Advanced Materials Research Vols. 268-270 (2011) pp 1326-1331
Online available since 2011/Jul/04 at www.scientific.net
© (2011) Trans Tech Publications, Switzerland
doi:10.4028/www.scientific.net/AMR.268-270.1326

Summary of Fluid Simulation Based on Physical Model

Jian Ni[a] ,Xifei Wei[b]

College of information and Electronic Engineering ,Hebei University of Engineering ,Handan, China
[a]nijianlaoshi@126.com,[b]weixifei123@163.com

Keywords: physical model;fluid animation ;Navier-Stokes equation

Abstract. Fluid animation based on physical model has produced a number of new research achievements ,and it has also become a hot spot of research in the field of computer animation in recent years.This paper presents a survey on the development of fluid simulation animation based on physical model,with Summarizing all kinds of methods adopted in the research direction.The methods applied mainly include Euler method and Lagrangian method which are compared in this paper. Finally ,this paper introduces the direction of future research.

Introduction

In recent years, people have been trying to use computer to realize the real world in the field of computer graphics, although the world look more simple, it is extremely complex. Along with the people of movies and games from the visual effect demand is higher and higher, naturally people want to see more actual fluid animation effect,however,the movement of fluid is extremely complex and the traditional methods of rendering fluid effect by art workers by frame have not workable.Then,the methods based on physical model become the main method of computer simulation and it has become a hot spot of research and an important direction.

The method based on physical model simulates fluid by solving the physical equation of the fluid motion state,mainly through numerical method in order to research Navier-Stokes equation. This paper mainly discusses is the basic method of solving the Stokes-Navier equation and compare these, phenomenon involved include smoke[1], waves[2], fire[3], explosion[4], bubble[5] and so on, and their simulation in Computation Fluid Dynamics (CFD) field is hot topic. CFD emphasizes the accurate solution of question itself and gives a reasonable analysis and requires that the results of computer simulation can predict fluid motion of reality,but fluid animation focuses on the fluid visual effect which is generated in order to cheat audience.So, compared to the commonly used method in CFD, arithmetic of fluid animation is lower in precision but it is faster in calculation speed. In a word,there are still a lot of topics[6] which are worth researching in fluid animation as an independent field of study.

In addition, drawing and rendering of fluid are also important aspects and it's a fairly important research direction in the field of fluid animation.Temperature accountes for a very important position and blackbody radiation can not be ignored for the phenomenon of the fire and explosion,for the phenomenon of room temperature[7] of smoke the effect can be fully realized by density field. Due to space reasons, the fluid drawing and rendering is not within the scope of this.

Physical Model-based Method for Fluid Animation

The last fluid simulation mainly used the method of parameter model which can show the real liquidity effect. But now we simulate the fluid motion by solving the Navier-Stokes equation which is constituted by the continuity equation, momentum equation and energy equation which are the most basic physical description for fluid motion.Their equation expression can be described as below:

(1) conservation of mass
(2) Newton second law
(3) conversation of energy

Although the equation of the form may be different as a result of different fluid description method (Lagrangian method and Euler method), but through some mathematical derivation, different forms of equation can be interchangeable.

Methods based on physics can be divided into two kinds: Euler method and Lagrangian method. Euler method is a grid-based approach,which starts to research from the space occupied by the fluid of each fixed point, and analyses each fixed point of the space which is filled with fluid movement on the fluid velocity, pressure, density and other parameters changing with time, and study the changes in these parameters when one point moves from one space to another space; Lagrangian method is based on the particle method,which starts to analyse the movement of each micro-group and this is mean that this method study the change of a given micro-group's parameters which contain speed,pressure and density over time,and which reseach these parameters' change when one fluid mico-group to other groups of fluid.

