社団法人 情報処理学会 研究報告
IPSJ SIG Technical Report

2007－CG－128 (11)
2007／8／17

重なり格子法を用いた水と空気との動力学的相互作用を考慮した効率的なシミュレーション

ファン チュイティチュ† 楽 詠瀬†
土橋宜典†† 西田友是†

流体シミュレーションは、コンピュータグラフィックスにおいて重要な研究分野になってきた。本研究の目的は、空気の影響により変化する水表面を高精細にシミュレーションすることである。水表面を正確に計算するために、全計算領域において、高解像度の格子を使用すると、計算コストが膨大となる。そこで、我々は重なり格子法により、異なる解像度の格子を用いて水表面を計算する方法を提案する。提案法により全計算領域において高解像度の格子を用いた場合に比べて、低い計算コストで高精細な水の波のシミュレーションができる。

An Efficient Method for Simulating Dynamics of Interaction between Water and Air using Overlapping Grids

Pham Thuy Thi Thu,† YongHao Yue ,† Yoshinori Dobashi ††
and Tomoyuki Nishita†

Fluid simulation has become an important topic in the field of computer graphics. The aim of this paper is the realistic simulation of water waves caused by the interaction between water and air flows (i.e wind). In order to calculate water waves accurately, high resolution grids are required. However this leads to increasing computational cost. To overcome this problem, we present a new method for simulating water waves due to wind by using overlapping grids with different resolutions. The method makes it possible to generate accurate shapes of water waves without increasing computational cost.

1. Introduction

In recent years, fluid simulation has become an important subjects in the field of Computer Graphics (CG). Animating fluids like water, smoke, and fire by physics-based simulation is increasingly important in visual effects, especially simulations of water waves are widely used in movies and computer games recently. To simulate water waves, there are several approaches using waves equation. However, in this paper, we consider both water and air as two kinds of fluids and water waves are simulated as the interaction between them. Fluids behaviors are governed by well-known Navier-Stokes equations[10]. In 1999, Stam proposed a method to solve Navier-Stokes equations called the Stable fluids method[11] which has made significant progress in fluid simulation because this method allows the use of a significantly large time step without losing stability. To track the movement of fluids with free surfaces like water, Osher and Sethian developed a technique called Level Set Method[9] which has been used in almost all liquids simulation recently. One problem that CG researchers always have to deal with is high computational cost, especially in simulations where detail is required, because in such cases we have to use a very high resolution grid to get accurate and realistic results. Octree data structure[8], Hybrid mesh[5] were proposed to deal with the case when solid objects are inside the fluid. However, if objects move inside the fluid, using these methods, we have to rebuild our mesh repeatedly which costs a lot of time. Dobashi proposed a fluid simulation method using overlapping grids[4] and applied it to fluids without free surfaces

† 東京大学 (The University of Tokyo)
†† 北海道大学 (Hokkdaido University)

like smokes. In this method multiple grids are used, one called the global grid for the entire space, the others called local grids for regions where accurate results are needed such as around objects or near viewpoint. If an object moves, the grid assigned to it is also moved. By this way, the grids do not have to be rebuilt repeatedly like in Octree data structure or Hybrid mesh.

In this paper, we extend Stable Fluids Method and Overlapping Grids Method to deal with multiple fluids and fluids with free surfaces. As an application, we simulate water waves caused by the dynamics of interaction between water and air. We show that by using overlapping grids near the water surface, we can obtain detailed images of the water surface in dynamic regions with considerably reduce computational cost in comparison with that when using one grid which has the same resolution as local grids.

The rest of this paper is organized as follows. In Section 2, we briefly review several methods for accurate calculation in fluid simulation. In Section 3, we described in detail Overlapping Grids Method and how it is applied in water-air interaction. Results are shown in Section 4 and conclusion and future work are discussed in Section 5.

2. Water-Air Interaction Solver

Stable Fluids Method is the basic method in fluid simulation, used to simulate fluid without free surface like smokes. To deal with fluids with free surface like water, it becomes more complicated. We not only have to calculate fluid's velocity accurately but also need a method to trace the evolution of water surface. Moreover, in case of two fluids which have very different density like water and air (the density of water is around 1000 times that of air), some modifications is needed in the calculation of fluid's pressures. In the rest of this section, we review several methods to compute fluid's velocity and pressure accurately. Although each of them has already appeared in previous literature, we couldn't find a paper that contains all of them.

