การปรับปรุงการควบคุมของไหลในสามมิติ โดยใช้การประมาณค่าเคอร์เนลแบบปรับตัวได้

นางสาว สายทิพย์ ลิ้มตระกูล

วิทยานิพนธ์นี้เป็นส่วนหนึ่งของการศึกษาตามหลักสูตรปริญญาวิทยาศาสตรมหาบัณฑิต
สาขาวิชาวิทยาศาสตร์คอมพิวเตอร์ ภาควิชาวิศวกรรมคอมพิวเตอร์
คณะวิศวกรรมศาสตร์ จุฬาลงกรณ์มหาวิทยาลัย
ปีการศึกษา 2552

AN ENHANCEMENT OF 3D FLUID CONTROL USING ADAPTIVE KERNEL ESTIMATION

Miss Saithip Limtrakul

A Thesis Submitted in Partial Fulfillment of the Requirements
for the Degree of Master of Science Program in Computer Science
Department of Computer Engineering
Faculty of Engineering
Chulalongkorn University
Academic Year 2009

Thesis Title	AN ENHANCEMENT OF 3D FLUID CONTROL USING ADAPTIVE KERNEL ESTIMATION
By	Miss Saithip Limtrakul
Field of Study	Computer Science
Thesis Advisor	Assistant Professor Pizzanu Kanongchaiyos, Ph.D.

Accepted by the Faculty of Engineering, Chulalongkorn University in Partial Fulfillment of the Requirements for the Master's Degree

.... Dean of the Faculty of Engineering
(Associate Professor Boonsom Lerdhirunwong, Dr.Ing.)

THESIS COMMITTEE

.... Chairman
(Professor Prabhas Chongstitvatana, Ph.D.)

.... Thesis Advisor
(Assistant Professor Pizzanu Kanongchaiyos, Ph.D.)

.... External Examiner
(Chakrit Watcharopas, Ph.D.)

สายทิพย์ ลิ้มตระกูล : การปรับปรุงการควบคุมของไหลในสามมิติ โดยใช้การประมาณค่าเคอร์เนลแบบปรับตัวได้. (AN ENHANCEMENT OF 3D FLUID CONTROL USING ADAPTIVE KERNEL ESTIMATION) อ. ที่ปรึกษาวิทยานิพนธ์หลัก : ผศ. ดร.พิษณุ คนองชัยยศ, 62 หน้า.

ในงานทางด้านคอมพิวเตอร์แอนิเมชัน ผู้ใช้หรือผู้ทำแอนิเมชันนั้นต้องการเครื่องมือสำหรับควบคุมการเคลื่อนไหวของของเหลว ซึ่งหนึ่งในวิธีการที่เป็นที่รู้จักและนิยมใช้กันอย่างแพร่หลายในการจำลองการเคลื่อนไหวของของเหลวในงานด้านกราฟิก คือ Smoothed Particle Hydrodynamics หรือ เอสพีเอช ถึงแม้ว่าเอสพีเอชจะเป็นวิธีแบบอนุภาคที่สามารถนำไปประยุกต์ใช้กับการควบคุมได้ง่ายและสามารถรักษาสมบัติการเคลื่อนที่ของของไหลได้ดี แต่อย่างไรก็ตาม ก็ยังมีความซับซ้อนเมื่อใช้ในการจำลองงานที่มีความละเอียดหลากหลาย ในงานวิจัยนี้ เราได้นำเสนอวิธีการจำลองของไหลซึ่งมีพื้นฐานมาจาก SPH โดยที่สามารถควบคุมได้และสามารถปรับขนาดของเคอร์เนลได้ เพื่อรักษารายละเอียดขนาดย่อยที่สูญหายไปหากใช้เคอร์เนลที่มีขนาดคงที่ ในการควบคุมการเคลื่อนไหวของของไหลเราได้ใช้วิธีการที่เรียกว่า Skeletal Particle โดยหาโครงสร้างหลักของวัตถุด้วย Reeb graph และวางอนุภาคที่ใช้เป็นตัวแทนของเพื่อนบ้านหรืออนุภาคใกล้เคียงเอาไว้ที่จุด critical point ของ reeb graph ที่ได้ และใช้อนุภาคตัวแทนนี้เป็นจุดศูนย์กลางในระหว่างการคำนวณเพื่อหาค่าเฉลี่ยให้กับเพื่อนบ้านตัวอื่นๆที่อยู่ในขอบเขตความยาวของเคอร์เนล ในการแก้ปัญหาของความยาวเคอร์เนลที่คงที่, ระหว่างการคำนวณ เราปรับค่าความยาวของเคอร์เนลโดยอาศัยเทคนิคที่ชื่อว่า Adaptive Kernel Density Estimation มาใช้เพิ่มประสิทธิภาพ โดยสัมประสิทธิ์ที่จะนำมาใช้ในการปรับค่าความยาวของเคอร์เนลนี้เราพิจารณาจากอัตราส่วนของความหนาแน่น และเงื่อนไขที่ได้จาก Reeb Graph มาช่วยในการพิจารณา ดังนั้นความยาวของเคอร์เนลจะถูกปรับให้เหมาะสมกับความหนาแน่นของอนุภาคที่มีอยู่ในบริเวณ ซึ่งส่งผลให้การหาค่าเฉลี่ยของสมบัติต่างๆนั้นผิดพลาดเนื่องจากอนุภาคไม่เพียงพอนั้นน้อยลง

ภาควิชา......วิศวกรรมคอมพิวเตอร์........ ลายมือชื่อนิสิต........สายทิพย์ ลิ้มตระกูล..........
สาขาวิชา.....วิทยาศาสตร์คอมพิวเตอร์.... ลายมือชื่อ อ.ที่ปรึกษาวิทยานิพนธ์หลัก..............
ปีการศึกษา..2552..............................

