Smoke Simulation using Smoothed Particle Hydrodynamics (SPH)

Shruti Jain
MSc Computer Animation and Visual Effects
Bournemouth University

21st November 2014

Abstract

This report is based on the implementation of Smoothed particle hydrodynamics(SPH) method for smoke simulation. SPH is a mesh free Lagrangian method for free surface fluids, where the fluid is divided into discrete elements called particles. Particles being a respresentation of mass, no extra calculations are needed for conservation of mass. Pressure and viscosity are derived using the Navier-Stokes equation. Radial symmetrical smoothing kernels operate on the particles which lie within the kernel range to define the field quantity of a particle.

The overall motion is controlled using smooth fields, and the small scale behaviour is governed by the statistical parameters and scale of turbulent wind field, which adds complexity to the motion (Stam 1995).

Contents

List of Figures

1 Introduction

Realistic fluid simulation has been increasingly in demand in the Visual effects industry since decades. Fluid simulation which is based on principles of fluid mechanics, can be used for simulating effects like smoke, water, fire etc. Various approaches, ranging from usage of 2D particle-based systems to high quality rendering systems based on complex algorithms, have been implemented to achieve realistic outcomes.

There has been a remarkable achievement in the techniques which were previously used for simulating effects in comparison to the current advancements. The modern techniques did not only overcome the shortcomings of the earlier methods, but proved to be more efficient and made simulation look more realistic.

Various methods are used to approximate the behaviour of smoke, of which the basic and most popular method is based on Navier-Stokes equations. The Eulerian method is suitable for fluids confined within a rectilinear volume, which is further divided into a grid of cubicle cells where each cell stores information about velocity, pressure, temperature etc. Lagrangian schemes (that is, schemes where the computational elements are not fixed in space) such as smoothed particle hydrodynamics (Müller, Charypar & Gross 2003) are also popular for fluid animation. The method is based on interpolation of particles and it uses them to directly render the fluid surface.

This assignment is based on the application of Smoothed Particle Hydrodynamics (SPH) method initially developed by Lucy (1977) and by Monaghan (1992). The calculations and implementation are mainly based on the research by Müller, Charypar and Gross (2003) and Monaghan (2005).

2 Related work

Stam (1999) proposed an unconditionally stable model based on finite difference of Navier-Stokes equations. His method was based on Foster and Metaxas (1997) work which overcame the existing instabilities, by the use of much larger time-steps. Fedkiw, Stam and Jensen (2001) presented a numerical method using the inviscid Euler equations, that overcame the problem of numerical dissipation by introducing a forcing term called "vorticity confinement" to add the lost energy back into the fluid. Müller, Charypar and Gross (2003) based their research on SPH method to simulate fluids with free surfaces than the usual grid-based techiniques. This method was earlier used by Stam and Fiume (1995) for fire and other gaseous phenomena. Losasso, Gibou and Fedkiw (2004) used the mesh refinement semi-Lagrangian characteristic tracing technique on unrestricted octree data structure. Area weighted averaging was used to calculate velocity and vorticity of the nodes. Mullen, Crane, Pavlov, Tong and Desbrun (2009), proposed fully Eulerian non-dissipative integration schemes with no numerical viscosity, which would preserve energy over long time scales for inviscid fluids. However, the approach did not eliminate diffusion since vorticity diffusion is unavoidable in a purely Eulerian context (Mullen et al. 2009).

3 Technical background

There are different methods for simulating smoke and the popular ones are described in this section. The implemented (SPH) method will be discussed in detail in section 4 and is not included in this section.

3.1 Navier-Stokes equation

There is a consensus among scientists that the Navier-Stokes equations are a very good model for fluid flow (Stam 1999). When the velocity and pressure are known for initial state for some initial time $\mathbf{t} = 0$, the Navier-Stokes equations are obtained by imposing that the fluid conserves both mass Eqn.(1) and momentum Eqn.(2) (Stam 1999).

