International Journal of Multimedia and Ubiquitous Engineering
Vol. 10, No. 3 (2015), pp. 407-414
http://dx.doi.org/10.14257/ijmue.2015.10.3.37

Research on Vortex-Based Fluid Animation

Qiaoming Pan[1], Mingjun Wang[2*] and Weihua Shen[3]

Engineering College, Lishui University
[1]lsxypqm@163.com,[2] mingjun_w@163.com, [3]weihua_s@163.com

Abstract

Fluid animation has always been one of the most interesting and challenging projects. Fluid simulation based on physical methods can get vivid result. With its unparallel advantage of simulating incompressible fluid, vortex method is one of the most efficient methods. This paper mainly introduces the classification of vortex methods, draws a conclusion of previous study of vortex methods and makes a comparison between them.

Keywords: *vortex, fluid, physically-based, vortex particle, vortex filament*

1. Introduction

In recent years, physically-based fluid animation has been an interesting project. Bridson [1] made a comprehensive introduction about fluid animation simulation methods in his book. Grid-based and particle-based methods can make vivid fluid effect, but tremendous computational complexity makes fluid real-time simulation extremely difficult. In order to obtain real-time display, grid resolution is reduced or computational complexity is diminished, which gives rise to substantial loss of fluid detail. Thus the adding noise method is put forward to make up for the missing fluid detail, which can improve the visual effect and simulate rich whirlpool and turbulent flow to some extent. However, the random quality of noise method does not correspond with fluid mechanics. Therefore, it is not easy to simulate vivid whirlpool and turbulent flow effect.

Vortex movement is a common but the most important fluid way of movement. It has always been one of the most challenging projects in the field of fluid mechanics, engineering application and fluid animation. Vortex N-S equation can control fluid movement, but due to its non-lineal property, data solution is used to gain approximate solution under most circumstances. Vortex method has its appealing advantage in computational efficiency and visual effect. Since Angelidis [2] used vortex method to simulate smoke for the first time, vortex-based fluid simulation has gradually become popular in animation filed. Even though vortex method is widely-used in fluid mechanics, the complexity of its theory and the limit of its application hinder the development of vortex-based fluid simulation. At present, vortex method is commonly-used to simulate incompressible fluid phenomenon. For a long time, researchers have tried to apply vortex method to compressible fluid, but few made convincing result. This paper mainly introduces certain vortex methods that are used to simulate fluid, makes comparison between them and draws a conclusion about previous vortex method study.

2. Mathematical Expression of Vortex Method

Given a velocity $u(x,t)=(u,v,w)$ in three-dimensional space, angular velocity Ω in this velocity can be expressed in the following form

[*] Corresponding Author

ISSN: 1975-0080 IJMUE

$$\Omega(x,t) = \frac{1}{2}\nabla \times u(x,t) = \frac{1}{2}\begin{vmatrix} i & j & k \\ \frac{\partial}{\partial x} & \frac{\partial}{\partial y} & \frac{\partial}{\partial z} \\ u & v & w \end{vmatrix} \tag{1}$$

Vorticity ω is twice the amount of angular velocity, that is

$$\omega(x,t) = 2\Omega(x,t) = \nabla \times u(x,t) \tag{2}$$

At present there is only one equation, the famous Navier-Stokes, which can express all kinds of fluid. The Euler equation of Navier-Stokes is the following:

$$\frac{\partial u}{\partial t} + (u \cdot \nabla)u + \frac{\nabla p}{\rho} = \mu \nabla^2 u + f \tag{3}$$

$$\nabla \cdot u = 0 \tag{4}$$

Among which, $u = (u, v, w)$ represents velocity, p stands for pressure, ρ for velocity, f for exterior force (such as buoyancy, gravity, vortex restraining force *etc.*). The method of calculating vortex restraining force is the solution of rotation in velocity field: $\omega = \nabla \times u$, the method of calculating vortex normal is $N = \nabla|\omega| / |\nabla|\omega||$.

