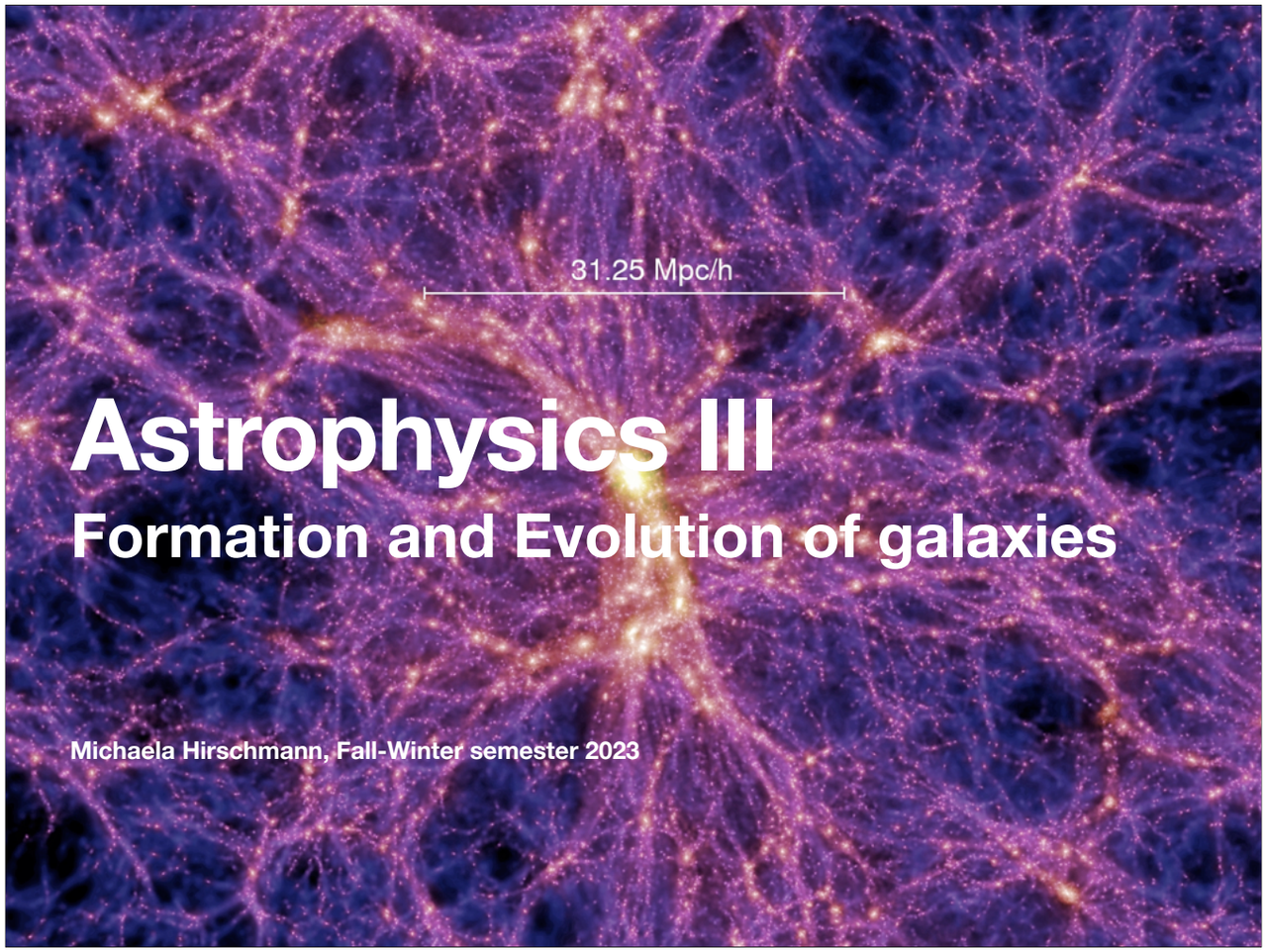


Astrophysics III

Formation and Evolution of galaxies

Michaela Hirschmann, Fall-Winter semester 2023

Lecture content and schedule

- *Chapter 1*: Introduction (galaxy definition, astronomical scales, observable quantities — repetition of Astro-I)
- *Chapter 2:* Brief review on stars
- *Chapter 3*: Radiation processes in galaxies and telescopes;
- *Chapter 4:* The Milky Way
- *Chapter 5*: The world of galaxies I
- *Chapter 6*: The world of galaxies II
- *Chapter 7:* Black holes and active galactic nuclei
- *Chapter 8:* Galaxies and their environment;
- *Chapter 9:* High-redshift galaxies
- *Chapter 10*:
  - Cosmology in a nutshell; Linear structure formation in the early Universe
- *Chapter 11*:
  - Dark matter and the large-scale structure
  - Cosmological N-body simulations of dark matter
- *Chapter 12*: Populating dark matter halos with baryons: Semi-empirical & semi-analytical models
- *Chapter 13:* Modelling the evolution of gas in galaxies: Hydrodynamics
- *Chapter 14*: Gas cooling/heating and star formation
- *Chapter 15*: Stellar feedback processes
- *Chapter 16*: Black hole growth & AGN feedback processes
- *Chapter 17*: Modern simulations & future prospects

Part I:
Observational basics & facts of galaxies
first 7 lectures

Part II:
Theory & models of
galaxy evolution processes
second 7 lectures

Outline of this lecture

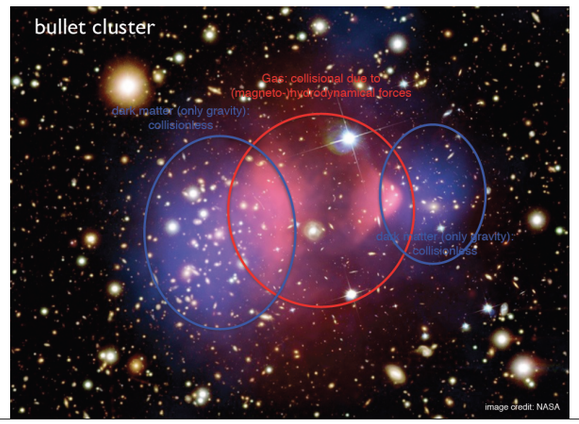
- Why hydrodynamics?
- Eulerian methods/mesh codes
  - Advection schemes
  - Riemann problems
  - Adaptive mesh refinement
- Lagrangian methods/SPH codes
  - SPH & Equations of motions
  - Modern SPH: Density-entropy formulation
- "Combined" methods: Moving-mesh scheme

The importance of modelling gas

- Everything we can observed/see is gas or is made from gas

image credit: NASA

- Need to follow gas dynamics
  - To form galaxies and stars in a spatially resolved fashion
  - To study the interstellar, intergalactic & intracluster medium in 3D
  - …e.g. to compare images of simulated with observed galaxies



— **Why do we need hydrodynamics for modelling galaxies?**
—> Everything, what we can see/observe, in particular the internal structure of galaxies, **is gas or is made from gas**!!

— How do we model the **evolution of gas**?
—> **Approximate it as a hydrodynamic fluid**.

— **Why is it reasonable?**

What is fluid/hydrodynamics ?