Euler Method. Because Eular method fixes observation point in space and considers the physical quantity of fluid (including density, temperature, speed) at different times of change, the whole fluid region can use a series of scalar or vector field to describe, that is, for a given time and location, there is a uniquely determined set of data corresponds with the fluid state. Such as the vector field $v(x, y, z, t)$ can be used to describe the velocity of the fluid, scalar field $p(x, y, z, t)$ used to describe the liquid pressure.Under the description of Euler the form of incompressible Navier-Stokes equations is as follows:

$$\nabla \cdot u = 0 \tag{1}$$

$$\frac{\partial u}{\partial t} = -(u \cdot \nabla)u + v\nabla^2 u - \frac{\nabla p}{\rho} + f \tag{2}$$

Among them, ρ is the density, p is the pressure, f is the volume, u is the speed, v is the sports coefficient , η is the dynamic viscosity coefficient,and the relation between v and η is $v = \eta / p$. Grid-based Euler method is that the above equations are discretized to the grid ,and gets the entire field through calculating state changes of each fixed grid nodes.there are two ideas to mesh: one is staggered grid that is the general scalar (such as pressure) distribution in the grid cell center, while the amount of velocity distribution in the cell surface, which is easy to ensure conservation conditions and this idea is often used[8]; the other is that the amount of all are in the same location[9] and this method is simple ,and does not require much interpolation, and need not be treated differently for each variable.

In the early research stage of fluid simulation based on physical model,the researchers do not usually deal directly with Navier-Stokes equations, but make reasonable assumptions and simplifications to the problem.Such as approximately simulating the fluid phenomenon by the use of wave equation[10] or the shallow water equations[11] and using a high degree of field to record surface location and so on. This method is simple and efficient, but can't capture many interesting three-dimensional phenomena, such as rolling waves, spray splash and so on. Fosterd[12], who is pioneer in the field of computer graphics for solving three-dimensional finite difference Navier-Stokes equations .Subsequently, Stam[13] introduced a semi-Lagrangian method for solving the convection of momentum equation and used the implicit method to calculate viscous and pressure, and finally obtained the unconditional stability of the numerical simulation. Now the method has become the standard Euler method framework of fluid animation, and the steps can be divided into physical, convection, viscous, and projection. Euler method and level sets based on surface tracking algorithm has achieved satisfactory results. MAC uniform grid[14−20] used to solve the Navier-Stokes equations to simulate a lot of fluid phenomenon, which includes smoke, fire, water and so on. In a word, Eular fluid simulation animation is true to the general audience can not distinguish between computer animation and real film of the stage.

Lagrangian Method. Lagrangian method is to simulate a continuous fluid medium into a discrete particle system.Particle systems are an irregular for the continuous dispersion medium.Because Reeve[21] introduced particle systems with a flexible performance capabilities for the irregular objects, they are used to simulate the flow, in addition to, appearing in the current splash and foam can be used to express the particle system. Each micro-group of the fluid is expressed by the particle which is location x and speed u. In the Lagrangian description, the incompressible Navier-Stokes equations are:

$$\nabla \cdot u = 0 \tag{3}$$

$$\frac{Du}{Dt} = v\nabla^2 u - \frac{\nabla p}{\rho} + f \tag{4}$$

Among them, u is the speed, v is the viscosity coefficient, ρ is the density, p is the pressure, f is the strength. Formula(4) is the momentum equation,and the left side of the formula for the particle acceleration and the right is the force acting on the particles. The equations have been simplified which is based on assumptions of incompressible fluid.In real life we can observe the phenomenon of fluid which are basically incompressible[22].

In the process of solving Navier-Stokes equations, the need for such irregular discrete particle systems to define the gradient operator and the Laplace operator, Monaghan[23] first introduced the SPH (Smoothed Particle Hydrodynamics) method to solve this problem, and simulated fire and other gaseous phenomenon. SPH is an interpolation method for particle system ,through introducing the smoothing kernel function to represent the impact of the surrounding particles, the method also be used to solve the large deformation of flexible objects, and now the method has become a popular method in the field of fluid animation. In recent years, literature[24] introduces LBM

(Lattice Boltzmann Model) into graphics field, LBM method is a Lagrangian method, which do not follow each of the actual particles. In the discrete grids the particles along the rail line to the adjacent lattice grid migration and collision with each other, so that the evolution of the distribution function determines the fluid process of change. the method relative to the Eular method has the advantage of easy programming, easier parallelism, and can easily handle complex boundaries.

Lagrangian method used particle system without using mesh is very suitable for irregular boundary conditions, as well as the interaction of a variety of mixed fluids. Usually, the Lagrangian method of computing resources required is relatively low, and not need to deal with the whole space, and ensures conservation of mass, and relatively easy to implement control.The method is widely used in real-time interaction, such as games, but for the reconstruction of a smooth interface compared intractable movement, freedom to change the interface topology Complex algorithms must be used to construct the surface geometry, so far, the simulation and the animation is not the true extent of grid-based simulation methods.