4970624021 : MAJOR COMPUTER SCIENCE
KEYWORDS : CONTROLLABLE FLUID/ FLUIDS SIMULATION/ SMOOTHED PARTICLE HYDRODYNAMICS/ ADAPTIVE KERNEL ESTIMATION

SAITHIP LIMTRAKUL : AN ENHANCEMENT OF 3D FLUID CONTROL USING ADAPTIVE KERNEL ESTIMATION. THESIS ADVISOR : ASSISTANT PROFESSOR PIZZANU KANONGCHAIYOS, Ph.D., 62 pp.

In computer animation, animation tools are required for fluid-like motions which are controllable by users or animators. One of the popular and widely used methods for simulating fluid flow in computer graphics is Smoothed Particle Hydrodynamics (SPH). Although SPH is practical for applying to fluid movement control and also preserves efficiently properties of fluid flow, it is complicated to simulate various details in same flow. This study proposes an enhanced method based on Smoothed Particle Hydrodynamics (SPH) which is controllable and automatically kernel length adjustable in order to preserve small details; lost by fixed kernel. Reeb graph is used to construct a structure of an input object. To control fluid flow, we use a kind of particle control methods called Skeletal Particles. Skeletal Particles or control particles are placed on the center of smoothing kernel in computation step. To solve the problem caused by fixed kernel length, we implement the Adaptive Kernel Density Estimation (AKDE) technique into the computation step. Therefore, the smoothing length is dynamically adapted to all given points to be appropriate with density of their regions. As a result, the errors of unstable mean values of particles in low density regions are decreased.

Department : Computer Engineering
Student's Signature SAITHIP LIMTRAKUL

Field of Study : Computer Science
Advisor's Signature

Academic Year : 2009

ACKNOWLEDGEMENTS

It is a great pleasure to acknowledge my thesis advisor, Assistant Professor Pizzanu Kanongchaiyos, Ph.D., for his intellectual advices and invariable assistances throughout this research. I would also like to express my grateful thanks to my thesis committee, Professor Prabhas Chongstitvatana, Ph.D, and Chakrit Watcharopas, Ph.D. for their beneficial guidance and suggestions.

I would like to thank the National Institute of Informatics, Japan for the opportunity to join the internship program. I am pleased to thank Professor Hayami Ken for his helpful suggestion during that time.

Also, this thesis was supported by the scholarship of the Development and Promotion of Science and Technologies Talents.

I want to extend my thanks to all research members especially my associates in computer graphic lab (CG Lab) for their assistance

Finally, I deeply wish to thank my parents for their love, understanding and invaluable supports throughout my graduate study.

CONTENTS

Page

LIST OF TABLES

LIST OF FIGURES

Figure Page

CHAPTER I

INTRODUCTION

1.1 INTRODUCTION AND PROBLEM STATE

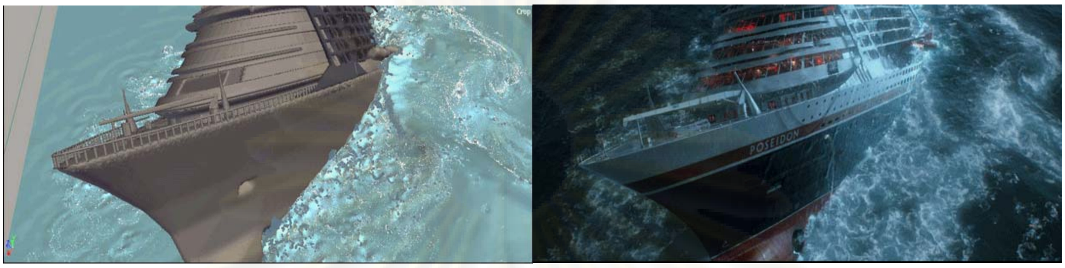
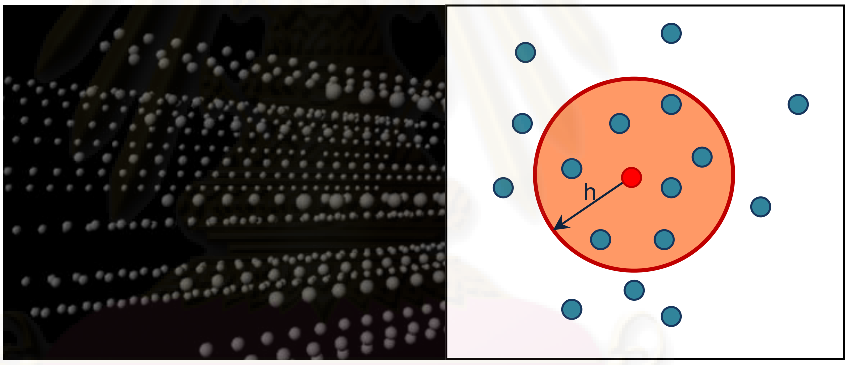


Figure 1-1 : Fluid animation (Ocean wave) in the movie, *Poseidon*

Virtual Studios, © 2010 Warner Bros. Entertainment Inc.

