4 Hydrodynamics

Generate Plummer gas spheres

A stars behaves like liquid spheres when interacting with another star, and those we treat with hydrodynamics. Gravity is obviously also important, but for this experiment we adopt the built in gravity solver in the hydrodynamics codes (that is easier, and sufficient for our current assignment).

Write a script that generate a single plummer distributions of gaseous (SPH) particles, with mass m and characteristic size r. In the script `hydro_simple.py` in the directory `example/syllabus` you can see how to do this. Adopt an adiabatic equation of state for the gas and make sure to turn self gravity on.

Run your first experiments with as few particles as possible, just to get the code working and get some feeling for the code. Is your model stable with $N = 10$ particles, and what about for $N = 100$ and $N = 1000$?

Plot the wall-clock time for running the simulation for one time-step as a function of N. And how does the energy error in your integration vary with N.

Run until equilibrium and convergence

After you have acquired a stable initial configuration, increase the number of SPH particles for a more robust result. Plot as a function of time the angular momentum, and plot the initial and final radial density profile. Calculate the kinetic and potential energies and compare them with the initial value.

Instead of plotting a large number of density profiles in one frame, one often refers to Lagrangian radii; those are the distance from the center of mass which contain x% of the initial mass. In AMUSE there is a standard routine for calculating Lagrangian radii. Use this function to plot, as a function of time the 10%, 25%, 50% and 75% Lagrangian radius of your gaseous sphere.

You have a converged model if your resulting density profile becomes independent of N. You can test this by making a plot, printing the numbers but the best way is by performing a Kolmogorov-Smirnoff test. The KS test (for short) returns a probability that one model is a coincidental random representation of your other model. It is a handy test to remember.

Smash them up

New generate a second Plummer sphere (or make a copy of your first one), and smash them into each other with radial velocity v from a distance $2r$. At first instance adopt $v = 0\,\mathrm{km/s}$, but increase it until you obliterate the two gaseous bodies. Of course, then there is no equilibrium model, and therefore you will have to stop the code if you become impatient.

Plot as a function of v the Lagrangian radii of the merger product (simulated until it is in equilibrium and for the converged solutions, if possible). Can you predict (or postdict) what happens if $v^2 \simeq 2Gm/r$?