Use the two sides of equation to get rotation then reach the vortex method controlling equation:

$$\frac{\partial \omega}{\partial t} + (u \cdot \nabla)\omega - (\omega \cdot \nabla)u = \mu \nabla^2 \omega + \nabla \times f \tag{5}$$

among which, $(u \cdot \nabla)\omega$ is vortex advection term, $(\omega \cdot \nabla)u$ is the stretching and transforming of vortex. Using Lagrangian way of expression is as follows:

$$\frac{D\omega}{Dt} - (\omega \cdot \nabla)u = \mu \nabla^2 \omega + \nabla \times f \tag{6}$$

2.1. Biot-Savart's Law

In the vortex-based fluid simulation, the updating of velocity field is realized through vortex function, that is, the same point might be influenced by one or more vortex, the closer vortex is, the greater the influence. Biot-Savart’s Law is used to calculate the impact that vortex has on the velocity of each point. Biot-Savart’s Law is originally used to get the solution of magnetic field strength, here it is used to approximate fluid velocity, according to (2), velocity is solved from vorticity ω as follows:

$$u(r) = \frac{1}{4\pi}\int \frac{\omega \times r}{r^3} d^3 r' \tag{7}$$

among which, r represents the distance that the present particle is to vortex center. Equation (7) can get the velocity of each point in vorticity field approximately.

2.2. Helmholtz Velocity Decomposing Theorem

Helmholtz velocity decomposing theorem implies that a vector field can be resolved into the sum of a gradient vector field and a solenoidal vector field. That is

$$u = \nabla \times A - \nabla \varphi \tag{8}$$

among which, $\nabla \varphi$ is irrotational gradient field, $\nabla \times A$ is divergence-free solenoidal vector field, A is as follows:

$$A(r) = \frac{1}{4\pi}\int \frac{\nabla' \times u(r')}{|r - r'|} d^3 r' \tag{9}$$

If u represents velocity, then $\nabla' \times u(r')$ indicates Vorticity ω. For any φ, there is $\nabla \times (\nabla \varphi) \equiv 0$, therefore, from the two sides of equation (8) can get:

$$\omega = \nabla \times (\nabla \times A) = \nabla(\nabla \cdot A) - \nabla^2 A \tag{10}$$

Vortex-based fluid simulation has the following advantages: For one, the description of vorticity field is more vivid than velocity field, especially in complex or fast-changing field. For another, vortex is mainly concentrated on the object surface and bubble wake, so vorticity field is more efficient than velocity field. Finally, since using vortex method to calculate translational item does not need to use the grid to discrete, there is no data dissipation in calculation.

3. Classification of Vortex Method

3.1. Vortex Restriction Method

For incompressible fluid, traditional method often has data dissipation disadvantage. If grid resolution is not high enough, details of simulated fluid will be lost, such as swirl and tumbling effects. In order to solve this problem, Fedkiw [3] put forward vortex restriction method [3], which can retrieve lost details of simulated fluid. The fundamental idea of vortex restriction method is adding a diffusion term in momentum equation, as illustrated in: (11)

$$\frac{\partial u}{\partial t} + (u \cdot \nabla) u + \frac{\nabla p}{\rho} = \mu \nabla^2 u + f + \varepsilon s \tag{11}$$

Among which, s indicates restriction item, ε is control parameter. s consists of two parts, one is the vector pointing to vortex centre and the other is a scalar.

Vortex restriction method can amplify the effect of blurred fluid and create vivid simulation. However, this kind of method can only amplify existing vortex without adding any new vortex, in other words, if detailed effect is obscure, there is nothing vortex restriction can do about restoration and amplification. Meanwhile, vortex restriction method also brings a new issue--energy dissipation, which makes the whole simulation process dissatisfy the law of conservation of energy. The intensity of vortex restriction ε needs to be set, if value of ε is too small, fluid details cannot be on full display. If value of ε is too large, simulation result won't be real, as illustrated in reference [4]. So He [5] improved traditional vortex restriction method and put forward self-adaptive vortex restriction method, which is about the changing of vortex restriction intensity according to spiral degree instead of being set by a human being.

Vortex restriction method can make up the detail loss caused by data dissipation, however, this method can only amplify the existing vortex without adding new vortex, and therefore it is at wit's end to simulate such fluid phenomenon with swirl and tumbling effects as explosion and torrent. Because the simulation process will lose tremendous fluid details, using vortex restriction method cannot retrieve all the lost details, it can only amplify part of them. Another way to add new detail is to add noise in simulated fluid, though it can create rich swirling effect, the visual effect is not real due to its violation of fluid dynamics theory.