In a computer graphics animation, a realistic simulation of natural phenomena has been continuously developed for several decades. Physically based models for different states of substances have been applied in general in order to permit animators to almost effortlessly create interesting, realistic, and sensible animation of natural phenomena such as water flow, smoke spread, etc. For this reason, several techniques are formulated for providing simulation. Fluid dynamics bases are used to solve the problems, which are describes by Navier-Stokes' equations. However, Navier-Stoke's equations are in the form of a vector field, when they are used in simulation, they must be projected to other systems for easily used in computation. The forms of the equations can vary depending on the viewpoints of fluids such as Eulerian or Lagrangian.

Point to the Lagrangian viewpoint, the continuum is treated as a particle system. A particle system is represented by a large collection of simple geometric particles containing some attributes which can be conducted; therefore, the system is proper for control. For decades, there are many techniques used for solving Navier-Stoke's equation based on particle systems. One of the mostpopular and widely used approaches is Smoothed Particle Hydrodynamics (SPH). SPH was developed by Lucy [1] , Gingold and Monaghan [2], Managhan [3, 4, 5], and [6] for the simulation of astrophysical problems and has been used more to study among other astrophysical topics. This method is a particle-based method which represents sample points that enable the approximation of the values and derivative of local physical quantities inside. It is general enough to be adopted to solve various problems. In computer graphics community, many researches also use SPH method to simulate fluid flow such as [7, 8, 9, 10, 11, 12]. However, the capability of SPH depends on its mean value that relies on kernel length. Hence, if it is defined improperly, it will waste calculation time in high density areas or particles for calculation are lack in the low density areas. To solve this problem, many researches proposed the techniques; called Adaptive SPH, to adjust the size of particle. Size adaptation leads particle to be fit with the density of its region. The main idea is to split and merge particles; enlarged size in dense area and reduced size in sparse area [13, 14, 15].



a b

Figure 1-2 : (a) shows particles system; (b) illustrates the concept of Smoothed Particle Hydrodynamics

In another direction to sort out the problem, some previous works proposed the method of adaptive SPH by adjusting the kernel length. In physical fluid dynamics field, [16, 17, 18, 19] provided an alternative technique to improve the performance of numerical solutions of dynamical problems. The technique provided is used for representing functions and derivations with adaptive parameters which can be automatically adjusted to optimal values regarding the location of the particles. To couple the technique with particles method which is based on SPH, the considered parameter is the smoothing length or influence radius; h . The key advance of SPH which associates with each particle is a smoothing length representing the finite spatial extent of the particle; the smoothing length (or radius) can differ in value to separate particle, as well as vary in time. By varying the kernel radius, it is possible to achieve significant improvements over the fixed kernel radius approaches. The substantial improvement on the approximation of functions and derivatives is obtained by allowing the kernel parameters to dynamically accommodate, both globally and locally, to all given points.

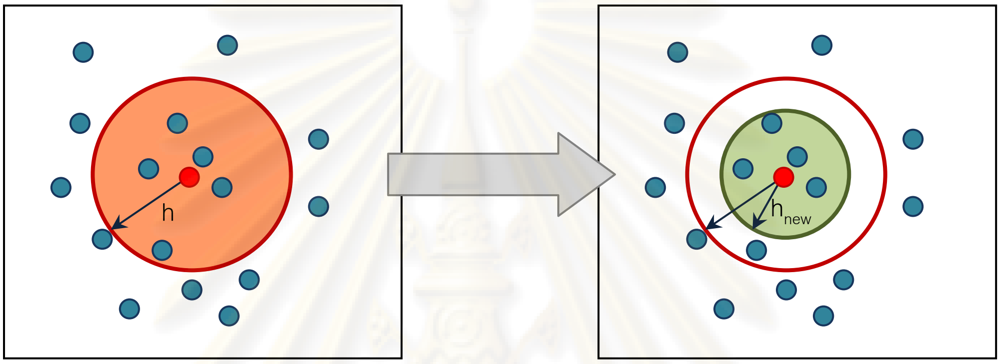


Figure 1-3 : illustrates the adaptive kernel length method

By the way, taking an interest to the various challenges in the fluid field, we attempt to simulate fluid-like motions which are controllable by user or animator, since applying the techniques to commercial animations such as advertisement and film. Many developments have been proposed to model controllable fluid simulation with the need in realistic motion, robustness, adaptation, and support more required control model.

Figure 1-4 : the example of advantage of controllable fluids such in films, Tar Monster in Scooby Doo 2

The basic of control is to add the force called "*control forces*" to the system in order to adjust the direction of particles be in the line with user's requirement. Focus on control fluid with Lagrangian viewpoint; with the characteristic of particles system, each particle has its own properties i.e. velocity, density, pressure etc. For this reason, it is convenient to control. There are many researches which proposed the fluids controlling methods based on particles system [20, 21, 22]. It is usually called '*particle control*' method.