- Study of the behaviour of fluids—liquids and gases—and their interactions via external forces.
- Focus on understanding the macroscopic properties of fluids, such as velocity, pressure, density, and temperature.
- Approximation that the individual gas/fluid elements are treated collectively as a continuous medium with well-defined macroscopic properties.
- Founded on the principles of conservation of mass, momentum, and energy —> the total mass, total momentum, and total energy of a fluid system are constant —> hydrodynamic equations
- Collisional Nature: gas particles in galaxies can interact collisions, pressure forces, and radiative processes. —> energy dissipation, mass redistribution, and shocks

—Because: Fluid/Hydrodynamics means to study the **behaviour of fluids and their interactions** via external forces.

—> Thereby, **we focus on understanding** the macroscopic properties of fluids (velocity, pressure, density, temperature), as they change in response to internal and external forces.

—> We make **the approximation** that individual gas/fluid elements are treated collectively as a continuous medium with well-defined macroscopic properties.

—**Hydrodynamics is founded on the principles of conservation of mass, momentum and energy**

—> **leading to the hydrodynamical equations** (also used for climate modelling, weather forecasts, space weather, airplanes etc.).

— Hydrodynamic equations describe the **collisional nature of gas**: it can interact via many different processes such as collisions, pressure forces, and radiative processes,

—> which lead to energy dissipation, mass redistribution and shocks.

—> This is fundamentally different from collisionless DM (and stars), just interacting gravitationally (e.g., cannot radiative away energy).

Advantages of hydrodynamic approach

- Several advantages compared to semi-analytics:
  - Interaction between DM and baryons is taken into account (without relying on approximations like adiabatic contraction)
  - Dynamics of the diffuse cooling gas is captured in full generality (no spherical symmetry or quasi-static evolution)
  - Once some sub-resolution schemes for feedback are adopted, hydro-sims can treat the subsequent evolution of the supernovae/AGN-driven winds fully self-consistently
  - Automatically account for morphological transitions during mergers, environmental processes etc.
- Hydrodynamics suitable for such complex problems, but there is no perfect numerical recipe for all hydrodynamic problems
- Depending on the problem of investigation, one may also include different other physics *(see e.g. Springel+10)* like
  - radiation-hydrodynamics (accounting for the interaction of photons with the gas)
  - magneto-hydrodynamics (accounting for the interaction of magnetic fields with the gas) and cosmic rays

— Compared to SAMs, such a **hydrodynamic approach has several advantages:**

—> **First**: We explicitly account for the interaction between DM, gas and stars and not just assume adiabatic contraction of the hot gaseous halo, but instead account for both dissipative and dissipationless processes.

—> **Second:** Dynamics of the gas cooling is captured in full generality without having to make assumptions such as spherical symmetry or quasi-static evolution as in SAMs.

—> **Third:** Once some sub-resolution schemes for feedback are adopted, subsequent evolution of stellar/AGN winds followed **without adopting simplified prescriptions to specify the degree of ejection (out of galaxy/halo) and the re-accretion/re-incorporation time-scale as done in SAMs.**

—> **Fourth: H**ydrodynamic simulations (at sufficiently high resolution) automatically account for morphological transformations during mergers, environment processes etc.

— If you are interested in more detail, how hydrodynamic simulations work, I refer you to the study of Springel 2010, where different methods are well explained.

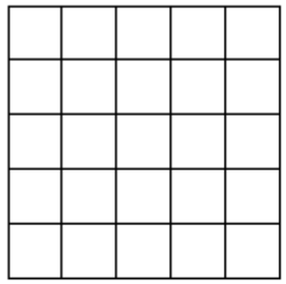
— In this chapter, we will **just focus on very basic hydrodynamic schemes,**

—>without taking into account any interaction of the gas with radiation or magnetic fields.

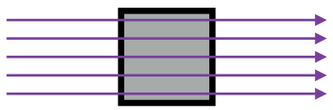
Eulerian versus Lagrangian methods

Eulerian/grid methods

discretise space
finite-volume scheme



use a grid fixed in space



Flow through fixed cell

Moving mesh

discretise space
finite-volume scheme

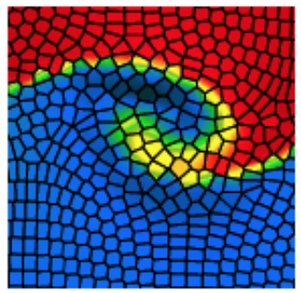
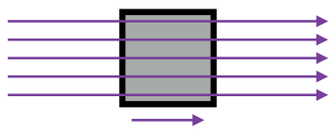


image credit: V. Springel

uses an unstructured mesh moving with the flow

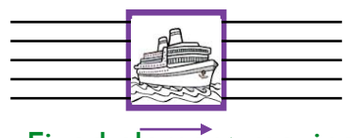


Flow through cell moving with flow

Lagrangian/SPH methods

discretise mass

use particles for the gas (like in N-body) which move with the flow



Fixed element moving with flow

— For **numerically solving the hydrodynamic equations of a gaseous fluid**,
—>we can distinguish between **Eulerian and Lagrangian** methods.
—> Eulerian methods discretise space and follow/compute the gas flow through a fixed cell.
—> Lagrangian methods, instead, discretise gas in mass and use particles that move with the flow.

— As we shall see, both approaches have their pro's and con's and
—>to combine the advantages, some combined, so called moving mesh/meshless techniques have been developed.

Outline of this lecture

- Why hydrodynamics?

- Eulerian methods/mesh codes
  - Advection schemes
  - Riemann problems
  - Adaptive mesh refinement

- Lagrangian methods/SPH codes
  - SPH & Equations of motions
  - Modern SPH: Density-entropy formulation

- Combined methods: Moving-mesh scheme

Let’s first start with the very basics of Eulerian methods.

The hydrodynamic equations

- Eulerian/Grid-based methods are based on the fluid-dynamical, so called Euler equations
- These equations are expressed in terms of conserved quantities:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0$$

- Continuity equation, mass conservation

$$\frac{\partial (\rho \mathbf{u})}{\partial t} + \nabla \cdot (\rho \mathbf{u}\mathbf{u}) + \nabla P = \mathbf{0}$$

- Momentum equation, momentum conservation

$$\frac{\partial \rho e_{\mathrm{tot}}}{\partial t} + \nabla \cdot ((\rho e_{\mathrm{tot}} + P)\mathbf{u}) = 0$$

- First law of thermodynamics, energy conservation

$$e_{\mathrm{tot}} = e + \frac{u^2}{2}$$

where e = internal energy per unit mass

$$P = (\gamma - 1)\rho e$$

- Equation of state for an ideal, monoatomic gas with the polytropic index γ = 5/3

— Eulerian methods are based on the **fluid-dynamical/hydrodynamical equations.**

— These equations are expressed in terms of **conserved quantities,** i.e. the density, momentum and energy density
—> the **continuity** equation is conserving density (rho is the density, u the velocity vector, P the gas pressure),
—> the **momentum** equation is conserving momentum,
—> the **first law of TD** is concerning energy, with specific total energy e_tot = specific internal (thermal) energy + kinetic energy per unit mass.

—We can also take advantage of the **equation of state for an ideal monoatomic gas** with the polytropic index gamma of 5/3, relating the pressure to the density and the specific thermal energy.