3.2. Vortex Particle Method

In order to offset the shortcomings of vortex restriction method and noise method, Sell [4] introduced vortex particle method. Vortex particle method is a comprehensive approach, which can be applied to Lagrangian particle and Euler grid, so it compensates the imperfection of pure Euler and pure Lagrangian method. Vortex particle method suits best to the simulation of incompressible fluid with rich swirl and tumbling effect. However, if the Lagrangian particle method is used to calculate velocity field, then particle velocity field needs to be recreated, huge amount of calculation is needed as well. Therefore, Sell used grid-based method to calculate particle velocity field to avoid the large amount of calculation, and used vortex restriction to realize trivial fluid detail. Vortex restriction method that Sell used requires the setting of vortex confinement factor, a factor that is too small or too big will influence the vividness of simulation. At the same time, if the diameter of vortex particle is too big, the vortex restriction factor choosing might be wrong which will result in infidelity. In order to solve this problem, He [6] put forward self-adaptive vortex particle method, which is about the automatic match in the influenced area with the particle velocity. If the velocity is zero, then delete the particle. This way vividness of simulation is enhanced, amount of calculation is reduced tremendously as well. Mullen [7] introduced a whole Euler method that is completely independent of time and space resolution, which solved grid-based data dissipation and could calculate particle velocity on a rough grid without undermining visual quality of fluid simulation.

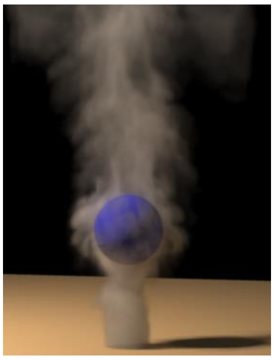

Figure 1. Fedkiw Simulated Smoke

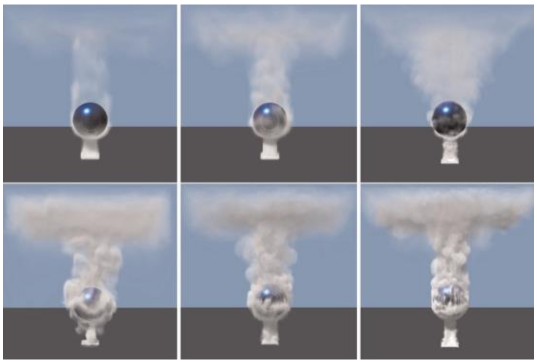

Figure 2. Sheng Feng He Self-adaptive Simulated Smoke

Particle-based methods have one common characteristic, that is, the more the particle, the less the simulation speed. So Park [8] used part of the vortex as fundamental element, used this element to describe the whole fluid movement. Though concentrating on calculating part of the vortex, calculation is simpler. The complexity of calculation is only 0 (N log N). Kim [9] introduced path control method, used set path to lead particle movement. However, these two methods cannot calculate the exact interaction between vortex and movement pattern of the particle. Thus the visual quality is affected. Another way to speed up simulation velocity is the hardware acceleration, such as GPU [10].

Vortex particle method is often based on Kolmogorov theory, so it suits the simulation of gaseous fluid like smoke instead of liquid fluid. Jang [11] introduced display separation method to replace Kolmogorov theory, which simulates water with turbulence effect. Coquerelle [12] combined vortex-based method with planar set method and the immersed boundary method to simulate the interaction between two liquid and rigid body. Using Vortex method to simulate two-way flow has incomparable advantage, especially the interaction between liquid and rigid body. Vortex method can easily simulate surface on rigid body and swirl in wake flow. Lee [13] used vortex particle method to simulate bubble in water from bottom to surface.

Vortex particle method can simulate vivid fluid phenomenon, such as smoke, explosion, torrent *etc*. However, there is no fluid simulation that suits everything; for certain fluid phenomenon such as small smoke, vortex particle is relatively lame.

