Fluids (Navier Stokes)

named after **Claude-Louis Navier** and **George Gabriel Stokes**

FLUIDS = LIQUIDS || GASES

STABLE FLUIDS

Most engineering tasks require that the simulation provide accurate bounds on the physical quantities involved to answer questions related to safety, performance, etc. The visual appearance (shape) of the flow is of secondary importance in these applications. In computer graphics, on the other hand, the shape and the behavior of the fluid are of primary interest, while physical accuracy is secondary or in some cases irrelevant. Fluid solvers, for computer graphics, should ideally provide a user with a tool that enables us to achieve fluid-like effects in real-time. These factors are more important than strict physical accuracy, which would require too much computational power.

There are two main approaches for solving fluids,

1. Solving using Smooth Particle Hydro Dynamics by using a bunch of particles.
2. By using a 3D grid of cubes called voxels – Most methods use this approach.

Computing the motion + rendering the fluids = makes Fluid solving hard

If in 3D,

Computing the + motion in 3D	evolving the surface of the fluid dynamically	= Lot of data

In the above case, if the fluid is a "gas" then the task will be easier because gases do not require any surface to be computed dynamically

POINTERS

- This is a solver to update both flow and the motion of densities within the flow

- Stable fluids is an approach to solve the Navier-Stokes equations using both Lagrangian and implicit methods.

- Navier strokes equations are not the actual equations for fluids

- Damping in fluids is termed as "VISCOSITY"

  Honey – highly viscous

  Water – Normal viscous

- There is no closed approach for simulating fluids

Ex: Consider a 3D fluid simulated using the cubes as the data structure,

1000 x 1000 x 1000 cubes is Normal

5000 x 5000 x 5000 cubes is too large

We require about 8GB of memory to store the positions and if we need to store velocities also, then we need 8GB x 3 = 24GB of memory.

SIMULATING A BUNCH OF PARTICLES INSIDE VOXEL BASED FLUID SIMULATION

- Particles have no "mass" in this case.
- Knowing the velocities, drop the particles inside the fluid and the particles will just follow the velocity of the fluid.
- "Particles == fluid" => if a group of particles are trying to move out of the fluid surface, then it means that we can witness a "**bulging**" on the surface of the fluid.

DENSITY OF FLUID

When pressure is applied, fluid finds every pore around it and moves inside the pore. If there are no pores around the fluid then it is impossible to compress a fluid to reduce its volume.

Therefore "**Volume of a fluid is preserved**"

Imagine a reddish colored donut shaped chunk of water inside a bigger water chunk present in a vessel. When the water moves, the particles of the donut shaped chunk gets scattered but the mass of the whole water in the vessel remains constant.

Therefore "**Mass is preserved**"

Hence, density is preserved or "**DENSITY IS CONSTANT**"

NAVIER STROKES EQUATIONS

The equations are derived from the basic principles of conservation of mass, momentum, and energy

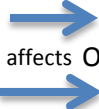The equation of incompressible fluid flow,

- Fluid solver - "**Lot of data and Lot of computation**"

- **Coupling**

    1-way coupling

    Object $\xrightarrow{\text{affects}}$ Fluid

    2-way coupling

    Object $\xrightarrow{\text{affects}}$ Fluid

    Fluid $\xrightarrow{\text{affects}}$ Object

- ∇ is different in this case,

    ∇ = partial derivative with respect to x , y and z

- U = velocity field

    "**At any moment velocity field varies very smoothly**"

- Velocity field is not uniform – it is of completely different velocities because it is fluid

$$\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} = -\frac{\nabla P}{\rho} + \nu \nabla^2 \mathbf{u},$$

equation (1)

The Navier Stokes equations are "Partial Differential Equations"
If you are given the following information about the system,

- The fluid's viscosity (remember viscosity represents the internal frictional forces of the fluid), and

- The initial conditions of the system (the velocity of the fluid at every point in the system at a certain time t=0)

You let the fluid simulate, defined by the Navier Stokes equations which describes fluid flow. Liquid with a velocity pushing upwards, for example, will displace other liquid above it and this will in turn displace other fluid, etc., and the whole system is churning.

Solving the Navier Stokes equations, you can

- Get the pressure of the fluid at any point in the system, and any time in the future, and also

- Get the velocity of the fluid at any point in the system, and any time in the future.

In the above Navier Stokes equation, the unknowns are the velocity "v" and the pressure "P". In this case we are having three equations and four unknowns. In order to solve this system we need a supplementary equation. This equation is the continuity equation, also called as incompressibility equation.

$$\nabla \cdot \mathbf{v} = 0.$$

equation(2)

Fluid is said to be incompressible if the volume of fluid "**passing in**" through a unit area is same as the volume of the fluid "**passing out"** of the same unit area in unit time.

In equation 1,

- Three scalar velocities and the pressure are the 4 unknowns in eqn(1)

- Pressure distribution in the fluid is not constant

- Pressure = Scalar field   field
  Velocity = Vector field   field

- In fluids, all the equations are for the **velocities** and not for the **positions**.

$\mathbf{u} \cdot \nabla \mathbf{u}$ ⟹ is responsible for the fluid flowing

$v \nabla^2 \mathbf{u}$ ⟹ If the velocities of the neighboring cubes are different, this part of the equation is responsible for increasing/decreasing the velocities of the cubes and making them to be at the same velocity.

$-\dfrac{\nabla P}{\rho}$ ⟹ is responsible for making the fluid flow from the higher pressure location to the lower pressure location.

If a user force is applied or a force due to an external object has acted upon the fluid, then the $\dfrac{\partial \mathbf{u}}{\partial t}$ increases. The velocities of the fluid system are changed dynamically to maintain $\nabla \cdot \mathbf{v} = 0.$

If velocity is so high that fluid crosses more than one cell in one time step, then fluid is said to be unstable. In such a case we can find the position of the fluid in the previous time step and copy the velocity of that point at that time step into the current unstable point in the fluid.

****