The Advantages and Disadvantages of Euler Method and Lagrangian Method. The main advantage of the Euler method is easy to reconstruct a smooth liquid surface and the larger time step, but it has the problem which are long computing time, boundary aliasing and poor extension.Lagrangian method makes use of traditional particle system to simulate fluid and easy to understand concepts and procedures easy to implement and very easy to catch with the small fluid details which are similar to bubble[25] and foam[26], and also easy to simulate turbulent water, but more importantly, SPH requirements for computing resources is far lower than the Euler method, therefore,the Lagrange approach is more applicable to real-time interactive areas, but it is difficult to select a smooth kernel function, and the method of using the ideal gas equation to contact the pressure and the air pressure can not strictly guarantee incompressibility of the fluid.

Euler method and Lagrangian method have their advantages and disadvantages. Euler method based on grid is often used with Lagrangian particle method in order to produce a more realistic simulation of fluid movement, which creates the widely used semi-Lagrangian algorithm.

The Direction of Future Research

Although the development of fluid animations based on physical up to now has been for many years,but there are still many issues which are worth deeply studying, and there are also many challenges that await researchers to overcome. For example, now there is no simulation system of real-time interaction, there is no real simulation results based entirely on physical model.Now this paper will introduce the possible future new research directions about fluid simulation based on physical model.

Capture the Details of Phenomena. In daily life, these small details such as the spray of water,small-scale vortex flow are all high frequency components of fluid motion.The most attractive in fluid animation is that these high-frequency details.According to Nyquist sampling theorem, only the sampling frequency is more twice than the maximum signal frequency which can be restored a complete signal without loss, however, a large number of particles or high-precision direct result of the grid is a linear increase in memory consumption and super-linear growth of computation time. Clearly, the simple way of increasing the sampling frequency is not feasible. Now the high-frequency details of fluid motion has become an integral part of generating real fluid animation. Papers in recent years has begun to tilt in that direction, but this is still a subject of no perfect solution.

Grid Generation Method. Mesh generation method in fluid animation based on physical model is a potential research direction.The quality of computational grid directly affects whether to obtain a more accurate results through numerical method solving Navier-Stokes equations, at the same time, good grid can properly disperses borders and captures the small-scale details.Grid generation in computational fluid dynamics has been an old subject, but it is still in infancy in graphics applications, so we can borrow many ideas and methods from the field of computer fluid to achieve the grid generation.

Numerical Viscosity. The contradiction between calculation accuracy and simulation speed and the contradiction between algorithm stability and numerical viscosity are the major contradiction in the field of fluid animation.The fluid animation tend to consider simulation speed and numerical stability as more important considerations.Therefore, the large time step, the low accuracy of the discretization scheme and the implicit method is widely used,but the introduction of numerical viscosity are inevitable. Numerical viscosity is also known as viscous dissipation, that is, the error of numerical algorithms will slowly consume the energy of the system, which will make the fluid motion of simulation be slower than the actual flow and gradually lose a lot of motion details.From the perspective of the development of fluid animation based on the physical model, early research has focused on obtaining fast and stable simulation method, but with the increasing demand for realism in the animation and the increase in computing resources, the algorithm of high accuracy and low numerical viscosity becomes more and more important.

Summary

Fluid animation based on physical model study has been a hot topic in computer graphics, which has broad prospect of application in the field of video effect, electronic games and other fields, and also have a good future in real-time interaction and virtual reality. However, it is still a challenging problem which is that designing a fluid system which is more convenient in interaction and can achieve real-time and can be almost completely reflect the real phenomena of nature.

References

[1] Fedkiw Ronald,Stam Jos,Jensen Henrik Wann.Visual simulation of smoke[A].In:Computer Graphics Proceedings,Annual Conference Series,ACM SIGGRAPH,Los Angeles,California,2001.15-22

[2] Hinsinger Damlen,Ncyret Fabfiee,Cani Marie Paule Interactive animation of ocean wave$[A]In: Proceedings of the 2002 ACM SIGGRAPH / Eurograph[es Symposium On ComputerAnimation,SanAntonio,Texas,2002.161-166

[3] Nguyen Due Quang,Fedkiw Ronald,Jensen Henrik Wann Physically based modeling and animation of fire[J].ACM Transactions on Graphics,2002,721-728