— Remember, we have **already worked with such equations for DM**, but they were including gravity; here for simplicity, we neglect self-gravity (but in modern simulations, gravity is of course included).

Basic methodology of a grid code

- Primitive variables: $\rho, \vec{u}, P$
- Conservative variables, density, momentum density and total energy density $q = (\rho, \rho\vec{u}, \rho e)$
- Standard approach: divide space into grid cells and store the cell-averaged conservative quantities at all the grid points
- Use re-construction schemes, which can take several neighbouring cells into account to reconstruct the field/distribution of any variable
- Advection: solve hydro-equations by computing the flux of mass, momentum and energy across grid cell boundaries/contact discontinuities, i.e. solve Riemann problem
- Calculate new cell-averaged conservative quantities and do re-constructions etc…

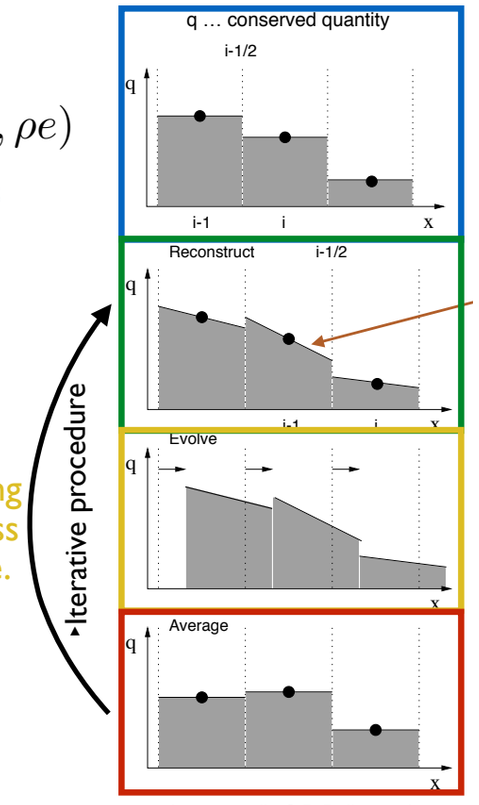


image credit: C.P. Dullemond

— Then let's continue with going through the **basic methodology of a grid/Eulerian code** to numerically solve these hydrodynamic equations.

— In principle the quantities to describe the gas are the density rho, velocity u, pressure P,
—> in grid codes, one is working with the **conserved quantities**, i.e. density rho, momentum density rho u, and total energy density rho e.

— In the standard approach:
—> One **divides space into grid cells** (see the top **blue** panel).
—> Then we assume a certain reconstruction scheme, i.e. how a quantity, e.g. the density, is distributed in a cell, which is normally a better approximation of reality than a constant distribution (green panel).
—> Next, at most cell boundaries, there are some **"discontinuities" between different conserved quantities,** which is the "**Riemann problem".**
—> By solving it, we get the **flux of mass, of momentum and of energy from one cell to another; this is the CORE STEP** (yellow panel).
—> Lastly, if you know fluxes across all cell boundaries, you can calculate the new cell-averaged conservative quantities, do again the reconstruction etc.

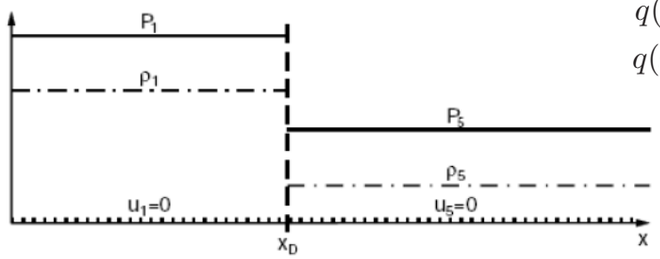
— This standard approach is an **iterative process** with that the fluid/gas is advected through stationary cells.
—>in other words you iterate this scheme to **model a moving gas flow**!

How to solve coupled hydro equations?

- With suitable advection scheme, we can solve a simple test case

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0$$

- Hydrodynamic equations are, however, coupled —> No simple global decomposition possible
- To finally fully solve the hydro equations at each cell interface (and to capture shocks and contact discontinuities), we have to solve the "Riemann" problem
- Specific example for Riemann problem: shock tube



$$q(x, t = t_0) = q_{\mathrm{L}} \quad \text{for } x < x_0$$
$$q(x, t = t_0) = q_{\mathrm{R}} \quad \text{for } x > x_0$$

— If the gas velocity u is constant, the fluid equations reduce to a **very simple problem/test case** to solve.

— But: hydro equations are coupled, **no simple global decomposition possible.**

— To finally fully solve the hydrodynamic equations at each cell interface **implies that we have to solve the Riemann problem.**

— Here you can see a s**pecific Riemann problem**:
—> a shock tube, which is a fluid with piecewise constant properties having a single discontinuity with initially zero velocity
—> simplified case what we have at each cell boundary if velocity is zero

— This **specific Riemann problem**, shock tube, is very useful for the numerically solving the fluid equations,
—> because there exists an **exact, analytic solution.**
—> Thus, it is a good test to **double-check numerically solved fluid equations.**

Riemann solvers

- There exists an exact solution for the general Riemann problem, can be found in textbooks (e.g. Courant & Friedrichs '48)
  - One can show that solutions are self-similar, i.e. they only depend on $q(x,t) = q(x - x_0/(t - t_0))$
  - Very time-consuming
  - Various approximate solutions, e.g.
    - Roe's linearised Riemann solver (Powell+99)
    - HLLE method (Harten, Lax, van Leer and Einfeldt)
    - HLLC method (Harten-Lax-van Leer-Contact)
- A more detailed description of these methods is outside the scope of this lecture…
- They all have their pro's and con's…

— In general, **to solve the Riemann problem (fluxes at cell interface),** there are different possibilities,
—> there exists an exact solution, which can be found in textbooks,
—> but this is very time consuming, **often approximate solutions** are considered such as…
—> They all have their pro's and con's, for more in-depth information, I refer you to a dedicated course on fluid dynamics.

Grid codes in 3D

- So far, discussed everything in 1D, but galaxies are 3D

- Directionally split schemes
  - Apply 1D hydro-solver alternately along the different directions
  - Less memory needed
  - But spherical symmetry is less well preserved

- Unsplit schemes
  - Compute fluxes for all interfaces of cells
  - Update cell values once per time step
  - E.g. used in Ramses

- (Self-)gravity (hydro in combination with DM-N-body) can be simply added in the momentum equation

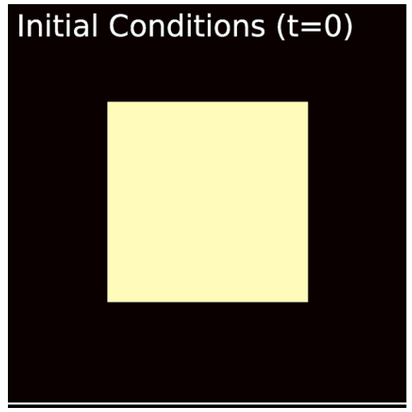
— So far we have discussed everything in 1D, but of course we want to **model galaxies and their gas in 3D.**
—> There are two different possibilities of how to do that, through directionally split schemes and unspoilt schemes (see slide for explanation).
—> In most modern grid codes, the **unsplit scheme is used**.