Conservation of mass

The below equations shows that mass should always be conserved and that the velocity field of an incompressible flow should not have divergence.

$$\nabla.\mathbf{u} = 0 \tag{1}$$

u = represents a velocity vector field
∇= vector of spatial partial derivatives i.e ($\partial/\partial x$, $\partial/\partial y$,$\partial/\partial z$)

Conservation of momentum

This equation couples the velocity and pressure fields and relates them through the conservation of momentum (Foster & Fedkiw 2001). This equation describes fluid motion in terms of the local velocity **u** which is a function of position and time.

$$\frac{\partial \mathbf{u}}{\partial t} = -(\mathbf{u}.\nabla)\mathbf{u} - \frac{1}{\rho}\nabla p + \nu \nabla^2 \mathbf{u} + \mathbf{f} \tag{2}$$

p = pressure field
ν = kinematic viscosity of the fluid
ρ = density of the fluid
"." = dot product between vectors
$\nabla^2 = \nabla.\nabla$

3.1.1 Equation terms

Advection

The term$-(\mathbf{u}.\nabla)\mathbf{u}$ represents self-advection of the velocity field and is called the advection term (Harris 2007). Advection shows the force exerted on a particle by the surrounding particles.

Pressure

The term -$\frac{1}{\rho}\nabla p$ is the pressure term and represents acceleration. When force is applied to a fluid, it does not instantly propagate through the entire volume, instead the molecules close to the force push on those farther away, and pressure builds up (Harris 2007).

Diffusion

The term $\nu\nabla^2\mathbf{u}$ represents the viscosity, which is a measure of the resistance for the flow of the fluid. This factor indicates dampness and decides fluid's viscosity. This resistance results in diffusion of the momentum (and therefore velocity) (Harris 2007).

External forces

The term $\mathbf{f}$ represents acceleration due to external forces. These forces may be either local forces or body forces (Harris 2007). Forces like pressure and viscosity act locally on the surface, the magnitude of this force will vary from one region of the fluid to another. Gravity force can be considered as a body force which will be evenly applied to the entire fluid.

3.1.2 Equation operators

Gradient

$$\nabla p = (\frac{\partial p}{\partial x}, \frac{\partial p}{\partial y}, \frac{\partial p}{\partial z}) \tag{3}$$

Gradient is defined by the direction and rate at which the fluid moves from high-pressure to low-pressure area. The gradient of a scalar field is a vector of partial derivatives of the scalar field (Harris 2007).

Divergence

$$\nabla.\mathbf{u} = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} \tag{4}$$

Divergence is applied to the velocity of the fluid to measure net change in the velocity. It represents the outflow of volume density from an infinitesimal volume surrounding the point. In the equation for conservation of mass, also known as the continuity equation, enforces the incompressibility assumption by ensuring that the fluid always has zero divergence (Harris 2007).

Laplacian

$$\nabla^2 = \frac{\partial^2}{\partial x^2} + \frac{\partial^2}{\partial y^2} + \frac{\partial^2}{\partial z^2} \tag{5}$$

Laplacian is a second order differential operator resulting from divergence being applied to the gradient. It represents the fluid density of the gradient flow and results into diffusion.

3.2 Euler equations

Smoke can be modeled as incompressible and inviscid. The effects of viscosity are negligible in gases especially on coarse grids where numerical dissipation dominates physical viscosity and molecular diffusion (Fedkiw, Stam & Jensen 2001). The Eulerian method is based on dividing the volume into identical voxels as shown in Figure 1.