In the rest of this paper, for simplicity our explanation are done in 2D case although it is still applied in 3D case.

2.1 Governing Equations

We consider water and air as two kinds of fluid and their behaviors are governed by Navier-Stokes equations for incompressible viscous fluids,

$$\mathbf{u}_t = -(\mathbf{u}\cdot\nabla)\mathbf{u} + \nu\nabla^2\mathbf{u} - \frac{\nabla p}{\rho} + \mathbf{g}, \quad (1)$$

$$\nabla\cdot\mathbf{u} = 0, \quad (2)$$

where $\mathbf{g}$ is the gravity, $\mathbf{u} = (u, v, w)$ is the velocity, p is the pressure, ρ is the density and ν is the kinematic viscosity of the fluid.

Our solver is based on Stable Fluids Method, the numerical algorithm for Navier-Stokes equations, and Level Set Method which presents the evolution of the interface between water and air. We employ the level set function ϕ, a signed distance function defined to be negative for water, positive for air. Thus, the sign of ϕ distinguishes water and air regions, and also determines the density ρ and kinematic viscosity ν at any point in the simulation space. The fluids dynamically evolve in space and time due to the velocity field $\mathbf{u}$ of the fluids. This is described by the level set equation:

$$\phi_t + \mathbf{u}\cdot\nabla\phi = 0. \quad (3)$$

We solve Equation (3) using semi-Lagrangian method[11]. Every time step, the advection of level set ϕ causes the density and viscosity fields appearing in Equation (1) to be updated. One drawback of Level Set Method is severe volume loss especially on relatively coarse grids. A method to preserve water volume is described in the Appendix.

2.2 MAC Grid

To solve Equations (1) and (2), we divide the simulation space into a finite number of cells. We use staggered MAC grid model[6] in which pressure, level set value, density of a cell are stored at the cell's center, while velocities are stored at the cell's faces. This model is in contrast to the the standard collocated grid used in Stable Fluids Method in which pressure and velocities are all stored at cells' centers. The model of MAC is illustrated in Figure 1. For cell (i, j) the velocities at the left and right cell faces are denoted as $u_{i-\frac{1}{2},j}$ and $u_{i+\frac{1}{2},j}$, while the velocity at the bottom and top face are denoted as $u_{i,j-\frac{1}{2}}$ and $u_{i,j+\frac{1}{2}}$. When advecting velocity field as well as level set function using semi-Lagrangian method, we need the velocities at cells'

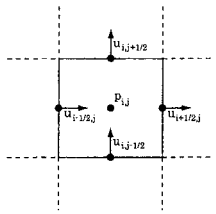

Fig.1 MAC grid

centers and they are determined as follows,

$$\mathbf{u}_{i,j} = \left(\frac{u_{i-\frac{1}{2},j} + u_{i+\frac{1}{2},j}}{2}, \frac{v_{i,j-\frac{1}{2}} + v_{i,j+\frac{1}{2}}}{2} \right). \quad (4)$$

The main advantage of a MAC grid over a collocated grid is that it allows first order central differences to be performed with respect to velocity and pressure. For example, to compute $\frac{\partial \mathbf{u}}{\partial t}$ at the center of cell (i,j), in case of collocated grid, we have to use the second order central differences,

$$\left.\frac{\partial \mathbf{u}}{\partial t}\right|_{i,j} = \left(\frac{u_{i+1,j} - u_{i-1,j}}{2dx}, \frac{v_{i,j+1} - v_{i,j-1}}{2dy} \right), \quad (5)$$

which completely ignores the velocity of the current cell (i,j), and in general makes inefficient use of given data.

In case of MAC grid, the derivative can be computed as

$$\left.\frac{\partial \mathbf{u}}{\partial t}\right|_{i,j} = \left(\frac{u_{i+\frac{1}{2},j} - u_{i-\frac{1}{2},j}}{dx}, \frac{v_{i,j+\frac{1}{2}} - v_{i,j-\frac{1}{2}}}{dy} \right), \quad (6)$$

which is unbiased and accurate to $O(\Delta x^2)$, and does not skip any value. Therefore, using MAC-grid, the velocity field can be computed more accurately than when using collocated-grid.