There are various methods to control fluids; in general, separated into 2 types: path-line control and object control. The path-line or motion path control is a technique which allows the users to define a direction curve to lead the fluid movements. The object guiding control is the technique which uses the boundary of an input model to be a control guide attracting fluid movements. Both of the methods are easy for control and implementation.

Although the simulating controllable fluid based on particles system is effective, we found that many controlling methods based on SPH have some limitations. For the SPH method, the average velocities are approximated from the neighboring particles which are in influences radius. It leads to poor estimations because of the fixed kernel estimation. This method; therefore, tends to become unstable in regions of low density such as the tails, due to the lack of particles to approximate the required functions. This causes the visual artifacts. Another problem which occurs is that it can not be automatically choose the number of control particles.

In this research, we focus on fluid-like motion simulation which can be used with key-frame, path-line or script specified by user. We propose an enhanced method to simulate the fluid motion of free surfaces based on Smoothed Particle Hydrodynamics (SPH) which is controllable and automatically kernel length adjustment in order to preserve small details; lost by fixed kernel, in low density regions.

Recently, new techniques to simulate controllable fluid have been developed based on the couple concepts between particles control and object guiding [23, 24, 25, 26, 27]. To archive our goal, we use the concept of particles control called

Skeletal Particle proposed by [21] which can be implemented with both key-frame and path-line. Since the advantage of using Reeb Graph to extract the skeleton, the object's topologies are preserved. Moreover, we obtain critical points for placing initial control particles. In addition, we implement Adaptive SPH by adjusting the kernel length method to solve the problem in low density regions. In order to simplify constraint for kernel length adaptation, we make use of object's contours in the same level of control particles; critical points, to represent as guiding object. The smoothing length of each skeletal particle is automatically adapted in every time step. For this reason, the errors of unstable mean values of particles being in low density regions are decreased.

1.2 OBJECTIVES OF STUDY

The main objective of this research is to propose the fluid control method based on Smoothed Particle Hydrodynamics (SPH) by using skeletal particle control method. The skeletal particle method is the effective model which can be controlled by user. However; by using SPH approach for interpolation, fixed smoothing length will be the cause of the visual artifacts in the result of animation. Therefore, we apply the adaptive kernel density estimation coupling with the constraint obtained from object's contour into the process in order to automatically adjust the smoothing length. As a result, it can preserve the small details by changing the influence radius every time step.

1.3 SCOPE OF STUDY

This research will study only in enhancing control of fluid simulation for incompressible fluids, such as water, smoke and gas. The basic approaches used to simulate controllable fluid are SPH and skeletal particles method which generated in an isothermal system. Our considerations are controllable and details preservation.

1.4 EXPECTED BENEFITS

We expect that our controllable fluid simulation method would provide ease and more accurate to animator to create various desired movements of fluid.

1.5 RESEARCH PROCEDURE

1. Review related researches in the field of study.
2. Study about the equation of the fluid flow, Navier-Stokes equations, and other related equations and theories, such as numerical estimations and Eulerian-Lagrangian approach.
3. Study about the fluids simulation control and Smoothed Particles Hydrodynamics (SPH) model, including SPH improvement.
4. Study about particle control method and Reeb graph.
5. Study about SPH-based simulation control and adaptive kernel.
6. Design and develop the appropriate mathematical model for approximation.
6. Implement SPH-based controllable fluid simulation with our proposed enhancement.
7. Visualize the results.
8. Compare the resulting fluid animation with previous controlling fluid simulation, such as accuracy; preserving detailing, and controllable.
9. Analyze and conclude the work.

1.6 OVERVIEW OF THIS THESIS

CHAPTER I	Introduction
CHAPTER II	Background: All kind of definitions and relevant theories
CHAPTER III	Literature reviews
CHAPTER IV	Proposed method
CHAPTER V	Implementation, results, and discussion
CHAPTER VI	Summary and future works

1.7 PUBLICATIONS

Some parts of this research had been accepted to publish in the Engineering Journal, vol.14, Issue 2, April 2010; published by the faculty of engineering, Chulalongkorn University. The title is “Reviews on Physically-based Controllable Fluid Animation”. The authors are Saithip Limtrakul and Pizzanu Kanongchaiyos.

CHAPTER II

BACKGROUNDS

In this chapter, we describe all kind of definitions and relevant theories which are used in this research.

2.1 Fluid

Figure 2-1 : an example of fluid in the nature

Fluid is substance that its primary characteristics are indefinite shape; the shape easily changing such as water and smoke, and capable of flowing. Commonly, the basic of fluids properties are represented as follows; Mass; m , and volume; V , are usual attributes of substance. Velocity; v , is a representation of fluid speed. The next is pressure; P , illustrate the force field. Density; ρ , explains dense of fluid. The last, kinematic viscosity; ν , it measures how viscous the fluid is.

Firstly, we will consider the volume of the fluid. Types of fluid can be classified by volume properties as either compressible or incompressible fluid. The fluid is called '*compressible fluid*' if its volume in the system can be changed. On the other hand, if the volume of fluid in the system is constant throughout the time, it is called '*incompressible*' fluid. Another consideration, point at its density, we call the fluid be '*Homogenous*' if its density is constant. In the other words, the system is isothermal that means the temperature is constant.

