Scribe notes: Fluids (Navier-Stokes) (March 9, 2011)

CS599: Physically based Modeling for Interactive Simulation and Games.

-By Rohit Vishal Kotian.

Fluid simulation is an important and popular tool in computer graphics for generation realistic animation of water, smoke, fire, etc. In order to simulate the required effect we use the famous Navier-Stokes equations which is nothing more than good old Newton's equation F = ma in disguise.

The Navier-Stokes equation helps us to understand how fluids accelerate due to the forces acting on it.

The Equations:

The main flow for the fluids is governed by the two equations:

1. Momentum equation.

$$\frac{\partial \vec{u}}{\partial t} + \vec{u} \cdot \nabla \vec{u} + \frac{1}{\rho} \nabla p = \vec{g} + \nu \nabla \cdot \nabla \vec{u}$$

2. Incompressibility condition.

$$\nabla \cdot \vec{u} = 0$$

Symbols used:

u: defines the velocity of the fluid.

p (Greek letter): defines the density of the fluid.

p: stands for pressure.

g: acceleration due to gravity.

v (Greek letter): Kinematics viscosity. It measures how viscous the fluid is.

1. Momentum equation.

The first equation as the name suggests ensures that the momentum of the fluid be preserved. The equation is normally divided into 4 different parts. Advection (or sometimes *Convection* or *Transport*; they all mean the same thing), viscosity, pressure and the body force part.

2. Incompressibility condition.

On the other hand, the second equation deals with the fluid being "**Incompressible**", which means the volume of the liquid does not change. Real fluid however tend to change its volume otherwise you won't be able to hear underwater. But for animation purposes we tend to ignore this part as it is practically irrelevant. Thus we are invariably turning this into a constraint as that for a rigid body. Now if we take any arbitrary chunk of fluid as some instant of time, we can find how fast the volume of this chunk is changing by integrating the normal component of its velocity around the boundary.

$$\frac{d}{dt}\text{Volume}(\Omega) = \iint_{\partial\Omega} \vec{u} \cdot \hat{n}$$

But for an incompressible fluid this rate has to be a zero.

We use Divergence Theorem to change this into volume integral. Now, the magical part: this equation should be true for any choice of Ω (any region of fluid). The only function that integrates to zero independent of the volume of integration is zero itself. Thus the integrand has to be zero everywhere:

$$\nabla \cdot \vec{u} = 0$$

This is the "**Incompressibility condition**", the other part of Navier-Stokes equations.

A vector-field that satisfies the incompressibility condition is called "divergence-free" for obvious reasons. One of the tricky parts of simulating incompressible fluids is making sure that the velocity field stays divergence-free.

We use Advection only on such "divergent-free" vector field. However when we use an arbitrary vector field w this may not hold true. Thus we have to implement Helmholtz-Hodge Decomposition on w. This leads to the following equation:

$$\vec{w} = \vec{u} + \nabla q$$

This makes the equation divergent free, where q is a scalar field.

Now in order to get the "**Poisson equation**" we simply multiply both sides by the gradient operator. Thus we get:

$$\nabla \cdot \vec{w} = \nabla \cdot \nabla q$$

We use the velocity computed from the previous time step in order to ensure that the incompressibility term holds true. Then we use Poisson equation to get the pressure which helps us to adjust the velocity field thereby helping us to get the next step in the fluid simulation. This also maintains the divergent free property.

Boundary Conditions:

An important part of fluid simulation is to get the boundary conditions right. In this note we will only cover two boundary conditions.

1. Solid walls.
2. Free surfaces.

An important case which is not covered is the boundary between two different fluids; most often important for animation, but if interested you can view paper mentioned in the reference [4].

A solid wall boundary is one in which a fluid is in contact with a solid surface. In a much simpler term the fluid will not be flowing out of the solid or even in. In terms of velocity, the normal component of the velocity has to be zero.

$$\vec{u} \cdot \hat{n} = 0$$

That is, if the solid isn't moving itself. In general, we need the normal component of the fluid velocity to match the normal component of the solid's velocity.

$$\vec{u} \cdot \hat{n} = \vec{u}_{\text{solid}} \cdot \hat{n}$$

In both these equations, ˆn is of course the normal to the solid boundary. This is sometimes called the "no-stick" condition, since we're only restricting the normal component of velocity, allowing the fluid to freely slip past in the tangential direction.

The other boundary condition that we're interested in is the free surface. This is where we stop modeling the fluid. For example, if we simulate water splashing around, then the water surfaces that are not in contact with a solid wall are free surfaces.

Conclusion:

To sum the whole fluid simulation:

We use advection, Viscosity and Body forces and get a divergent free velocity vector. We use this in conjunction with the location of particle (inside the fluid or on the surface) and use the pressure solver to get the next velocity for the next time step.

References:

1. J. Stam: Stable Fluids, SIGGRAPH 1999
2. Fluid Simulation for Computer Animation, Robert Bridson and Matthias Müller-Fischer
3. http://en.wikipedia.org/wiki/Fluid_simulation
4. Jeong-Mo Hong and Chang-Hun Kim. Discontinuous fluids. ACM Trans. Graph. (Proc. SIGGRAPH),24:915–920, 2005.