2.3 Updating Velocity Field and Pressure Field

The velocity field is updated from nth time step $\mathbf{u}^n$ to $(n+1)$th time step $\mathbf{u}^{n+1}$ through the following steps.

(1) Apply the external force. We apply gravity $\mathbf{g}$ to both water and air regions.

(2) Calculate the advection term $\mathbf{u} \cdot \nabla \mathbf{u}$ using the semi-Lagrangian method.

(3) Compute the effect of the viscous term $\nu \nabla^2 \mathbf{u}$ by employing implicit central differencing. In our case we apply ν_{water} for water region ($\phi \leq 0$) and ν_{air} for air region ($\phi > 0$).

(4) Project the velocity field so that the condition $\nabla \cdot \mathbf{u} = 0$ is satisfied. Suppose that the $\mathbf{u}^*$ is the velocity obtained after previous three steps, we have to compute pressure p using the following equation:

$$\nabla \cdot \left(\frac{\nabla p}{\rho} \right) = \frac{\nabla \cdot \mathbf{u}^*}{\Delta t}, \quad (7)$$

and using p to compute the velocity in the next time step

$$\mathbf{u}^{n+1} = \mathbf{u}^* - \frac{\Delta t}{\rho} \nabla p. \quad (8)$$

In the case of water and air's interaction, density ρ is spacially varying, Equation (7) is discreted as

$$\begin{aligned} &\sum_{i,j} \{ \left(\rho^{-1}_{i+\frac{1}{2},j} + \rho^{-1}_{i-\frac{1}{2},j} + \rho^{-1}_{i,j+\frac{1}{2}} + \rho^{-1}_{i,j-1\frac{1}{2}} \right) p_{i,j} \\ &- \rho^{-1}_{i+\frac{1}{2},j} p_{i+1,j} - \rho^{-1}_{i-\frac{1}{2},j} p_{i-1,j} \\ &- \rho^{-1}_{i,j+\frac{1}{2}} p_{i,j+1} - \rho^{-1}_{i,j-\frac{1}{2}} p_{i,j-1} \} \\ &= -\frac{h}{\Delta t} \sum_{i,j} ({\mathbf{u}^*}_{i+\frac{1}{2},j} - {\mathbf{u}^*}_{i-\frac{1}{2},j} + {\mathbf{u}^*}_{i,j+\frac{1}{2}} - {\mathbf{u}^*}_{i,j-\frac{1}{2}}), \end{aligned} \quad (9)$$

where h is the interval between two grid points, $p_{i\pm1,j\pm1}$ are the pressure values taken from the centers of the neighboring cells, $u_{i\pm\frac{1}{2},j}$, $u_{i,j\pm\frac{1}{2}}$ and $\rho_{i\pm\frac{1}{2},j}$, $\rho_{i,j\pm\frac{1}{2}}$ are the velocities and densities taken form the cell faces.

2.4 Handling Multiple Fluids

In case of multiple fluids, Hong and Kim used ghost fluid method to deal with pressure jumps due to surface tension across the interface between two fluids[7]. In this paper we take into consideration the difference between the fluids' densities to simulate the interaction between them. Using Equation (9), we need fluids' densities at cell faces. Because fluid's density is only sampled at a cell center, the density at a cell face is computed using level set values. For example, the density at the face between cell (i,j) and cell $(i+1,j)$ is calculated as follows.

(i) If $\rho_{i,j} = \rho_{i+1,j}$, i.e. cell (i,j) and cell $(i+1,j)$ are both in water or air region, then $\rho_{i+\frac{1}{2},j} = \rho_{i,j} = \rho_{i+1,j}$.