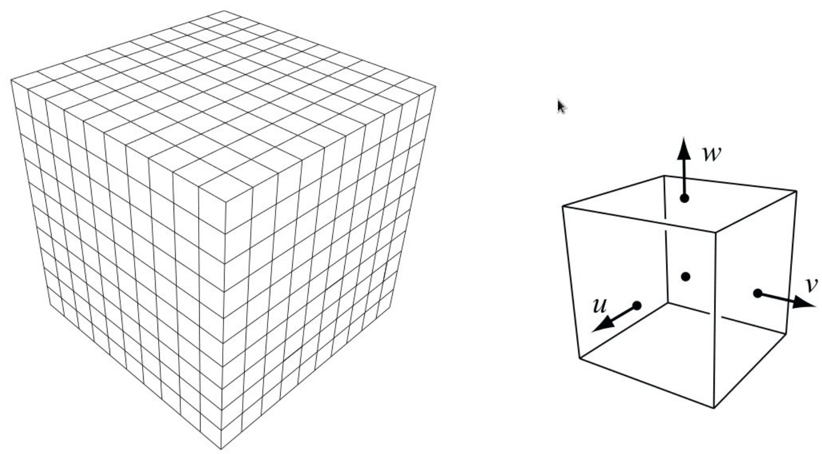

Figure 1: Discretization of domain into identical voxels (left). Velocity components defined on the faces of each voxel (right).

Euler equations correspond to the Navier-Stokes equations with zero viscosity and no heat conduction. The inviscid incompressible Euler equations can be represented as:

$$\nabla.\mathbf{u} = \mathbf{0} \tag{6}$$

$$\frac{\partial \mathbf{u}}{\partial t} = -(\mathbf{u}.\nabla)\mathbf{u} - \nabla p + \mathbf{f} \tag{7}$$

The terms of these equations are similar to those for the Navier-Stokes equations described in section 2.1.

Density and Temperature

The other factors that affect the velocity of the fluid particles are smoke's density ρ and temperature T. Temperature is inversely proportional to the density, and increase in temperature will cause a decrease in the density and vice versa. The equations are based on the assumption that these two scalar quantities are simply moved (advected) along the smoke's velocity (Foster & Fedkiw 2001). For velocity u = (u,v,w), the equations would be:

$$\frac{\partial T}{\partial t} = -(u \cdot \nabla)T \tag{8}$$

$$\frac{\partial \rho}{\partial t} = -(u \cdot \nabla)\rho \tag{9}$$

3.3 Vortex particle method

This hybrid technique is a combination of Lagrangian vortex particle method and Eulerian grid based method. Vorticity confinement reintroduces the small scale detail lost when using efficient semi-Lagrangian schemes for simulating smoke and fire (Selle 2005).

The Navier-Stokes equations can be put into vorticity form by taking the curl of equation (2) to obtain the equations below:

$$\omega_t + (u.\nabla)\omega - (\omega.\nabla)u = \mu\nabla^2\omega + \nabla \times f \tag{10}$$

u = velocity,
f = represents buoyancy, vorticity confinement etc.
ω= vorticity value at each vortex particle

The pressure term is not present for constant density fluids. Although these equations can be solved on a grid, particle based methods have the distinct advantage of avoiding grid based numerical dissipation that smears out the flow making it appear more viscous (Selle 2005).

3.3.1 Equation terms

Vorticity

The symbol ω represents vorticity value stored at each vortex particle, which includes both magnitude and direction. The vorticity confinement force is computed by taking the curl of the velocity field to obtain the vorticity (Selle 2005):

$$\omega = \nabla \times u \tag{11}$$

For compressible fluids the magnitude of vorticity is higher. The vorticity at a point is defined by summing the contributions from all nearby particles (Selle 2005).

Vorticity advection

The term $(u.\nabla)\omega$ represents vorticity advection. This is the trilinear interpolation used to define a velocity of advecting each particle for a given velocity, u, determined via the grid based method (Selle 2005).

Vorticity stretching

The term $(\omega.\nabla)u$ represents vorticity stretching. This term represents the enhancement of vorticity by stretching and is the mechanism by which the turbulent energy is transferred to smaller scales.

Dissipation

The term $\mu\nabla^2\omega$ is responsible for strong dissipation of large velocity gradients. However, the goal in using particle based methods is to eliminate dissipation, so this term is ignored for solving the inviscid form of the the equations (Selle 2005).

4 Implementation method

SPH method is an approach based on particles where the properties are stored at particle level. The calculations are based on Lagrangian method whereas the force density field is derived from the Navier-Stokes equation. Some characteristics, like pressure, mass and density, are tightly coupled to each other that highly impact the fluid's velocity and appearance.

