Problem Set 10

Exercises for the Lecture Fundamentals of Simulation Methods
Prof. Dr. Ralf Klessen (Lecture Tuesday 9h - 11h and Thursday 9h - 11h)
Loke Lönnblad Ohlin (Tutor Group Thursday 11h - 13h, e-mail: loke.ohlin@uni-heidelberg.de)
Toni Peter (Tutor Group Thursday 14h - 16h, e-mail: toni.peter@uni-heidelberg.de)
Marcelo Barraza (Tutor Group Friday 11h - 13h, e-mail: barraza@mpia-hd.mpg.de)

Submit the solution to your tutor in electronic form by **Wednesday January 8, 2020**.

1. Numerical hydrodynamics – part 3

1.1. A numerical solution to a Riemann problem (*13 points*)

To finish our discussion of the Riemann problem, let us now use the classic solver as well as the Riemann solver developed in the last exercises to solve an actual Riemann problem on a grid (see `codes_hydro.zip` on elearning2). The problem we want to solve is defined as $\rho_L = 10^5$, $\rho_R = 1.24 \times 10^4$, $p_L = 1$, $p_R = 0.1$ and $u_L = u_R = 0$.

Use the adiabatic (not the isothermal) versions of the solvers. Use 200 grid points between -100 and $+100$, and integrate until $t = 5000$.

1. Solve this problem and plot the results for $\rho(x)$, $u(x)$, and $p(x)$ at the final time ($t = 5000$).
2. Compare the results between the classic and the HLL Riemann solver.
3. Redo the problem with 10x higher spatial resolution.
4. Explain the shape of the solution: where is the contact discontinuity, where is the expansion wave, and where is the shock wave?

1.2. Kelvin-Helmholtz instability (*17 points*)

In this exercise we apply a public domain magnetohydrodynamics code, as frequently used in computational astrophysics, to a standard test case: the Kelvin-Helmholtz instability as introduced in the lecture a few weeks ago.

We consider a 2D domain of extension $[0, L] \times [0, L]$ with periodic boundaries on the left and right sides, and reflecting boundaries on the top and bottom. Let the upper half of the box be filled with gas ($\gamma = 5/3$) at density $\rho_1 = 1.0$, pressure $P_1 = 1.0$, and velocity $u_1 = 0.3$ in the x-direction (i.e. to the right). The lower half has density $\rho_2 = 2.0$, the same pressure $P_2 = P_1$, and moves with velocity $u_2 = -0.3$ to the left.

In order to avoid a perfectly sharp boundary in the initial conditions between these two phases (which is prone to triggering secondary instabilities at grid corners) we introduce a small transition region that smoothly connects them:

$$\rho(x,y) = \rho_1 + \frac{\rho_2 - \rho_1}{1 + \exp\left[(y - 0.5)/\sigma\right]}, \tag{1}$$

and similarly

$$u(x,y) = u_1 + \frac{u_2 - u_1}{1 + \exp\left[(y - 0.5)/\sigma\right]}, \tag{2}$$

with $\sigma = 0.01$.

On these unperturbed initial conditions, we now impose a seed perturbation in the velocity in the y-direction of the form

$$v(x,y) = A\,\cos(kx)\,\exp(-k|y - 0.5|), \tag{3}$$

with wavenumber $k = 2 \times (2\pi/L)$ and perturbation amplitude $A = 0.05$. For simplicity, we refrain from imparting a perturbation in ρ and u as well that would be consistent with the velocity perturbation in the y-direction at the linear theory level. That means, we hope that we get away with this on the grounds that the perturbation should anyway grow (which it expected if the shear flow is indeed unstable against arbitrarily small transverse perturbations).

(a) We want to simulate this problem with the ATHENA mesh code developed by the group of Jim Stone (Institute for Advanced Studies in Princeton), whom you have met at the astronomical colloquium a few weeks ago. You can download the latest version of the code (Athena 4.2) from this link:
https://princetonuniversity.github.io/Athena-Cversion/AthenaDocsDownLd.
Information on how to install this program on your computer is given here:
https://princetonuniversity.github.io/Athena-Cversion/AthenaDocsTutQuickStart.
For those of you, who want to learn more about the code, the full documentation is found here: *https://princetonuniversity.github.io/Athena-Cversion/AthenaDocs*.

We want to run the problem with ATHENA until time $t = 3.0$ and create images of the resulting density field as time progresses. To this end, you need to implement the corresponding initial conditions in a problem generator according to the design of this code, and then compile the code appropriately. For the problem generator, you can use the file `kelvin.c` provided on moodle and place it into the subdirectory `src/prob` of ATHENA. Edit the file to finish off the implementation of the initial conditions. There are primarily three lines to fill out, see the comments in the file. Specifically, provide the initial density and velocity field as described above.

In the primary code directory, configure the code as

```
./configure --with-problem=kelvin --with-gas=hydro
--with-eos=adiabatic --with-flux=roe
```

followed by the compilation step with `make all`.

Next, you also need to setup a parameterfile that is passed to ATHENA at run time. This sets things such as the resolution you want to use, the number and times of outputs you want to have, the desired simulation time span, etc. You can try the parameterfile `kelvin.param` supplied on elearning2 to get started, which you may modify as you see fit (for example to change the resolution or the parameters of the initial conditions generator). Then run the code with
`./bin/Athena -i <parameterfile>`,
where you replace the name of the parameterfile with your file `kelvin.param`.

You should get a sequence of ".ppm" image files displaying a slice of the density field. The last image is `kh.0060.d.ppm`. Carry out a series of simulations with different resolutions, equal to 64×64, 128×128 and 256×256 mesh cells, and produce images for them at the same nominal pixel resolution, for example 512×512 pixels, by enlarging the images accordingly. Compare the last output data visually and discuss.

(b) We now want to check whether we can verify the linear growth rate of the perturbation. As discussed in the lecture, the growth rate of a single mode k is given by $\propto \exp(\omega t)$, with

$$\omega = k|u_1 - u_2|\sqrt{\rho_1 \rho_2}/(\rho_1 + \rho_2) \, . \tag{4}$$

Make a plot of the log of the mean kinetic energy in the y-direction as a function of time (you can get this quantity from the history output in `kh.hst`, column 1 has the time, column 9 the kinetic energy in the y-direction), and overplot a growth line reflecting the above timescale. Why is the growth initially slower than expected based on above equation? What could be the reason that there is a large slow-down at late times?

(c) Now repeat the Kelvin-Helmholtz simulation of (a) but add a constant velocity of $\Delta u = 5.0$ everywhere to the initial conditions. At time $t = 3$, would you expect the result to look different than in (a)? Compare with what you actually obtain when doing this test, and discuss the result.