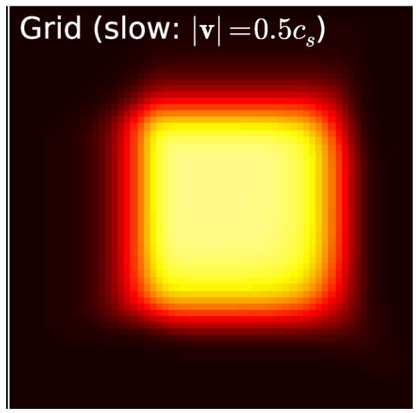
— And of course, for additional **self-gravity and gravity from stars and DM**, we have to calculate the gravitational forces by simply extending the momentum equation by a “rho nabla Phi”.
—> Note that the N-body method to get the gravitational force is connected to hydrodynamics, both are run together.

Success and Limitations

- Main strength is the accurate hydro,
- Automatically accounts for contact discontinuities, shocks etc.

- One major limitation of grid codes is their spatial resolution when we use a fixed grid size, increasing the grid size globally is computationally very expensive
- Advection errors: somewhat diffusive (no exact energy conservation) —> unphysical forces
- No exact conservation of angular momentum
- No Galilean invariance (Galilean transformation to a different inertial system)





— **The main strength of the Eulerian method** is its accurate hydrodynamics, precisely accounting for contact discontinuities etc.

—**But Eulerian also suffer from some limitations**:
1. One major limitation of classic grid codes is their spatial resolution (when using a fixed grid size, increasing the grid size globally is computationally very expensive) —> but see next slide for a solution.

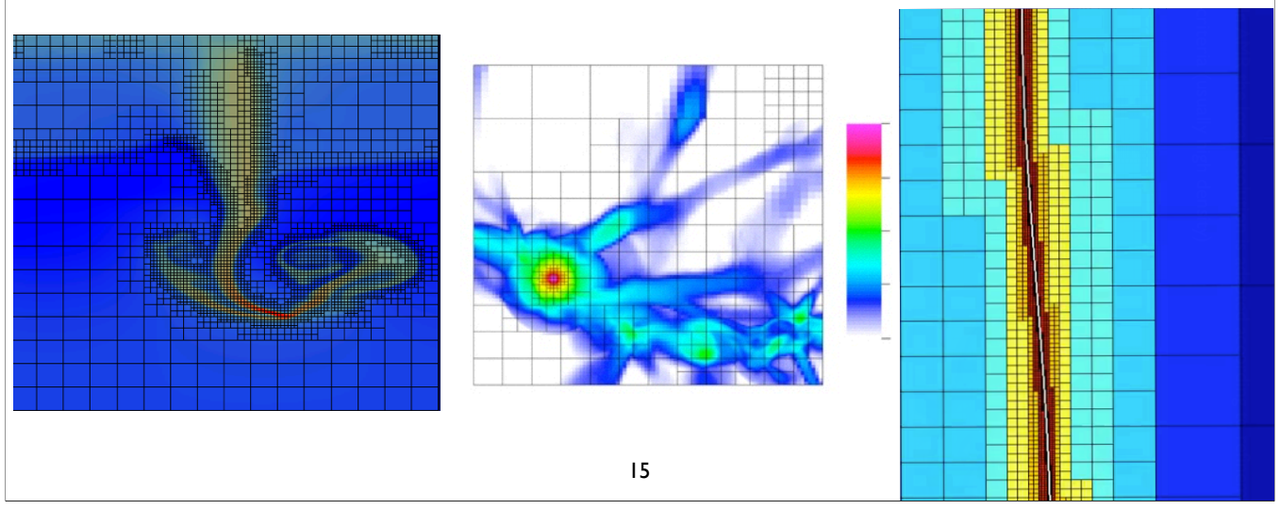
2. Grid codes can have some **advection errors (due to the reconstruction scheme)**, when fluids (especially with sharp gradients) move across cells. These errors produce **artificial diffusion**, and can manifest as unphysical forces.
—> This has the consequence that these schemes **are not exactly energy conserving,**
—> shown by the **hydrostatic square advection test** (illustrated by the panels): initial condition of a high-density square of gas in hydrostatic equilibrium with the ambient gas, moving at constant velocity (|v| ~ 150).
—> You can see that after some time **contact discontinuities are 'smeared out'** due to advection errors.

3. Grid codes also do **not exactly conserve angular momentum.**
—> For example rotating discs are 'torqued' into alignment with the grid cardinal axes,
—> which can be problematic for e.g. rotationally supported disk galaxies.

4. Related to that: grid codes are **not fully Galilean invariant**: Motions would not be exactly the same in different inertial systems.

Adaptive mesh refinement

- A good possibility to overcome the limitation in spatial resolution is the adaptive mesh refinement technique:
  - the local resolution (i.e. number of grid cells) is adapted according to refinement criteria (typically high density, more cells at high density regions)



— **One of the problems**, namely the **limited spatial resolution**, can be overcome easily, typically done in modern grid codes:
—> through **adaptive mesh refinement technique, where the local resolution is adapted to some refinement criteria**
—> i.e. more cells are generated for higher density regions, as you can see in these images.

Summary: Eulerian methods on a grid

- Gas flow through fixed cells/grids
- Basic variables: density, momentum density and energy density
- Their evolution described by Euler equations —> conservation of mass, momentum and energy
- Transport of fluid flow through cells: reconstruction — advection (solving Riemann problem) — cell averaging
- Variable local resolution —> adaptive mesh refinement
- Different advantages and disadvantages…

Outline of this lecture

- Why hydrodynamics?

- Eulerian methods/mesh codes
  - Advection schemes
  - Riemann problems
  - Adaptive mesh refinement

- Lagrangian methods/SPH codes
  - SPH & Equations of motions
  - Modern SPH: Density-entropy formulation

- Combined methods: Moving-mesh scheme

Let’s turn to Lagrangian methods.

The hydro-equations in Lagrangian form

- **Follow a gas element along its path** and see how it changes its direction of motion, its density & its pressure…
- To derive the corresponding Lagrangian form of the hydro equations, we need do introduce the co-moving convective derivative D_t as

$$\frac{d}{dt} = \frac{\partial}{\partial t} + \vec{u} \cdot \vec{\nabla}$$

- With that we can re-write the hydro-equations for an inviscous fluid:

$$\frac{d\rho}{dt} + \rho \vec{\nabla} \cdot \vec{u} = 0$$

$$\frac{d\vec{u}}{dt} = -\frac{\vec{\nabla} P}{\rho}$$

$$\frac{de}{dt} = -\frac{P}{\rho} \vec{\nabla} \cdot \vec{u}$$

- Continuity equation, a gas parcel changes its density when the gas motion converges
- Momentum equation, a gas parcel will be accelerated due to a force which is the pressure gradient (+ grav. pot.)
- First law of thermodynamics, the thermal energy of a gas parcel changes only as a result of adiabatic compression/expansion

where e = thermal energy per unit mass

- Exercise: conversion for momentum and energy equation

$$P = (\gamma - 1)\rho e$$

- Equation of state for an ideal, monoatomic gas with the polytropic index γ = 5/3

—**Lagrangian** methods are fundamentally different compared to Eulerian methods:
—> Instead of following the flow through a stationary grid, **we follow a gas element along its path** and see how its properties change.