For the state of the fluid flow consideration; the flow of fluid which is affected by its viscosity called '*viscous flow*'. It occurs; for example, oil paint or mud.

The opposite, the flow which is softly affected by the viscous feature called '*inviscid*' flow. In addition, the movements can be divided into '*turbulent*' and '*laminar*'. The smooth flow is called laminar flow; conversely, the chaos flow is called turbulent. The example is shown in figure 2-2.

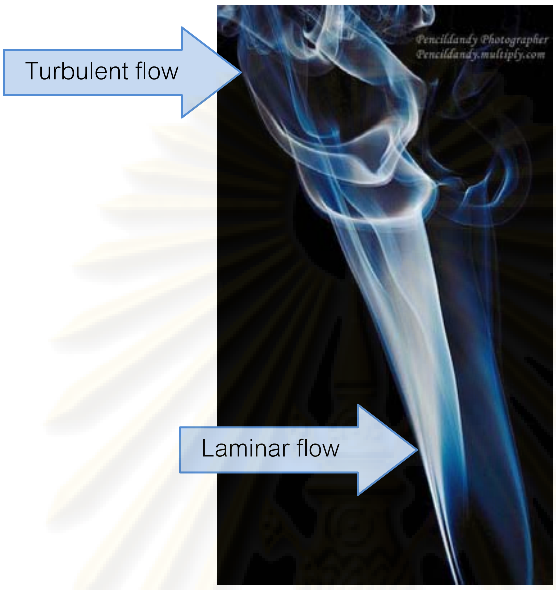


Figure 2-2 : the states of fluid flow

(source image: http://pencildandy.multiply.com)

2.2 Fluid Simulations and Animations

In our everyday lives, fluids can be found everywhere. Its existence is directly related to our routines; for example, the lukewarm vapor of hot coffee, the ripples of river's surface, and the disastrous smoke of a fire. The characteristics of fluids are, in a sense, useful in both education and entertainments fields. We require tools or simulators to simulate the fluid phenomena in order to estimate or predict the results of its movement.

Formerly; in computer animation, the animators simulate the fluid movement by using non-physically based method; do not solving the governing equations of fluid dynamics. For examples, random velocity field method which is used to generate a 3D grid of random vectors of velocity and used them for interpolation. A vortex method is similar, but forces the velocity field to contain swirly fluid-like vortices by projecting an arbitrary velocity field into a divergence-free one.

Nevertheless, people have higher requirements on the visual effect of animation. The artists or animators have to spend much time on the complex scene. Various varying movements of fluid are difficult for the animators to simulate frame by frame. Physical model, unlike key frame or procedural based techniques, allows animators to effortlessly create fluid phenomena due to its governing equation which describes the fluid behaviors. Therefore, the physically based methods became the alternative method for creating realistic fluid animations.

To simulate the fluid flow based on physical method, we must have a mathematics model for representing the state of the fluid at any time. The most significant properties for simulation are the velocity field because we can determine how the fluid moves itself. In 1845, Navier and Stokes introduced the equations for describing the behaviors of fluid, called '*Navier-Stokes' equation*'. Three fundamental governing the equations describe the following characteristics; the continuity indicating that the fluid mass conservation; momentum preserving Newton's second law, and energy conservation equation. The equations are common to make simplifying assumptions when modeling complex phenomena.

In order to solve the mentioned equations, we can use computational fluid dynamics (CFD) which is the method used for actual flow prediction. In the field of computer graphics; however, fluid simulations require far less precision than those used in physics field. Animators require the systems or tools that can be easily implemented, less computation, and give them believable result or look fairly good. So, we assume incompressible and homogeneous fluid for the simulation. The combination of incompressibility and homogeneity leads the fluid whose density and temperature are nearly constant. The assumptions are common in fluid dynamics because fluids regularly move at low speeds, whereas high speed moving phenomena are governed by other equations.

Hence, suppose incompressible and homogeneous fluid, the details of those equations are described as follow;

2.2.1 Navier-Stokes' equations

We simulate fluid dynamics on the spatial coordinate by x, which for two dimensional fluids is $\mathrm{x} = (x, y)$ and three dimensional fluids is equal to (x, y, z). The fluids are represented by its velocity field $\vec{u}(\mathrm{x},t)$. Given that the velocity and pressure are known for the initial time $t=0$, then the equations of motion for viscous incompressible fluid take the following form :

$$\nabla \cdot \vec{u} = 0 \tag{2.1}$$

$$\frac{\partial \vec{u}}{\partial t} = -(\vec{u} \cdot \nabla)\vec{u} - \frac{1}{\rho}\nabla p + \nu \nabla^2 \vec{u} + F \tag{2.2}$$

where ν is the kinematic viscosity, ρ is the (constant) fluid density, p is pressure, and F is any external forces that effect on the fluid. The equation (2.1), *continuity equation*, state that the rate at which in any small region of fluid, the total amount of mass per volume entering is exactly equal to the amount leaving volume. Thus, mass is always conserved during flow. The equation (2.1) notifies that the fluid velocity field always has zero divergence.

