic solution of the Shock tube, you are not far from being able to replace your intercell flux with the Godunov flux : if you manage to do it this would be a great improvement of your code. Again, compare your new results with the original one.

References

Toro E., 2009, Riemann Solvers and Numerical Methods for Fluid Dynamics: A Practical Introduction. Springer Berlin Heidelberg, https://books.google.fr/books?id=SqEjX0um8o0C