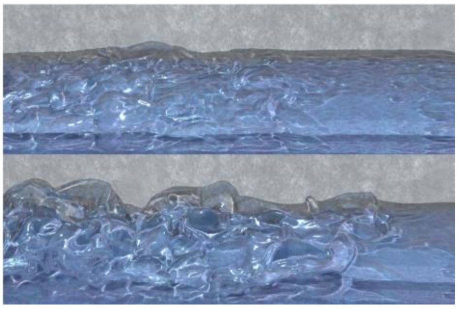

Figure 3. Selle Simulated Water

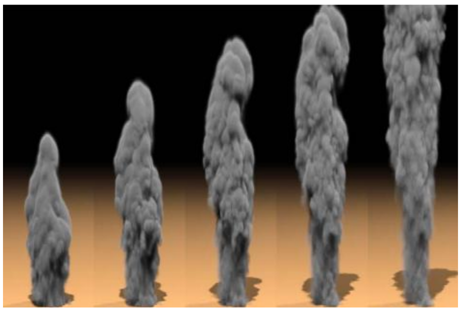

Figure 4. Selle Simulated Smog

3.3. Vortex Filament Method

From naked eyes, some fluid phenomena in nature are complicated without any rules, others can be observed with regularity beneath their complex appearance, such as cigarette smoke and tobacco. For the latter, vortex particle based method might find it hard to simulate smooth tobacco and cigarette smoke. So Angelidis [2] introduced vortex filament method to simulate the above-mentioned smog. Vortex filament method can be treated as the improvement of vortex particle method. It is different from vortex particle method in that the particle of vortex filament is interrelated to one another, which reduces large amount of repeated calculation for regridation or mesh refinement. Since the position of particle at any time must be marked, adding a new particle to two adjacent particles is very convenient. Each vortex line is a vorticity field concentrated on a curve. This vorticity field is either closed or extends to infinity along tangent of the curve. The size of it is vortex throughput of the intersecting surface of the ends of closed curve. We can treat vortex particle collection as a tubular structure whole. Vortex particle can ensure the non divergence quality of velocity field, but not the non divergence quality of vorticity field. Vortex filament can do the both. What's more, vortex filament is a very convenient structure, which flows with fluid without division, fusion or disappearance. Since circulation amount won't be changed, we only need to follow the shape of vortex filament, which reduces the amount of calculation as well.

The main parts of vortex are vortex filament and vortex circulation, which Angelidis mainly simulated of smog. Velocity is the calculation of vortex filament, which is approximated by Biot-Savart law. However, this can only simulate some characteristics of smog, such as vortex filament and vortex circulation, instead of a holistic view of smog particle movement. Therefore, Angelidis [14] added harmonic analysis and main component analysis to define fluid flow, which achieved good visual effect and made calculation amount increase with the number of particle.

In order to get more vivid fluid effect, fluid mechanics equation is required to describe fluid particle movement, which undoubtedly boosts calculation amount. So Weißmann [15-16] and Pinkall [17] used polygon to approximate vortex circulation. In vortex filament based fluid simulation, fluid movement is driven by vortex filament, which is similar to lagrangian method, the difference of which only lies in the object of calculation-vortex filament instead of a single particle. Any single vortex filament can influence velocity field in space, the solution of velocity needs huge amount of calculation. So Barnat [18] introduced a structure made of many vortex filaments and defined a new connection standard, which joins the adjacent vortex filament rings and made them a whole. Thus the amount of calculation brought by the influence that vortex filament has on other position has been reduced.

In order to speed up calculation, rendering method needs to be refined. One way is the use of GPU, which requires specific hardware condition. The other way is to minimize the number of vortex filament [18-19] without impairing visual effect or realize LOD animation [2] through one level vortex filament structure.

Even though vortex filament method can simulate vivid fluid phenomenon with good expansion quality, computational efficiency of it still needs to be improved. Particle movement in smog is driven by vortex filament, that is, it is affected by many vortex filaments at the same point. The more the vortex filament is, the larger the amount of calculation is.

4. Comparison between Different Vortex Methods

The biggest advantage that vortex-based simulation has is the simulation of incompressible fluid with vivid turbulence, swirl and tumbling effect. Table 1 illustrates some differences between the three methods. From Table 1 we can see, vortex restriction method can simulate rich details, but it can only amplify existing vortex instead of adding new ones, thus it can only get obscure result when fluid details lose too many. Vortex particle method and vortex filament method can both get more vivid fluid effect. Vortex particle can ensure the non-divergence of velocity field but not the vortex field. However, vortex filament method can do both. Vortex filament is a convenient structure, filament can move with fluid without division, fusion or disappearance. The circulation of vortex won't be changed, so we only need to follow the shape of vortex filament.

Table 1. Comparison between Different Vortex Methods

	Fundamental Principle	whether can add new detail	power source that drives fluid unit movement	the part that requires the largest amount of calculation
Vortex Restriction Method	retrieve detail loss caused by data dissipation and amplify these details	No, can only amplify existing detail	energy of particle itself	uncertain
Vortex Particle Method	$\omega_t + (u \cdot \nabla)\omega - (\omega \cdot \nabla)u = \mu\nabla^2 + \nabla \times f$	Yes	Vortex	calculate the impact that vortex has on velocity in space area
Vortex Filament Method	$\omega_t + (u \cdot \nabla)\omega - (\omega \cdot \nabla)u = \mu\nabla^2 + \nabla \times f$	Yes	Vortex Filament	calculate the impact that vortex filament has on velocity in space area

5. Research Tendency and Future Direction

5.1. Efficient Space Division Method

In vortex-based fluid simulation, the power that drives fluid unit movement is vortex or vortex filament, *etc*. So the updating of velocity field in space is through calculating the interaction of vortex or vortex filament between velocity fields. That is to say, any position in velocity field might be affected by vortices or vortex filament. For any position, if we calculate the interaction of all the vortices to it, the complexity of calculation is 0 (N2). When the number of vortex or vortex filament is too big, the amount of calculation is enormous. Treating all the vortices as a whole might be one solution, however, it will reduce the precision of calculation, and thus the visual effect is undermined. In order to reduce the amount of calculation, we can use certain grid, such as octree grid [20] *etc*.