The overall motion is controlled using smooth fields, and the small scale behavior is governed by the statistical parameters and scale of turbulent wind field, which adds complexity to the motion (Stam 1995).

According to SPH interpolation rule, a scalar quantity A is interpolated at location r by a weighted sum of contributions from all particles (Müller, Charypar & Gross 2003):

$$A_S(r) = \sum_j m_j \frac{A_j}{\rho_j} W(r - r_j, h) \tag{12}$$

j = iterator over all particles
m_j= mass of the particle
r_j= Position of the particle
ρ_j= Density
A_j= The field quantity at r_j

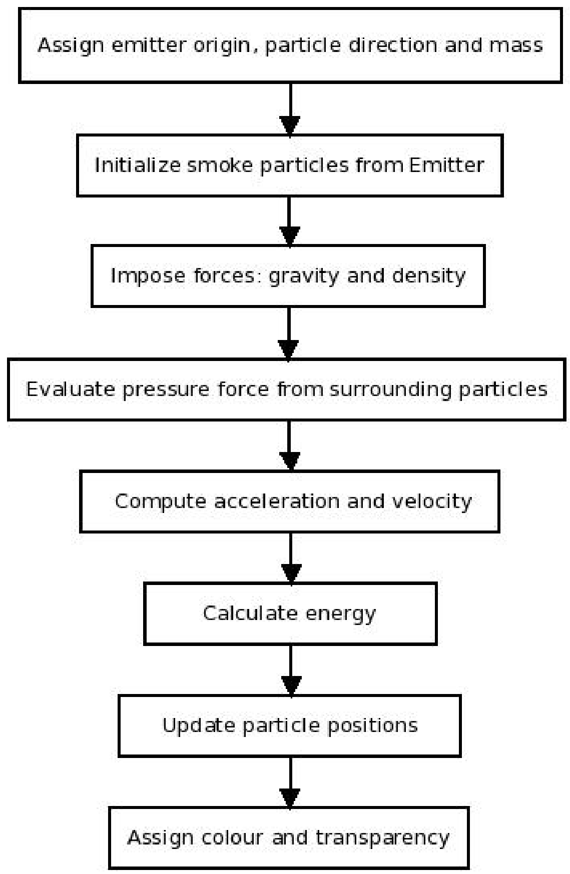

Figure 2: Process flow diagram

The high-level steps for implementation of the approach is given by Figure 2.

4.1 Gradient and Laplacian

With the SPH approach, the derivatives gradient and Laplacian will only affect smoothing kernel (Müller, Charypar & Gross 2003).

The gradient of A can be denoted as:

$$\nabla A_S(r) = \sum_j m_j \frac{A_j}{\rho_j} \nabla W(r - r_j, h) \tag{13}$$

The Laplacian of A can be evaluated as:

$$\nabla^2 A_S(r) = \sum_j m_j \frac{A_j}{\rho_j} \nabla^2 W(r - r_j, h) \tag{14}$$

4.2 Density

Considering the mass m_i is constant and same for all the particles, the density ρ_i varies which needs to be evaluated for every time step. Substituting into Eqn.(12), the density at location r can be evaluated (Müller, Charypar & Gross 2003).

$$\rho_S(r) = \sum_j m_j \frac{\rho_j}{\rho_j} W(r - r_j, h) = \sum_j m_j W(r - r_j, h) \tag{15}$$