Equation (2.2) states that the momentum is conserved. The equation has several components which describes the change in velocity of the fluid at a given position is related to four terms; self-convection (advection), pressure, internal resistance (diffusion), and external forces. The term on the left hand side, $\frac{\partial \vec{u}}{\partial t}$, is the time derivative of the fluid velocity. On the right hand side, it can be separated into 4 terms as the following descriptions [28].:

Advection $((\vec{u} \cdot \nabla)\vec{u})$: The first term represents the *'self-advection'* of the velocity field which accounts for the direction (or velocity vector) which is changed by convection.

Pressure ($\frac{1}{\rho}\nabla p$) : The second term is pressure gradient. It states that fluid particles are push in a direction from high to low pressure.

Diffusion ($\nu \nabla^2 \vec{u}$) : The third term which is called diffusion term, describes how quickly the fluid damps out variation by measuring the parameter ν which is represents a kinematic viscosity of the fluid. The higher its value is said that thick fluids and it flow slowly.

External Forces *(F)* : The final term contains the external forces applied to the fluid. These forces may be either *local forces* or *body forces*. Local forces are applied to specific area whereas body forces act globally on the fluid.

To simulate fluid flow, the principal simulation consists of 4 steps; apply forces to the velocity field, advect the velocity field, diffuse the field to take viscosity into account, and enforce conservation of mass. Generally, there are 3 physically-based approaches used in computer graphics; e.g. Lagrangian method, Eulerian method, and Lattice Boltzmann method [29]. The details of each approach will be described in the next chapter.

2.2.2 Smoothed Particles Hydrodynamics

In order to interpolate the values, there are many researches which proposed for interpolating the equations. The most popular interpolation method for particle system is Smoothed Particle Hydrodynamics (SPH). The details of this method are described as follow.

Smoothed Particle Hydrodynamics (SPH) is one of the easiest particles methods. The SPH formalism was firstly introduced by physicists for cosmological fluid simulation in 1977. SPH was developed by Lucy [1] and Gingold and Monaghan [2] for the simulation of astrophysical problems and have been more used to study among other astrophysical topics, i.e. large scale structure in the universe, galaxy formation, supernova and solar formation. Because of its Lagrangian nature, SPH method is general enough to be adopted to solve various problems not only in computational fluid dynamics; both compressible and incompressible flow, but also in multiphase flow, heat and mass transfer, and solid mechanics. We will consider SPH method for solving incompressible fluid in this research.

SPH is an interpolation method for particle systems. In SPH, the fluid is sampled by a set of elements called *particles*. Each particle contains some attributes or physical properties, such as local mass, density, velocity, or pressure. With SPH, the values and derivatives of continuous physical quantities can be approximated by a set of discrete particles. To achieve this, SPH distributes quantities in a local neighborhood of each particle according to a smoothing kernel.

Let a fluid represented by a set of particles $i \in \{1, 2, ..., M\}$ with position x_i, masses m_i and additional attributes A_i. A fluid property $A_S(\mathrm{x})$ at position x in space is computed by weighted sum of the fluid properties A of neighboring particles within finite distance as eq. (2.3)

$$A_S(\mathrm{x}) = \sum_{j \in N} m_j \frac{A_j}{\rho_j} W(\mathrm{x} - \mathrm{x}_j, h) \tag{2.3}$$

where $A_S(\mathrm{x})$ is the summation of interpolated particular field variables at particle i. j iterates over all particles which are in a set of neighbor of particle i notated as N, and its position x_j. A_j is the field quantity at x_j. $m_{j,}\rho_j$ are the mass and density of particle j respectively.

The kernel function $W(\mathrm{r}, h)$ is typically a radial symmetrical smoothing kernels with smoothing length, h. Moreover, it is normalized function with finite support, i.e. $W(\mathrm{r}, h) = 0$ for $\left|\mathrm{r} - \mathrm{r}_j\right| > h$ and $\int W(\mathrm{r}, h) d\mathrm{r} = 1$.

Since certain volume is represented by a quantity $\frac{m_j}{\rho_j}$. While the mass m_j is constant throughout the simulation and has same for all particles, so the density ρ_j varies and needs to be evaluated at every time step. By substitution into equation (2.3),We get the smoothed density at particle i as eq. (2.4)

$$\rho_S(\mathrm{x}) = \sum_{j \in N} m_j W(\mathrm{x} - \mathrm{x}_j, h) \tag{2.4}$$

With SPH approach, such derivatives only affect the smoothing kernel. The gradient and Laplacian of the smoothed attribute function $A_S(\mathrm{x})$ are eq. (2.5) and (2.6), respectively.

$$\nabla A_S(\mathrm{x}) = \sum_{j \in N} m_j \frac{A_j}{\rho_j} \nabla W(\mathrm{x} - \mathrm{x}_j, h) \tag{2.5}$$

and

$$\nabla^2 A_S(\mathrm{x}) = \sum_{j \in N} m_j \frac{A_j}{\rho_j} \nabla^2 W(\mathrm{x} - \mathrm{x}_j, h) \tag{2.6}$$

2.2.2.1 The criteria for selecting a smoothing kernel

Since stability, accuracy and speed of SPH method depends on choosing an appropriate smoothing kernel. In general, the chosen kernel should be symmetric and normalized and satisfy the following

A. Non-negative value

$$W(\mathrm{r}-\mathrm{r}', h) \geq 0 \tag{2.7}$$

A non-negative kernel which is described by the eq. (2.7) is desirable since it has no represent the physical meaning of any quantities i.e. mass, density or pressure.