5.2. Using Hardware to Accelerate

Physics-based fluid simulation requires large amount of calculation. Luckily we can use certain excellent graphics hardware to realize parallel computing, such as GPU. With the development of computer hardware, a majority of or even all the calculation can be handled by GPU. Vortex-based fluid simulation can also use GPU to accelerate. Rossinelli [10] used GPU to deal with the part of regridation and got precise real-time simulation of vortex effect. Vortex-based fluid simulation has its incomparable advantage of simulating the combination of fluid and rigid body with rich swirling effect. Since its calculation space is open, the management of boundary condition is more complicated. So we can consider using GPU to deal with the boundary condition, as in reference [20].

5.3. Mixed Methods

Traditional physics-based fluid simulation finds it hard to simulate rich turbulence and swirl effect. To combine vortex method with other ways, as an auxiliary method to add fluid details, is a solution, such as the combination of SPH method with vortex method [21], which can make up for the weaknesses of both sides and add rich fluid details as well.

5.4. Model Degradation

Theoretically, physics-based fluid simulation can simulate the exact fluid phenomenon in nature. However, due to the complexity of fluid movement and restriction of computer hardware, we can only get an approximate effect. The degradation of fluid model can obviously reduce the amount of calculation. Treuille [22] first used a precise non real-time solution to generate a series of fluid model with high precision, used simulation result as basic function of low dimensional subspace, then solved fluid control equation in low dimensional subspace. That is, to project a high dimensional question to low dimensional subspace and solve it in low dimensional subspace.

6. Conclusions

This paper introduces the basic concept and classification of vortex method, draws a conclusion about previous studies and makes comparison between them. This paper aims to give a reference about vortex method, usher beginners into the latest development without wasting time on huge amounts of literature.

When it comes to the simulation of incompressible fluid phenomenon, Vortex-based method has its incomparable advantages such as computational accuracy, complicated boundary condition, and direct simulation of Navier-Stokes method, which can simulate rich fluid detail with high computational efficiency. However, present vortex method mainly works for incompressible fluid phenomenon, gaseous fluid like smog, which cannot be applied widely. This is because the arithmetic is mainly based on Kolmogorov theory, which does not suit the simulation of liquid fluid vortices. In the field of fluid mechanics, vortex method for compressible fluid is still at research stage. At present, the only vortex method that can be used for incompressible fluid is expansion vortex method, but this method has not been applied in the field of computer graphics. In future we hope to make full use of vortex method, using it to the simulation of various fluid phenomena.

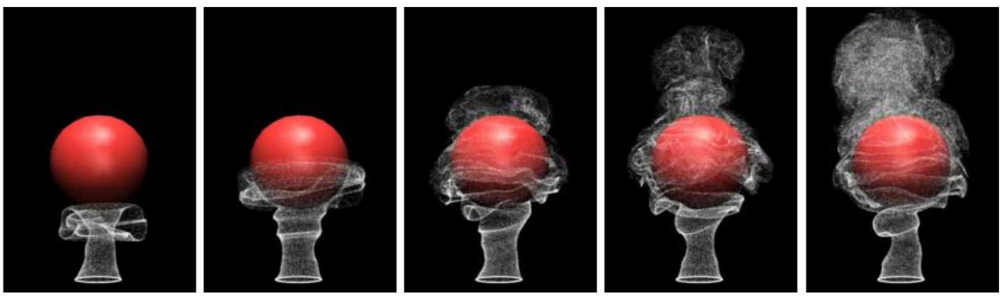

Figure 5. Weißmann Vortex Filament based Smog

Acknowledgement

The authors would like to thank both anonymous referees for their careful reading and helpful suggestions.