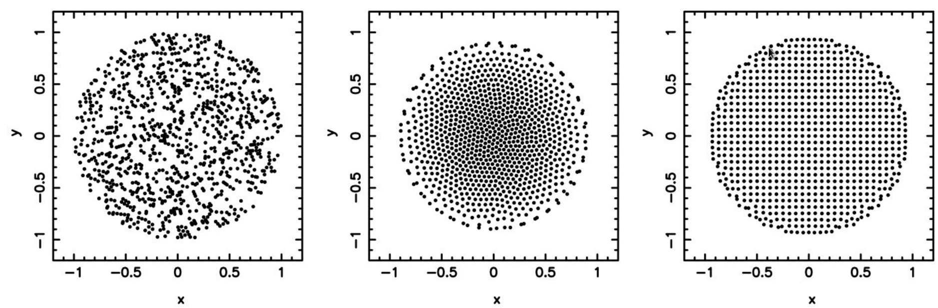

Figure 3: Density of SPH particles: Left frame shows placement according to a constant probability density. Middle frame shows the positions of equal mass particles settled down in a Toy star potential starting with particle positions in the left frame. Right frame shows variable mass SPH particles settled down in Toy star potential where cubic spline kernel was used. (Monaghan 2005)

The density distribution for different compositions is shown in Figure 3.

4.3 Pressure

The pressure p on the particle is calculated by the surrounding particles. By applying the SPH rule of Eqn.(12) to the pressure term $-\nabla p$ from Navier-Stokes Eqn.(2) yields (Müller, Charypar & Gross 2003):

$$\mathbf{f}_i^{pressure} = -\nabla p(\mathbf{r}_i) = -\sum_j m_j \frac{p_j}{\rho_j} \nabla W(r_i - r_j, h) \tag{16}$$

Pressure force is the pressure applied to one particle by another. Since, the pressure at different locations will be different, the pressure force for each particle will be different This force is asymmetric and to calculate symmetric pressure force for particles i and j, the arithmetic mean of the pressures of interacting particles is used (Müller, Charypar & Gross 2003).

$$\mathbf{f}_i^{pressure} = -\sum_j m_j \frac{p_i + p_j}{2\rho_j} \nabla W(r_i - r_j, h) \tag{17}$$

4.4 Acceleration

The acceleration is used to calculate the velocity and position of the particle. The acceleration equation which conserves linear and angular momentum can be defined as (Monaghan 2005):

$$\frac{\nabla p}{\rho} = \nabla\left(\frac{p}{\rho}\right) + \frac{p}{\rho^2}\nabla\rho \tag{18}$$

The above equation can be re-defined by using the SPH interpolation rule Eqn.(12) (Monaghan 2005):

$$\frac{dv_a}{dt} = -\sum_j m_j \left(\frac{p_j}{\rho_j^2} + \frac{p_i}{\rho_i^2}\right) \nabla_i W(r_i - r_j, h)$$

4.5 Energy

The thermal energy can be defined per particle, and can be correlated with the temperature and heat within the system. The time rate of change of thermal energy is (Monaghan 2005):

$$\frac{du}{dt} = \frac{p}{\rho^2}\frac{d\rho}{dt} = -\frac{p}{\rho^2}\nabla \cdot v$$

By using the SPH form Eqn. (12) , the equation can be defined as (Monaghan 2005):

$$\frac{du_i}{dt} = \frac{p_i}{\rho_i^2}\sum_j m_j v(r_i - r_j, h) \cdot \nabla_i W(r_i - r_j, h)$$

4.6 Smoothing color

The color field of the particle should be correctly defined, a color value between 0 and 1 can be used to define the intensity of the particle. The color needs to be properly adjusted in accordance with the surrounding particles, as there cannot be a particle with color value 1 with an adjacent particle of 0 value for color. Color should gradually fade out. The equation for the smoothed color is given by (Müller, Charypar & Gross 2003):

$$c_s(\mathbf{r}) = \sum_j m_j \frac{1}{\rho_j} W(r - r_j, h) \tag{19}$$

$$\mathbf{f}^{surface} = \sigma\kappa\mathbf{n} = -\sigma\nabla^2 c_s \frac{n}{|\mathbf{n}|} \quad (20)$$

$\mathbf{n} = \nabla c_s$, gradient of the smoothed color field represents the surface normal
$\kappa = \frac{-\sigma\nabla^2 c_s}{|\mathbf{n}|}$, divergence of n measures surface curvature

4.7 External forces

External forces such as gravity and collision forces are directly applied to the particles.