B. Compact Support (Bounded support)

$$\int W(\mathrm{r}-\mathrm{r}', h)d\mathrm{r}' = 1 \tag{2.8}$$

As shown in eq. (2.8), the function with compact support limits the interaction range by specifying a smoothing length, h, in order to reduce the computational overhead. The kernel with compact support are bounded by finite radius, therefore its value is zero outside the smoothing length or vanished at infinity.

Figure 2-3 : The compact support of kernel [9]

C. Delta-Kronecker property

$$\lim_{h\to 0} W(\mathrm{r}-\mathrm{r}') = \delta(\mathrm{r}-\mathrm{r}') \tag{2.9}$$

Equation (2.9) mimics the Dirac δ-function in the limit $h \to 0$. This condition ensures convergence of the model

D. C^1 consistency

$$W(\mathrm{r}-\mathrm{r}',h) \in C^1(R^n) \tag{2.10}$$

The kernel function has to be at least singly differentiable as being represented in eq. (2.10).

There are various smoothing functions designed for interpolating different fluid attributes, more details are explained in [10].

2.2.2.2 Modeling Fluids with Particles using SPH

Consider the first two equations for fluid computation, Navier-Stokes equation and the continuity equation are eq. (2.11) and (2.12), respectively.

$$\rho\left(\frac{\partial \vec{u}}{\partial t}+\vec{u}\cdot\nabla\vec{u}\right) = -\nabla p+\rho\vec{g}+v\nabla^2\vec{u} \tag{2.11}$$

$$\frac{\partial \rho}{\partial t}+\nabla\cdot(\rho\vec{u}) = 0 \tag{2.12}$$

Thus, the acceleration a_i of the particle i we have,

$$\mathrm{a}_i = \frac{du_i}{dt} = \frac{1}{\rho}\left(f_i^{pressure}+f_i^{viscosity}+f_i^{external}\right) \tag{2.13}$$

where $f_i^{external}$ in eq. (2.13) are external body forces such as gravity force, surface tension force or control forces.

Substituting eq.(2.3) into the above equations. By application of the SPH rule, the pressure term, $-\nabla p$, yields

$$f_i^{pressure} = -\nabla p(\mathrm{r}_\mathrm{i}) = -\sum_j m_j \frac{p_i+p_j}{2\rho_j}\nabla W(\mathrm{r}_i-\mathrm{r}_j,h) \tag{2.14}$$

Equation (2.14) is a various simple solution and suitable for our purpose of speed and stability.

Because the pressure at particle locations has to evaluate, we use a modified ideal gas suggested by Desburn.[7]

$$P_i = k(\rho_i-\rho_0) \tag{2.15}$$

The state eq. (2.15) was introduced to make the simulation numerically more stable.

However, we can use the following state equation which is feasible for compressible fluid too, defined as eq. (2.16).

$$P_i = \frac{\rho_i c_0^2}{\gamma}\left(\left(\frac{\rho_i}{\rho_0}\right)^{\gamma} - 1\right) \qquad (2.16)$$

Where ρ_0, c_0 and γ are reference density, speed of sound at the reference density, and the ratio of specific heats which depends on particular fluid being simulated. γ is taken as 7 usually employed for water simulation and as 5/3 for mono-atomic gas.

Substituting (2.3) into the Navier-Stokes equations again. By application of the SPH rule, the viscosity term; $\mu\nabla^2\vec{v}$, yields eq. (2.17).

$$f_i^{viscosity} = v\nabla^2 u(\mathrm{r}_i) = v\sum_j m_j \frac{u_i - u_j}{\rho_j}\nabla^2 W(\mathrm{r}_i - \mathrm{r}_j, h) \qquad (2.17)$$

Another equation, SPH approximation of the continuity equation is formulated as eq. (2.18),

$$\frac{d\rho_i}{dt} = \rho_i \sum_j \frac{m_j}{\rho_j}(u_i - u_j)\nabla W(\mathrm{r}_i - \mathrm{r}_j, h) \qquad (2.18)$$

where ρ_i is the current density, ρ_0 is the initial density and k is the stiffness of the fluid.

2.3 FLUIDS CONTROL

In the graphics community, animators not only need realistic animation, but also need a desirable fluids dynamics. They would like to control fluid behaviors as their imagination. Consequently, fluid control methods should be evaluated with the following criteria [30]:

- *Control capability*: A control method should be able to impose the fluid movement to approximately satisfy the constraints which are given by user-specifications, such as key frames or target shape.
- *Ease to use:* The method should be able to produce desirable fluid animations without complex computation or excessive user interference.
- *Fluid- like motion*: The control method should preserve the natural movement of fluids as much as possible.
- *Stability:* The fluid movement controlled by the method should be stable without obvious oscillations.

For control, we have to interpret the control forces in order to insert into the simulation steps. In general; the force of control is usually interpreted to one of these 3 constraints; as an external forces function, as a velocity constraint, or as a direct constraint of velocity. We have to satisfy that it has tradeoff between control and realism.