References

[1] R. Bridson, "Fluid Simulation for Computer Graphics", A K Peters, **(2008)**.
[2] Neyretf Angelidisa, "Simulation of Smoke Based on Vortex Filament Primitives", InSCA'05: Proc. of the Symposium on Computer Animation, **(2005)**, pp. 87-96.
[3] R. Fedkiw, J. Stam and H. Andjensen, "Visual simulation of smoke", Proc. of ACM SIGGRAPH **(2001)**, pp. 15-22.
[4] R. Jsellea, F. Iwr, "A Vor-tex Particle Method for Smoke, Water and Explosions", ACM Trans. Graph, vol. 24, vol. 3, **(2005)**, pp. 910-914.
[5] S. He, H. C. Wong, W. M. Pang and U. H. Wong, "Real-time smoke simulation with improved turbulence by spatial adaptive vorticity confinement", Computer Animation and Virtual Worlds (CASA2011 Special Issue), vol. 22, **(2011)**, pp. 107-114.
[6] S. He, H. C. Wong, W. M. Pang and U. H. Wong, "An Efficient Adaptive Vortex Particle Method for Real-Time Smoke Simulation", 12th International Conference on Computer-Aided Design and Computer Graphics, vol. 12, **(2011)**, pp. 317-324.
[7] P. Mullen, K. Crane, D. Pavlov, Y. Tong and M. Anddesbrun, "Energy-preserving Integrators for Fluid Animation", Proc. ACM/SIGGRAPH Conf., vol. 28, no. 3, **(2009)**.
[8] S. I. Park and M. J. Kim, "Vortex fluid for gaseous phenomena", SCA'05: Proceedings of the 2005 ACM SIG-GRAPH/Eurographics symposium on Computer animation, **(2005)**; New York.
[9] Z. Yuan, F. Chen and Y. Zhao, "Path-based Control of Smoke Simulations", ACM Transactions on Graphics (TOG) - Proceedings of ACM SIGGRAPH, **(2011)**.
[10] D. Rossinelli and P. Koumoutsakos, "Vortex methods for incompressible flow simulations on the GPU", The Visual Computer, vol. 24, **(2008)**, pp. 699-708.
[11] T. Jang, H. Kim, J. Bae, J. Seo and J. Noh, "Multilevel vorticity confinement for water turbulence simulation", Vis. Comput., vol. 26, **(2010)**, pp. 873-881.
[12] M. Coquerelle, J. Allard, G.-H. Cottet and M.-P. Cani, "A Vortex Method for Bi-phasic Fluids Interacting with Rigid Bodies", **(2006)**
[13] H.-Y. Lee, J.-M. Hong and C.-H. Kim, "Simulation of Swirling Bubbly Water using Bubble Particles", The Visual Computer, vol. 25, no. 5-7, **(2009)**, pp. 707-712.
[14] A. Angelidis, F. Neyret, K. Singh, Nowrouzezahraid, "A controllable, fast and stable basis for vortex based smoke simulation", Eurographics/SIGGRAPH Symposium on Computer Animation **(2006)**, pp. 25-32.
[15] S. Weißmann and U. Pinkall, "Real-time interactive simulation of smoke using discrete integrable vortex laments", VRIPHYS, **(2009)**, pp: 1-10.
[16] S. Weißmann and U. Pinkall, "Filament-based smoke with vortex shedding and vari-ational reconnection", ACM Trans. Graph, vol. 29, **(2010)**.
[17] U. Pinkall, B. Springborn and S. Wei Mann, "A new doubly-discrete analog of smoke ring flow and the real time simulation of fluid flow", J. Phys. A: Math. Theor, vol. 40, no. 42, **(2007)**.
[18] A. Barnat, "Smoke Sheets and Vortex Filaments with Flexible Reconnection", **(2011)**
[19] L. A. Ek, R. Vistnes and O. E. Gundersen, "Animating Physically Based Explosions in Real-Time", AFRIGRAPH' 07: Proceedings of the 5th international conference on Computer graphics, virtual reality, visualization and interaction in Africa, **(2007)**, pp. 61-69.
[20] L. Youquan, L. Xuehui and W. Enhua, "Real time 3D fluid simulation based on GPU with complex boundary", Journal of Software, vol. 17, no. 3, **(2006)**, pp. 568-576.
[21] B. Zhu, X. Yang and Y. Fan, "Creating and Preserving Vortical Details in SPH Fluid", Pacific Graphics 2010, vol. 29, no. 7, **(2010)**.
[22] A. Treuille, A. Lewis and Z. Popovic, "Model reduction for real-time fluids", ACM Trans Graph, vol. 25, no. 3, **(2006)**, pp. 826–834.