5 Conclusion and future work

The mentioned approach will be implemented and is expected to yield a reasonable accurate outcome. The advantages of using the SPH approach are that preserving energy and linear momentum are easily achievable. Whereas the Eulerian method does not conserve angular momentum, SPH does. SPH approach guarantees conservation of the total energy when combined with self-gravity, which is not the case in mesh-based approaches. The major disadvantage of SPH approach is the lack of accuracy for multi-dimensional flow. As the impact of noise on particles is calculated by the particles closely surrounding it, the repulsive forces may not get settled in all dimensions, which may cause jitter. As a further improvement, SPH method can be combined with other methods to overcome the problem of multi-dimensional flow. Also, the SPH method can be implemented on the GPU to make the simulation faster and provide a real-time high quality render.

References

A. Selle, N. Rasmussen, and R. Fedkiw, "A vortex particle method for smoke, water and explosions," in *ACM SIGGRAPH 2005 Papers*, SIGGRAPH '05, (New York, NY, USA), pp. 910-914, ACM, 2005.

F. Losasso, F. Gibou, and R. Fedkiw, "Simulating water and smoke with an octree data structure," in *ACM SIGGRAPH 2004 Papers*, SIGGRAPH '04, (New York, NY, USA), pp. 457-462, ACM, 2004.

J. J. Monaghan, "Smoothed particle hydrodynamics," *Annual review of astronomy and astrophysics*, vol. 30, pp. 543-574, 1992.

J. J. Monaghan, "Smoothed particle hydrodynamics," *Reports on progress in physics,* vol. 68, pp. 1703-1759, Jul. 2005.

J. Stam, "Stable fluids," in *Proceedings of the 26th annual conference on Computer graphics and interactive techniques*, SIGGRAPH '99, (New York, NY, USA), pp. 121-128, ACM Press/Addison-Wesley Publishing Co., 1999.

J. Stam and E. Fiume, "Depicting fire and other gaseous phenomena using diffusion processes," in *Proceedings of the 22nd annual conference on Computer graphics and interactive techniques*, SIGGRAPH '95, (New York, NY, USA), pp. 129-136, ACM, 1995.

L. B. Lucy, "A numerical approach to the testing of the fission hypothesis," *Astronomical Journal,* vol. 82, pp. 1013-1024, Dec. 1977.

M. J. Harris. (2007, Sep). *Chapter 38. Fast Fluid Dynamics Simulation on the GPU (5th ed.)* [Online]. Available: http://http.developer.nvidia.com/GPUGems/gpugems_ch38.html

M. Müller, D. Charypar, and M. Gross, "Particle-based fluid simulation for interactive applications," in *Proceedings of the 2003 ACM SIGGRAPH/Eurographics symposium on Computer animation*, SCA' 03, (Switzerland, Switzerland), pp. 154-159, Eurographics Association Aire-la-Ville, 2003.

N. Foster and D. Metaxas, "Modeling the motion of a hot, turbulent gas," in *Proceedings of the 24th annual conference on Computer graphics and interactive techniques*, SIGGRAPH' 97, (New York, NY, USA), pp. 181-188, ACM Press/Addison-Wesley Publishing Co., 1997.

N. Foster, and R. Fedkiw, "Practical animation of liquids," in *Proceedings of the 28th annual conference on Computer graphics and interactive techniques*, SIGGRAPH' 01, (New York, NY, USA), pp. 23-30, ACM, 2001.

P. Mullen, K. Crane, D. Pavlov, Y. Tong, and M. Desbrun, "Energy-preserving integrators for fluid animation," in *ACM SIGGRAPH 2009 Papers*, SIGGRAPH' 09, (New York, NY, USA), ACM, 2009.

R. Fedkiw, J. Stam, and H. W. Jensen, "Visual simulation of smoke," in *Proceedings of the 28th annual conference on Computer graphics and interactive techniques*, SIGGRAPH' 01, (New York, NY, USA), pp. 15-22, ACM, 2001.