(ii) If $\rho_{i,j} \neq \rho_{i+1,j}$, it means that one cell is in water and the other is in air region. Remember that level set value $lset_{i,j}$ indicates the distace from $cell(i,j)$ to water surface, therefore, by comparing the absolute value of $lset_{i,j}$ and $lset_{i+1,j}$, we can determine whether the face between cell (i,j) and cell $(i+1,j)$ is in water or air.

$$\rho_{i+\frac{1}{2},j} = \begin{cases} \rho_{i,j} & \text{if } |lset_{i,j}| > |lset_{i+1,j}|; \\ \rho_{i+1,j} & \text{if } |lset_{i,j}| < |lset_{i+1,j}|; \\ \frac{\rho_{i,j}+\rho_{i+1,j}}{2} & \text{if } |lset_{i,j}| = |lset_{i+1,j}|. \end{cases} \quad (10)$$

2.5 Linear System Solver

The collection of equations in the form of Equation (9) makes a linear system

$$\mathbf{Ap} = \mathbf{b} \quad (11)$$

where $\mathbf{p}$ is the vector of unknown pressures required to make the velocity field divergence free. Vector $\mathbf{b}$ is the divergence of the velocity field $\mathbf{u}^*$, matrix $\mathbf{A}$ presents the constraint between $\mathbf{p}$ and $\mathbf{b}$. To avoid making matrix $\mathbf{A}$ singular, we set boundary conditions for pressure as follows.

- Dirichlet boundary condition for top row, i.e. $p_{i,0} = constant$.
- Neumann boundary condition of 0 for the other boundaries, i.e.
$$\left.\frac{\partial p}{\partial x}\right|_{0,i} = \left.\frac{\partial p}{\partial x}\right|_{Nx+1,i} = \left.\frac{\partial p}{\partial y}\right|_{i,Ny+1} = 0, \quad (12)$$
where Nx, Ny are the horizontal and vertical resolutions of the grid. By these boundary conditions, $\mathbf{A}$ is a positive-definite, symmetric matrix, therefore Equation (11) can be efficiently solved using SOR (Successive Over Relaxation) method[1]. A further discussion on the boundary conditions can be found in the doctoral thesis of Mark Carlson[3].

3. Overlapping Grids Method

3.1 Overview of Overlapping Grids Method

The basic idea of Overlapping Grids Method is to reduce computational cost by using different resolution grids in different regions of fluid. There are a single global grid and multiple local grids.

- One coarse global grid is assigned to the entire simulation space.
- One local grid with high resolution is assigned to each region where fluid's behavior is complicated and needs to be displayed in details such as around water surface.

One advantage of Overlapping Grids Method is that local grids can be moved along with the movement of water surface without being rebuilt every time step like Octree Data Structure[8] or Hybrid Meshes[5]. We only have to add inertial force caused by the movement of a local grid to every grid point of that local grid as external force. Navier-Stokes equations are solved on global grid as well as on each local grid. There are two important points we want to mention when solving Navier-Stokes equations on overlapping grids.

- Inertial force is added as external force for moving local grids.
- Data such as velocity, pressure and level set value is exchanged between global grid and local grids. Before the calculation at each time step, data is copied from global grid to local grids, and after calculation, they are copied back from local grids to global grid.

The data exchange process is illustrated in Figure 2 and is described as follows.

- Copy from global grid to local grids: we only copy data from global grid to the boundaries of local grids.
- Copy from local grids to global grids: all data of local grids except that on the boundaries is copied to the corresponding positions in global grid.

In case multiple local grids are used and they are also overlapping each others, data is exchaged as follows.

Assume that we have n grids $g_1, g_2, \cdots, g_n$ in increasing resolution order, g_1 is the global grid.

- Copy from low resolution grids to higher resolution grids: for each grid $g_k (k = 1, \ldots, n)$ except the global grid, data from grid g_i $(i < k)$ corresponded to the boundary of grid g_k is copied to the boundary of grid g_k and this process is carried out from $i = 1$ to $i = k - 1$.
- Copy from high resolution grids to lower resolution grids: for each grid $g_k (k = 1, \ldots, n)$ except the highest resolution grid g_n, all data except data on the boundary of grid g_i $(i > k)$ is copied to the corresponding position in grid g_k. This process is carried out from $i = n$ down to $i = k + 1$.