[4] Yngve Gary D,O'Brien James F,Hodgins Jessiea K.Animating explosions [A].In:Computer Graphics Proceedings,Annual Conference Series,ACM SIGGRAPll, New Orleans Louisiana.2000.29-36

[5] Hong Jeong Mo,Kim Chang Hun. Animation of bubbles inliquid,Computer Graphics Forum,2003,22(3):253-262

[6] Liu Y Q,Liu X H,Zhu H B,et al.Physically based fluid simulation in computer animation.J Comput Aid Des Comput Graph,2005,17:12

[7] Lamorlette Amauld.Foster Nick Structural modeling of flames for a production environment [J] .ACM Transactions on Graphics,2002.21(3):729-735

[8] Stare Jos.Stable fluids[A].In:Computer Graphics Proceedings,Annual Conference Series,ACM SIGGRAPH,Los Angeles,California,1999 12l-128

[9] Liu Youquan, Liu Xuehui, Wu Enhua. Real-time 3D fluid simulation on GPU with complex obstacles[A]. In:Proceedings of Pacific Graphics 2004,Seoul,2004.247-256

[10] Kass M,Miller G.Rapid,stable fluid dynamics for computer graphics.Comput Graph,1990,24 (4):49-57

[11] Bridson R.Shallow water discretization.Lect Note Animat Phys,2005

[12] Foster N,Metaxas D.Realistic animation of liquids.Graph Model Image Process,2007

[13] Stam J. Stable fluids.In:Proceedings of the 26th Anuual Conference on computer graphics and interactive techniques,SIGGRAPHP 99 .New York,1999

[14] Foster N,Metaxas D.Modeling the motion of a hot, turbulent gas.In:Proceeding of the 24th Anuual Conference on computer graphics and interactive techniques,SIGGRAPHP 97,1997

[15] Nguyen D Q,Fedkiw R,Jensen H W,Physically based modeling and animation of fire.2002

[16] Carlson M,Mucha P J,van Horn R,et al.Mellting and flowing.In:Proceeding of the 2002 ACM SIGGRAPH / Eurographics Symposium on Computer Animation,SCA 02.New York:ACM,2002.167-174

[17] Goktkin T G,Bargteil A W,Obrien J F.A method for animating viscoelastic fluids.In:ACM SIGGRAPH 2004.New York:ACM,2004,463-468

[18] Hong J M,Kim C H.Discontinuous fluids.ACM Trans Graph,2005

[19] Wang H,Mucha P J,Turk G.Water drops on surfaces.In:ACM SIGGRAPH 2005

[20] Kim B,Liu Y,Llamas I,et al.Simulation of bubbles in foam with the volume control method.ACM Trans Graph,2007

[21] Reeves W T.Particles systems-a technique for modeling a class of fuzzy objects.Comput Graph,1983

[22] Bridson R,Muller-Fischer M.Fluid simulation:SIGGRAPH 2007 Course Notes.In:ACM SIGGRAPH 2007.New York:ACM,2007.1-81

[23] Monaghan J J.Smoothed particle hydrodynamics.Annual Rev Astron Astr,1992,30:543-574

[24] Li Wei,Wei Xiaoming,Kaufman Arie.Implementing lattice Boltzmann computation on graphics hardware[J].The Visual Computer,2003,19(718):444-456

[25] Hong J M,Lee H Y,Yoon J C,et al.Bubble alive.In:Turk G,ed.ACM SIGGRAPH Conference Proceedings.New York:ACM,2008,1-4

[26] Takahashi T,Fujii H,Kunimatsu A,et al.Realistic animation of fluid with splash and foam.Comput Graph Forum,2003,22(3):391-400

Computational Materials Science
10.4028/www.scientific.net/AMR.268-270

Summary of Fluid Simulation Based on Physical Model
10.4028/www.scientific.net/AMR.268-270.1326

DOI References

[10] Kass M, Miller G. Rapid, stable fluid dynamics for computer graphics. Comput Graph, 1990, 24 (4): 49-57.
http://dx.doi.org/10.1145/97880.97884
[26] Takahashi T, Fujii H, Kunimatsu A, et al. Realistic animation of fluid with splash and foam. Comput Graph Forum, 2003, 22(3): 391-400.
http://dx.doi.org/10.1111/1467-8659.00686