—To do that, we need to first derive the **hydrodynamical equations in Lagrangian form, shown on this slide.**
—> On the next slide, this is done for the **Continuity equation, and in the exercises also for the momentum and energy equations.**

—And as before, we also take advantage of the equation of state, relating P with density and energy for an ideal, monoatomic gas with polytropic index gamma of 5/3.

Classic continuity equation:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \underset{\sim}{u}) = 0$$

Co-moving derivative: $\frac{d}{dt} \equiv \frac{\partial}{\partial t} + \underset{\sim}{u} \cdot \nabla$

"partial time derivative at fixed position changed, due to move of the fluid element"

$$\frac{\partial \rho}{\partial t} + \rho \nabla \cdot \underset{\sim}{u} + \underset{\sim}{u} \cdot \nabla \rho = 0$$

$$= d\rho/dt$$

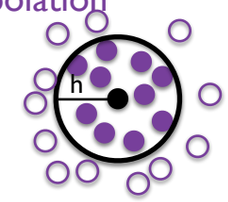
$$\Rightarrow \boxed{\frac{d\rho}{dt} + \rho \nabla \cdot \underset{\sim}{u} = 0}$$

Continuity equation in Lagrangian form

This equation reflects the fact that, for a moving fluid element, changes of density with time come both from intrinsic local variations and from the element "sampling" different parts of the field as it moves.

Smoothed particle hydrodynamics

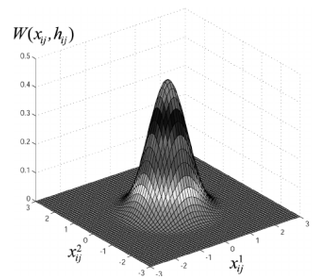
- Idea: Treat a hydrodynamic fluid in a completely mesh-free fashion —> Use a set of sampling particles to represent the fluid
- Technique to approximate the continuum dynamics of fluids through the use of particles which can be viewed as interpolation points
- The density of a particle in SPH is given by smoothing over nearest neighbour particle masses within a "Kernel"



$$\rho(\vec{x}) = \sum_j mW(\vec{x} - \vec{x_j}, h)$$

- Kernel depends on inter-particle distance r=|$\mathbf{x_i}$-$\mathbf{x_j}$| and on "smoothing length" h
- Most commonly the cubic spline Kernel is used

$$W(r,h) = \frac{8}{\pi h^3} \begin{cases} 1 - 6(r/h)^2 + 6(r/h)^3 & 0 \le r/h \le 1/2 \\ 2(1 - r/h)^3 & 1/2 \le r/h \le 1 \\ 0 & 1 < r/h \end{cases}$$



— One of the most used **Lagrangian approaches in astrophysics is the smoothed particle hydrodynamics (SPH) approach.**
—> This approach was developed in the 70s specifically for astrophysical problems.
—> The idea behind is to treat a **hydrodynamic fluid in a completely mesh-free fashion**, and to approximate the continuous dynamics of fluids through **the use of particles** considered as interpolation points.

— **What do I mean with interpolation points?**
—> The basic axiom of SPH is the **smoothing over masses of nearest neighbour particles with a volume-weighting Kernel function to get the density of a particle.**
—> The density **rho of a particle** is described by the expression shown on the slide: **rho is the sum of the kernel-weighted mass of the nearest neighbour particles**

— Such a **"Kernel"** is basically the heart of SPH.
—> The kernel is **volume-weighting** the mass of neighbour particles (with high contribution from close particles, low contribution from more distant particles). Thus,
—> (i) the kernel is depending on the **distance** from the main particle and
—> (ii) the kernel is depending on the **smoothing length h** (= sphere within which nearest neighbours are considered)

— **One can use different Kernels**, meaning different volume-weighting schemes,
—> e.g. the cubic spline Kernel, which is specified and shown here.
—> Nowadays, thanks to better convergence and numerical behaviour, the **Wendland kernel with 100-200 neighbours is used** [Note that the kernel is normalised to unity].

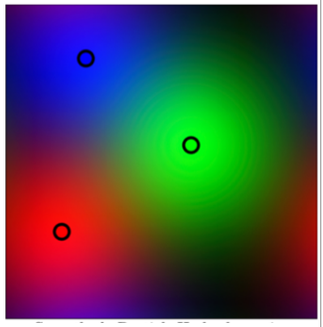
Smoothed particle hydrodynamics

- For any field F, we can define a smoothed version

$$F_s(\vec{x}) = \sum_j \frac{m_j}{\rho_j} F_j W(\vec{x} - \vec{x_j}, h)$$

- The derivative of field F is given by

$$\vec{\nabla} F_s(\vec{x}) = \sum_j \frac{m_j}{\rho_j} F_j \vec{\nabla} W(\vec{x} - \vec{x_j}, h)$$



- Define constraints for the smoothing length h_i, e.g. requiring that the Kernel volume contains a constant mass for the estimated density

$$\rho_i h_i^3 = const. \propto M_{kernel} = N_{ngb} * m_i$$

- In principle, one could take these equations for **F** (replace with **P** or **u***), and insert these expressions into the Lagrangian fluid equations to get the equations of motions.

*Note that bold print means vector!

—Then, for any field, like the Pressure P, energy E, velocity u, we can define **a smoothed, interpolated version through a convolution with the kernel function W(x, h).**

—How can we get **constraints on the smoothing length**, how to choose that?
—> Typically interpolation **points/particles have a density** such that rho times h^3 is constant.
—> This means that the **smoothing length h is variable** to sample different densities **by setting a fixed Kernel mass, i.e. a fixed number of neighbours.**
—> This implies for a high density a small smoothing length h, and for a low density a high smoothing length h, leading to an **automatic spatial refinement** in high-density regions (not introduced afterwards as in grid codes).

—In principle, **we could just take these equations for any quantity F, replace it with the Pressure or velocity, in insert them into the Lagrangian hydrodynamic equations to get the equations of motion**.
—> **However**, this approach has been shown to be not fully momentum or energy conserving.
—> Instead, **one can derive the equations of motion via a proper Lagrangian formalism,** which I won't show here as it does not give any additional insights, you can find it in Springel & Hernquist 2002**.**

Entropy conserving equation of motion

- After some algebra, as demonstrated by Springel & Hernquist 2002, we obtain the EoM

$$\frac{d\vec{u}}{dt} = -\sum_j m_j \left(f_j \frac{P_j}{\rho_j^2} \overrightarrow{\nabla} W(\vec{x}_i - \vec{x}_j, h_j) + f_i \frac{P_i}{\rho_i^2} \overrightarrow{\nabla} W(\vec{x}_i - \vec{x}_j, h_i) \right)$$

- where the coefficients f_i are defined as $f_i = \left(1 + \frac{h_i}{3\rho_i} \frac{\partial \rho_i}{\partial h_i}\right)$

- In essence, we have transformed a complicated system of partial differential equations into a much simpler set of ordinary differential equations, we only have to solve one equation
- The continuity equation does not have to be evolved explicitly: density can be, at any point, calculated from the particle positions and masses
- The thermal energy and pressure can be derived from the density
- In these forms, the velocity (and thus, the positions dr/dt = u) of SPH particles can be integrated forward in time as in N-body simulations (e.g. using the leap-frog scheme)

—After some algebra, one obtains the **equation of motion, i.e. an expression for du/dt (dr/dt is simply u!)** as shown here.
—> du/dt is dependent on the particle mass, the pressure, the density, the smoothing length h, and the kernel function W.