3.2 Water-Air Interaction Simulation Using Overlapping Grids

To simulate the interaction between water and air, local grids are placed so that they cover the water surface like in Figure 3. We let local grids

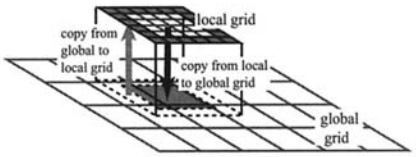

Fig.2 Data exchange between global grid and local grids

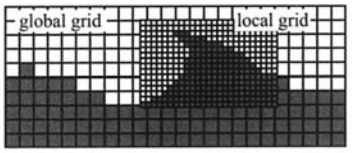

Fig.3 A local grid is placed to cover water surface.

overlap each other at the boundaries of local grids so that the computational accuracy does not degrade at the boundaries. In case dynamic waves occur, the local grids may no longer cover the entire water surface, therefore these local grids are moved in vertical direction along with the up and down movement of the surface so that they can always cover the surface.

To decide the velocity of a local grid, we take the average value of the velocity of water at grid points near water surface inside that local grid. The inertial force acting on each local grid is calculated as follows.

$$\mathbf{f} = -\frac{d\mathbf{u}}{dt}, \tag{13}$$

where $\mathbf{u}$ is the velocity of the local grid. One more point to be considered here is using Level Set Method in overlapping grids. After we obtain the velocity field at each time step in all grids, we use this velocity field to update the level set values in the global grid as well as the local grids using Equation (3). For a moving local grid, velocities at grid points in the local grid we used to advect level set function is the relative velocities to the local grids, not to the global grid. Therefore, before the advection process, the velocity of the local grid must be substracted from the velocity at each grid point. The subtracted velocity is added again after the advection. Level set values are also copied from global grid to local grids before advecting and after level set values are updated, they are copied back from local grids to global grid. Finally, level set values are reinitialized in all grids using Fast Marching

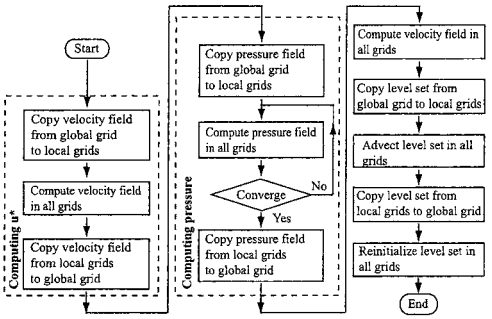

Fig.4 Computation flow in one time step when using overlapping grids.

Method[2].The computation flow in one time step is shown in Figure 4.

4. Results

We applied our method to simulate dynamics of water waves caused by local wind in 2D case. The grid points in the regions in which winds blow have the velocity of the winds. The places and directions of the winds are indicated by the arrows in the result images. Local grids are displayed as rectangles with black boundaries.

We compared the computational costs when using our method with those when using only one global grid. Images obtained when using one grid are shown in Figure 5. Figure 6 shows the results when using one grid on the left and the result when using overlapping grids on the right. Figure 7 shows how local grids are moved along with the movement of water surface. In this figure, the two local grids in the middle are moved up. In Figure 6 and 7, when using overlapping grids, the resolution of local grids is 4 times that of global grid. Comparision of the computation times is shown in Table 1. Our

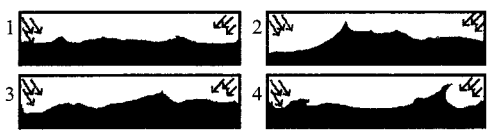

Fig.5 Results when using one grid with resolution 256×64.

Fig.6 Results when using one grid (left) and overlapping grids (right).

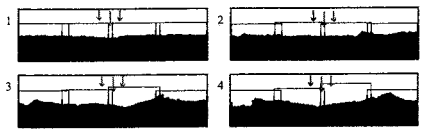

Fig.7 Local grids are moved along with the movement of water surface. The resolution of global grid is 128 × 32 and that of local grids is 256 × 64.

