Course: Mech-516 Computational Gasdynamics

Mini-project 3.

Topics: Second-order TVD schemes.
Released: February 24, 2022
Due: April 12, 2022, 5.30pm

Task 1. Numerical experiments with a TVD scheme for the Euler equations.

Incorporate a second-order in space and time (on smooth solutions) TVD scheme into the Euler code you have developed for the mini-project 2. You may choose any such scheme you prefer. If you do not have any particular preferences I would recommend you to take the MUSCL-Hancock scheme, simply because it is one of the simplest ones and it would be easy to generalize your already existing code for this scheme.

Problem to be simulated is the same as in the mini-project 2. In the low pressure chamber we have atmospheric conditions $\rho_1 = 1$, $p_1 = 1$ (the values are nondimensionalized) while in the high pressure chamber the gas is pressurized and heated so that $\rho_2 = 1$ and $p_2 = 2$. Initially the gas is at rest in both chambers. Then the diaphragm separating the chambers is removed and a shock tube flow evolves in time. We are interested in the solution at time moment $t = 25$. The gas has the specific heat ratio $\gamma = 1.4$. The diaphragm is located at $x = 0.0$ and the computational domain contains 100 cells ($\Delta x = 1$). The computations should be carried out with a constant time step $\Delta t = 0.5$ (till $t = 25$). At the computational domain boundaries, situated in this case in quiescent gas, at all times you may calculate the fluxes using the values from the control volume adjacent to the boundary. The slopes in the boundary control volumes can be assumed to be zero.

Exercise 1-1. Comparison of solutions obtained with and without a slope limiter.

Compute the above problem using the TVD scheme of your choice without a limiter (in this case determine the slope for a control volume, for instance, as an arithmetic average of the adjacent slopes, i.e. $\Delta W_i = 0.5(\Delta W_{i+1/2} + \Delta W_{i-1/2})$ and with a limiter of your choice. Also compute the solution using zero slopes (i.e., with the first-order version of the scheme). Plot the solutions together with the exact solution and comment on the differences you observe.

Exercise 1-2. Study of different limiters.

Calculate the problems using the following limiters: (a) MINMOD limiter; (b) SUPERBEE limiter; (c) one limiter of your choice possessing intermediate compressive (anti-dissipative) properties, such as van Leer limiter (smooth or non-smooth), van Albada limiter, β-limiter etc.

Plot the solutions together with the exact solution and comment on the results for different limiters. In addition to the qualitative observation of plots, use L_1 or L_2 norm to quantitatively evaluate the performance of limiters.

For the above exercises, the plots should contain the exact solution (solid line) and numerical solutions (different symbols without connecting lines). Density, velocity and pressure profiles should be plotted (separately!).

Important warning: in some cases (for instance, Exercise 1-1, computation without a limiter, or Exercise 1-2, computations with smooth limiters) you might not be able to start the computation due to getting negative pressures etc. at the first time step. If this turns out to be the case, either use a different limiter or calculate the first time step (or a few first steps) using zero slopes (i.e., effectively with the first-order scheme).

Task 2. Numerical simulation of a practical problem: blast wave interaction with an obstacle (wall).

Using the best version of your second-order TVD scheme (on the basis of Task 1), carry out the numerical modeling of blast wave interaction with a solid wall (assuming one-dimensional flow model).

The problem formulation is as follows. Let us imagine an explosive device located at $x = 0$, away from a solid wall located at $x = 1.5$. We assume that it is instantly detonated, resulting at $t = 0$ in a volume of pressurized, high-temperature gas extending from $x = -0.1$ till $x = 0.1$. (The same situation will occur at rupture of a pressurized vessel).

Assuming that atmospheric conditions are given by $\rho_1 = 1$ and $p_1 = 1$ (non-dimensional values), the initial pressure and density in the pressurized gas volume can be determined from the given ratios p_2/p_1 and ρ_2/ρ_1. Assume that the specific heat ratios of atmospheric air and detonation products are the same and equal to 1.4.

Calculate the problem for $p_2/p_1 = 1000$ and $\rho_2/\rho_1 = 100$ (i.e., assume that inside the pressurized, hot volume $p = 1000$ atm and $T = 3000$ K) and produce the following output:

(a) A few representative pressure and density instant distributions before and after blast wave reflection from the wall to illustrate the flow development; in this Task you should use solid lines to plot your numerical solutions (there is no exact theoretical solution in this case).

(b) Pressure history (i.e. pressure as a function of time) at a certain location not too far from the wall, e.g. $x = 1.3$. Obtain the same pressure history for a few successively refined meshes and make a judgement on what mesh size is needed to determine the peak (maximum) pressures at this point with a prescribed accuracy. A plot of peak pressures vs. number of nodes or grid step might be instructive here.

For this problem, numerical grid should be generated in such a way so that the wall and the initial boundaries of the high-pressure region would correspond to control volume faces. The computational domain boundary opposite to the wall should be placed far enough so that any waves would not reach it in the course of computation. An alternative is to notice that $x = 0$ is the symmetry point which could be replaced with a solid wall. It is advisable to have at least 10 (better more) control volumes inside the initial pressure region, otherwise waves inside it will not be resolved properly, which in turn may influence the results near the wall (which are of our main interest).

What is to be submitted

The electronic submission on MyCourses is mandatory. Another mandatory requirement: a *single* archive file (zip, rar) containing all the files must be uploaded. The archive file must be named using students name and project number (e.g., JohnSmith-Mech516Project2.zip). Your submission should contain the following:

- A report describing your numerical experiments and the results in sufficient detail. I do not require a lengthy report but it should be detailed enough for me to judge what has been done and how. It should provide the numerical scheme(s) you used, information on the computational domain, grid and time steps, and of course the results of computations (plots) with comments on them.
- The texts of your computer codes. The codes must be briefly commented.

Debugging hints:
Similar to the previous projects, before computing the requested problems you need to make sure that your code is free from errors ("bugs"). For that simple test problems should be used. For instance, at first, to compute a uniform flow (first, with zero velocity everywhere, and then with a non-zero velocity everywhere). Obviously, your code should produce unchanged solution, if all is correct. Then a single moving shock with constant parameters on both sides can be used etc.