—In essence, we have transformed a complicated system of partial differential equations into a much simpler set of ordinary differential equations, where we have to effectively solve only equation (du/dt), and where mass, momentum and energy is conserved due to the Lagrangian formulation.

—Note that the continuity equation does not have to be evolved explicitly as it is automatically done in Lagrangian schemes:
—> **Density is no longer a variable, but can be, at any point, calculated from the particle positions and masses.**
—> **Pressure** is dependent on density and entropy fct, the former is given by the particle position and mass, the latter is conserved/constant.
—> **Thermal energy** can be derived through the equation of state dependent on pressure and density.

—And then, we can compute the velocities and thus, also the positions via dr/dt = u of SPH gas particles by integrating the equations du/dt and dr/dt.
—> This **moves the system of gas particles forward** in time as done for N-body simulations (using the same time integration scheme).

—Considering **self-gravity of the gas-fluid and of DM**, adds some gravitational force terms in the equation for du/dt.

Advantages of classic SPH

Advantages:

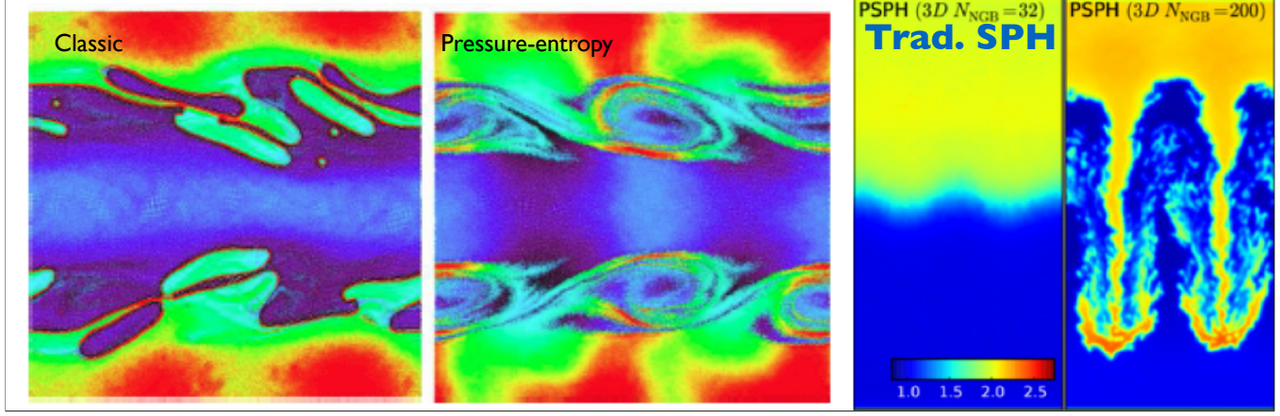
- Numerically rather robust
- Excellent conservation properties
- Lagrangian nature: trace flow and galilean invariant
- Good spatial resolution in high-density regions due to adaptive nature in density
- Couples straight forward to N-body gravity methods
- Intuitive for including sub-resolution physics

Advantages of SPH codes are …

Problems of classic SPH

Short-comings:

- Poor description of fluid-mixing: dynamical instabilities/contact discontinuities (Rayleigh-Taylor, Kelvin-Helmholtz) —> modern SPH (smoothing over Pressure)
- Surface tension error: cold Kauffmann "blobs" form, not destroyable —> modern SPH
- Slow numerical convergence
- Compromised accuracy when modelling ISM and galaxy formation



—But **SPH codes also have a number of short-comings/limitations**:
—> Rather slow numerical convergence,
—> **Poor description of fluid-mixing such as dynamical instabilities and contact discontinuities** (Rayleigh-Taylor and Kelvin-Helmholtz, see below).

—In the bottom right panel you can see a **Rayleigh-Taylor instability**:
—>This is an instability of an interface between **two fluids of different densities,** which occurs when a **denser fluid lies on top of a lighter/less dense fluid**, e.g. important for cloud formation in the atmosphere, or supernova explosions (also Lava lamps from the 80s).
—> The rightmost sub-panel shows how the instability should look like and left sub-panel shows what we get with SPH (instabilities are "smoothed out").

— Another important dynamical instability is **the Kelvin-Helmholtz instability**: it occurs for a **velocity difference between two contact fluids with different densities due to viscosity** (like wind blowing over water).
—> In the left panels, you can see that for classic SPH, it is not working properly (again too smoothed).
—> It is been shown that **these fluid-mixing problems can be solved by smoothing over the gas pressure instead of the density (Pressure-entropy formulation of SPH)**, what we refer to as "modern" SPH (mainly used in modern simulation codes).

Summary: Smoothed Particle Hydro

- Gas flow described by particles moving with the flow
- Their evolution described by hydro equations in Langragian form —> conservation of mass, momentum and energy
- Particles are interpolation points, Kernel-weighted density, pressure etc.
- Key quantity: smoothing length, related to the Kernel volume
- Solve one equation of motion for u (integrate), and change hydro properties of gas particle (r, u, e) accordingly
- Modern SPH solves problems of contact discontinuities/fluid mixing (by smoothing over pressure/entropy instead of density)
- Automatic refinement on density/pressure, exactly Galilean invariant, conserves angular momentum (& entropy) exactly
- Somewhat less accurate hydro (contact disc.), slower convergence, but largely alleviated in modern SPH

Outline of this lecture

- Why hydrodynamics?

- Eulerian methods/mesh codes
  - Advection schemes
  - Riemann problems
  - Adaptive mesh refinement

- Lagrangian methods/SPH codes
  - SPH & Equations of motions
  - Modern SPH: Density-entropy formulation

- "Combined methods": Moving-mesh scheme

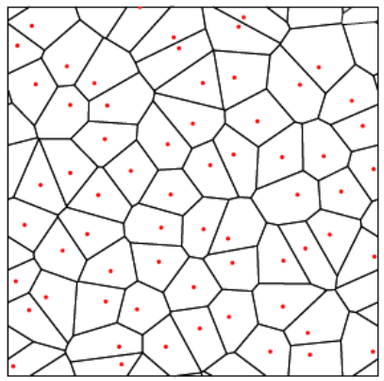
You have seen that both **grid and SPH codes have their pro's and con's.**

—> What about a method combining only the advantages and avoiding disadvantages?

—> **Combined methods have been developed**, such as the moving-mesh scheme.