Table 1 Computational times

	Grid size	Comp.time(s/f)
Stable fluids and Level set method	256 × 64	(a) 0.239
	512 × 128	(b) 4.604
Present method	global: 128 × 32	(c) 0.092
	local: 256 × 64	
	global: 256 × 64	(d) 0.906
	local: 512 × 128	

experiments were conducted on machine with Intel(R) Core 2 Extreme 2.94GHz and 2.00GB RAM. From Table 1, it is indicated that using high resolution only around water surface, computational time is 2.5 to 5 times faster ((a) is compared with (c), (b) is compared with (d)). Note that when the grid's resolution increases, the iteration for computing pressures using SOR method also increases. Thus, the overall computation time increases more than in theory which is 4 times (about 20 times in our experiments) when we increase the grid's resolution 4 times. The resolutions of grids should be as low as possible in order to make the simulation be performed within practical time, on the other hand, the resolution should be high enough to obtain accurate results. By using overlapping grids, it is possible to obtain accurate results within practical time.

5. Conclusions and Future Work

We have presented an efficient method using overlapping grids for simulating detailed images of water motion. We have presented the water-air interaction solver, Overlapping Grids Method to deal with fluids with free surface. Using our method, it is possible to reduce computational cost but still keep accurate computation. Our experiments have shown that detailed images of dynamic water waves could be obtained with considerably low computational cost.

One important future work is to implement our method in 3D case. Moreover, there is still room for further research on the way to use overlapping grids to simulate water waves. One local grid can be divided into multiple grids when the difference between the heights of water surface at different locations become too large. These new grids are moved with different velocities so that they can still cover all water surface. By this way, overlapping grids can be used in a more efficient way.

References

1) R. Barrett, M. Berry, T. F. Chan, J. Demmel, J. Donato, J.Dongarra, V.Eijkhout, R.Pozo, C.Romine, and H. V. der Vorst. *Templates for the Solution of Linear Systems: Building Blocks for Iterative Methods, 2nd Edition.* SIAM, Philadelphia, PA, 1994.
2) R. Bridson, R. Fedkiw, and M. Muller-Fischer. Fluid simulation: Siggraph 2006 course notes. In *SIGGRAPH '06: ACM SIGGRAPH 2006 Courses*, pp. 1–87, New York, NY, USA, 2006. ACM Press.
3) M. T. Carlson. *Rigid, melting, and flowing fluid.* PhD thesis, Atlanta, GA, USA, 2004. Director-Greg Turk and Director-Peter J. Mucha.
4) Y. Dobashi, T. Yamamoto, and T. Nishita. Visual simulation of fluids using overlapping grids. In *Proceedings of VC Symposium*, pp. 59–64, 2006.
5) B.E. Feldman, J.F. O'Brien, and B.M. Klingner. Animating gases with hybrid meshes. In *SIGGRAPH '05: ACM SIGGRAPH 2005 Papers*, pp. 904–909, New York, NY, USA, 2005. ACM Press.
6) F. H. Harlow and J. E. Welch. Numerical calculation of time-dependent viscous incompressible flow of fluid with free surface. *The Physics of Fluids*, 8:2182–2189, 1965.
7) J.-M. Hong and C.-H. Kim. Discontinuous fluids. In *SIGGRAPH '05: ACM SIGGRAPH 2005 Papers*, pp. 915–920, New York, NY, USA, 2005. ACM Press.
8) F. Losasso, F. Gibou, and R. Fedkiw. Simulating water and smoke with an octree data structure. *ACM Trans. Graph.*, 23(3):457–462, 2004.
9) S. Osher and J. A. Sethian. Fronts propagating with curvature-dependent speed: Algorithms based on Hamilton-Jacobi formulations. *Journal of Computational Physics*, 79:12–49, 1988.
10) H. Sohr. *The Navier-Stokes Equations: An Elementary Functional Analytic Approach.* Bertelsmann Springer Publishing Group.
11) J. Stam. Stable fluids. In *SIGGRAPH '99: Proceedings of the 26th annual conference on Computer graphics and interactive techniques*, pp. 121–128, New York, NY, USA, 1999. ACM Press/Addison-Wesley Publishing Co.

Appendix

Suppose that the initial volume of water is V_0 and that at nth time step is V_n. These volume can be easily calculated by counting the number of grid cells which has negative or zero level set value. According to volume error $dV = V_0 - V_n$, we compute the number of grids that water surface needs to be shifted up as dV/S, where S is the area of water surface.