Moving mesh hydrodynamics

- To combine advantages of both Lagrangian and Eulerian codes: "moving mesh scheme", e.g. Arepo (Springel+10)
- Transform a set of points into an unstructured mesh using a Voronoi tessellation (cell size dependent on density, each particle represents a cell)
- Cells/points are allowed to move with the fluid like in a Lagrangian scheme
- Hydro-solver: based on a second-order unsplit Godunov scheme with an exact Riemann solver
- Transform updated cell average quantities back to particle scheme and move "particles" i.e. grid cells



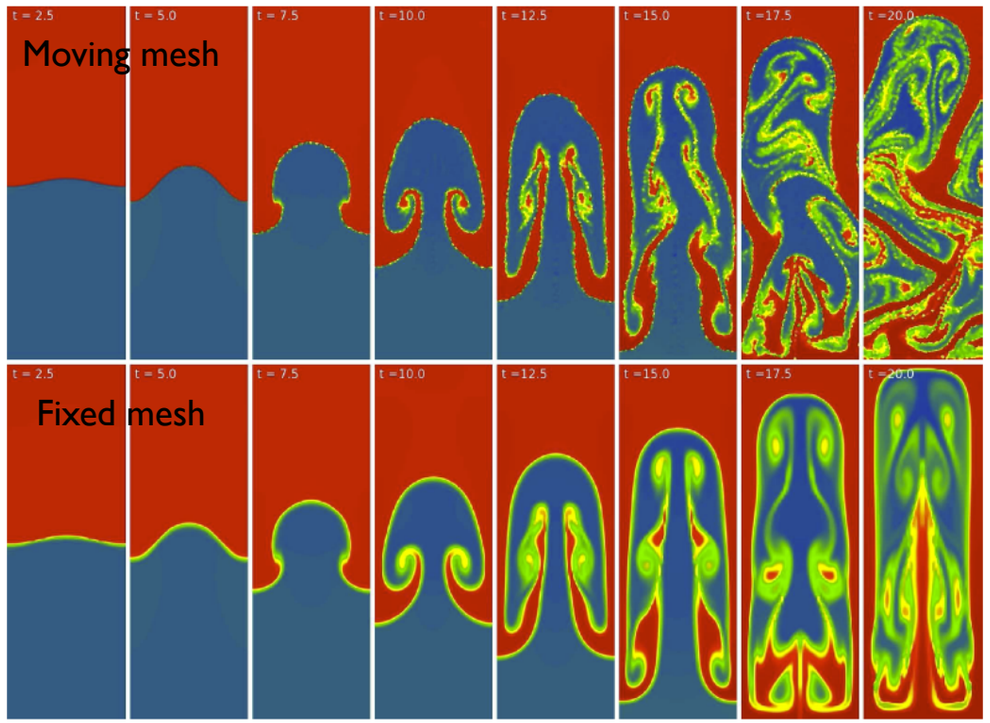
For a given set of points, a Voronoi tessellation of space consists of non-overlapping cells around each of the sites such that each cell contains the region of space closer to it than any of the other sites. This definition holds both in 2D

- **Galilean invariant, automatic refinement, good in capturing shocks and contact discontinuities**

— In a m**oving-mesh approach,** we consider the **gas flow through a MOVING mesh.**
—> What is done is to **transform a set of points into an unstructured mesh** using the Voronoi Tesselation scheme as you can see here in the image on the slide.
—> The **boundaries of the cells are drawn in such a way** that they are exactly halfway between neighbouring seeds/points shown in red.
—> The cells are allowed to **move with the fluid** like in Lagrangian schemes.
—> At the same time the **hydrodynamic solver is accurate:** it's done as in Eulerian schemes/grid codes, solving the Riemann problem at contact discontinuities.
—> Updated cell average quantities are transformed back into a particle scheme and particles/cells are moved to a new position with a new velocity.

— Big **advantange** is that this is scheme
—>is fully **Galilean invariant** thanks to the different geometry of moving mesh, and not having cartesian grid;
—>has an **automatic refinement** as each particle is represented by a cell, meaning in dense regions, cells are automatically smaller and more numerous;
—>is excellent for **shocks/contact discontinuities**: due to exact Riemann solver a grid interfaces.

Moving mesh hydrodynamics



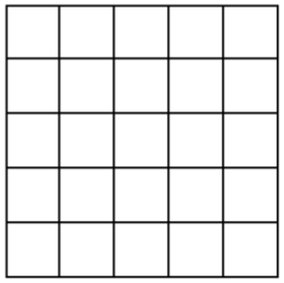
- Thanks to automatic refinement in Arepo, better performance for Rayleigh-Taylor instabilities

— Here you can also see that the moving-mesh codes perform better for the Rayleigh-Taylor instabilities compared to fixed, cartesian-mesh codes.
—> In the latter, the directionality of the flow follows the main axes of the grid and prevents chaotic fluid flows visible for moving-mesh codes.

Summary: Eulerian vs Lagrangian methods

Eulerian methods

discretise space
finite-volume scheme



use a grid fixed in space

Pro

Accurate hydro

Con

Not galilean invariant
No conservation of angular momentum
Diffusive
Limited spatial resolution
alleviated in AMR

Moving mesh

discretise space
finite-volume scheme

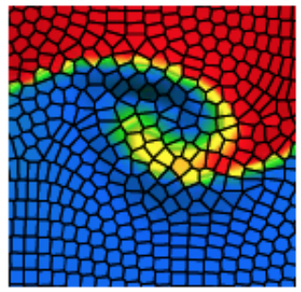


image credit: V. Springel

uses an unstructured mesh moving with the flow

Accurate hydro, automatic refinement on density, exactly Galilean invariant

Overhead ~30% for mesh construction

Lagrangian methods

discretise mass

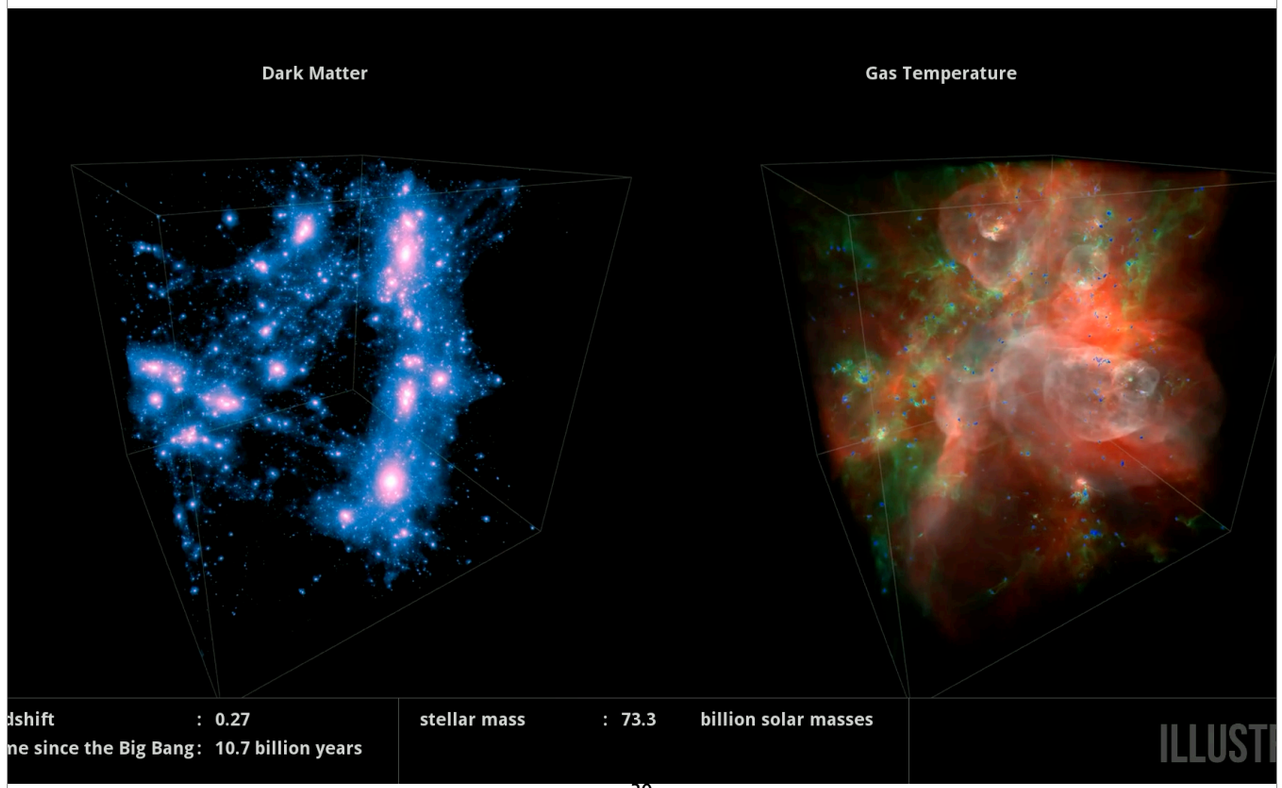
use particles for the gas (like in N-body) which move with the flow

Automatic refinement on density, exactly Galilean invariant, conserves angular momentum (& entropy) exactly
Somewhat less accurate hydro (contact disc.), slower convergence
But alleviated in modern SPH

—Overall, **moving-mesh schemes combine the advantages of both Lagrangian and Eulerian techniques and avoid the problems of each technique, it is numerically more stable and converges rather quickly**.

—> The only **disadvantage we have for moving-mesh codes is an overhead for the computation of the mesh construction (i.e. the Voronoi Tesselation).**

Famous example simulation: Illustris



Here you can see one of the **most famous cosmological simulations (Illustris) conducted with a moving-mesh code, called AREPO**.

—> Gas follows the DM, gas color-coded by T.
—> In the densest regions, gas accumulates, cools and forms stars.
—> Flickering of gas content is first due to stellar fb, and
—> later-on, once BHs get active, AGN feedback is visible as big explosions, expelling gas from galaxies and heating the ambient halo gas.

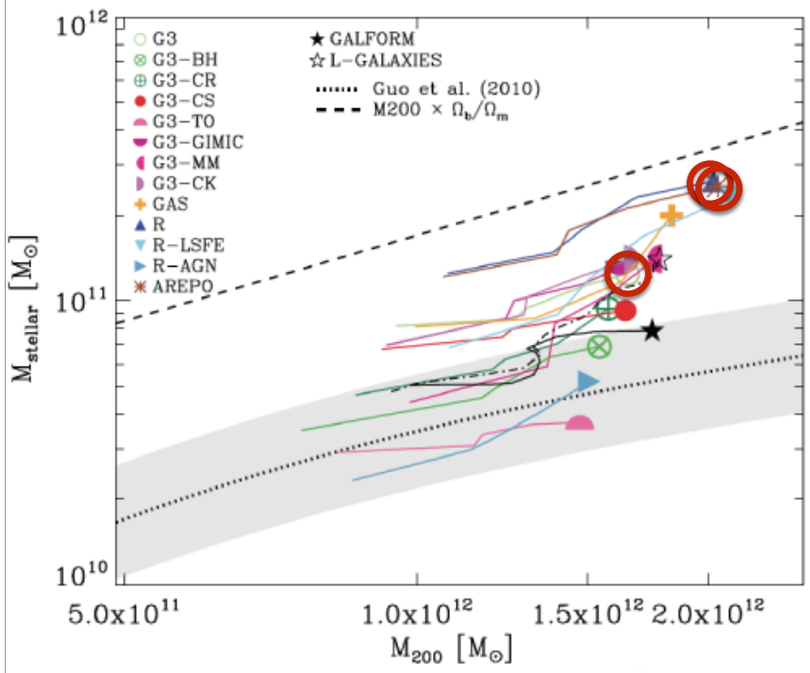
After…

having discussed these different and complex hydrodynamic schemes,

how relevant are they compared to other models for baryons (SF, feedback etc.)?

Relevance of hydro vs baryon physics

Aquila comparison project (Scannapieco et al.)



- Results from different hydrodynamic codes and feedback implementations show a great diversity
- Changes in the feedback model are more dramatic than from different hydrodynamic schemes!

— There have been some **comparison projects**, such as the Aquila comparison project,
—> in which different numerical codes were run with different sub-resolution and different hydrodynamic schemes over the same initial conditions of a few typical MW-type halos.
—> In this plot you can see stellar mass versus halo mass M200**.**
—> Red circles indicate three different hydro schemes,
—> Other points are related to these hydrodynamic schemes, but with various different models for SF and stellar feedback

— The two key **take-away messages** are the following:
—> 1. Results from different numerical codes and feedback implementations show a large diversity.
—> 2. Changes in the sub-grid models for SF & feedback are more dramatic than for different hydrodynamic schemes.

— Of course, it is important to employ a precise as possible hydrodynamic numerical scheme, but the main uncertainty in current hydrodynamic galaxy simulations comes **from sub-resolution models like stellar feedback**!

Summary -- Chapter 13

- The evolution of gas in the Universe can be most accurately modelled via hydrodynamic simulations accounting for (self-)gravity
- Individual gas particles/cells are treated collectively as a continuous medium with well-defined macroscopic properties (density, pressure, velocity, & energy)
- System described by Euler equations (mass, momentum and energy conservation) in Eulerian and Lagrangian forms
- Different numerical schemes exist to solve these equations

  —> adaptive mesh, SPH, moving mesh [mesh-less] codes

  —> each of them has their advantages and disadvantages, but moving-mesh codes combine advantages

Up next…

- *Chapter 1*: Introduction (galaxy definition, astronomical scales, observable quantities — repetition of Astro-I)
- *Chapter 2:* Brief review on stars
- *Chapter 3*: Radiation processes in galaxies and telescopes;
- *Chapter 4:* The Milky Way
- *Chapter 5*: The world of galaxies I
- *Chapter 6*: The world of galaxies II
- *Chapter 7:* Black holes and active galactic nuclei
- *Chapter 8:* Galaxies and their environment;
- *Chapter 9:* High-redshift galaxies
- *Chapter 10*:
  - Cosmology in a nutshell; Linear structure formation in the early Universe
- *Chapter 11*:
  - Dark matter and the large-scale structure
  - Cosmological N-body simulations of dark matter
- *Chapter 12*: Populating dark matter halos with baryons: Semi-empirical & semi-analytical models
- *Chapter 13:* Modelling the evolution of gas in galaxies: Hydrodynamics
- *Chapter 14*: Gas cooling/heating and star formation
- *Chapter 15*: Stellar feedback processes
- *Chapter 16*: Black hole growth & AGN feedback processes
- *Chapter 17*: Modern simulations & future prospects

Part I: Observational basics & facts of galaxies first 7 lectures

Part II: Theory & models of galaxy evolution processes second 7 lectures

Today's lecture …