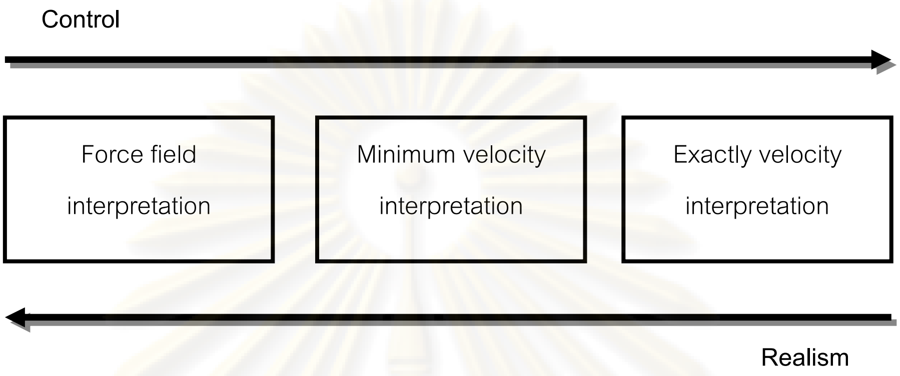


Figure 2-4 show tradeoff between control and realism

Foster and Mataxas [31] firstly introduced controllers for animating phenomena. In the process to simulation, these values are changed; fluid properties, external pressure field, internal pressure field, velocity field and, boundary properties. Consequently, we can apply to those processes in order to control the fluid movements. Recent techniques on fluids control have corresponded physic-based fluid dynamics.

2.3.1 Particles control

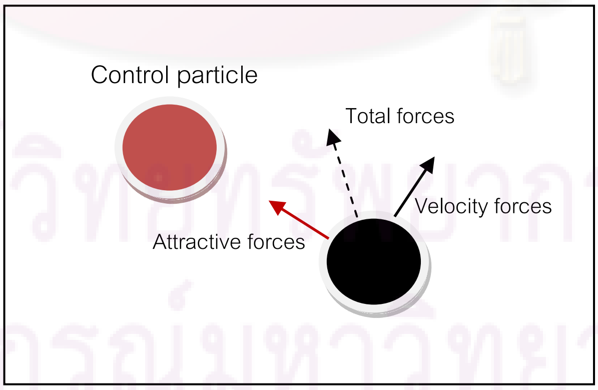


Figure 2-5 : show elements of the total forces

Since, we take an interest in Lagrangian viewpoint, the simulation is based on particle system. In order to control, some particles are supposed to be representatives or guiding of the movement of their neighbor; also called control particles.

To define the movement of fluids, we usually group constrained motions into path-defining control and object guiding control. Path defining control allows the user to specify the direction of fluid movement by a curve or a line; as show in the following figure.

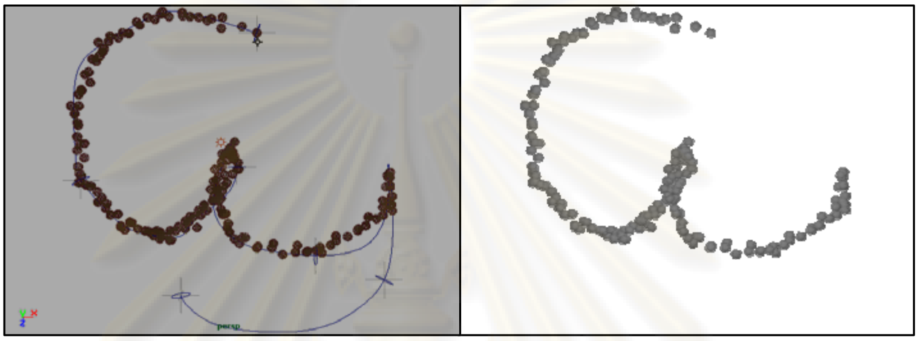


Figure 2-6 : path defining control

Another specified motion constraint; object guiding control or target shape constrained, allows the user to provide the destination shape to guiding the movement of fluid flow.

Figure 2-7 : Object guiding control [26]

According to the previous researches, they introduced the technique for reducing the detail of an input object by representing as its skeletal path. [21] used Reeb Graph for constructing the skeleton of the object, and used it as a control path. [13] use the technique called medial axis. They use the axis as boundary constraints to determine that which particle should be split or merged. By using the path representation, the cost of computation time will be decreased. In addition, it can be controlled easily.

2.4 Reeb Graph

The Reeb graph is a compact representation of the topological structure of a three-dimensional object. Its definition is based on Morse theory which states that the topology of a given manifold M can be studied by analyzing the critical points of a real valued function $f : M \rightarrow R$. The Reeb Graph of f is the quotient space of the graph of f in $M \times R$ by the equivalence relation "~" given below

$$(x_1, f(x_1)) \sim (x_2, f(x_2))$$

The above relation holds if and only if $f(x_1) = f(x_2)$ and x_1 , x_2 are in the same connected component of $f^{-1}(f_1)$. That is the Reeb Graph represents the two points, $(x_1, f(x_1))$ and $(x_2, f(x_2))$ as the same node if the value of f are the same and they belong to the same connected component of the inverse of $f(x_1)$ or $f(x_2)$. The graph represents all points that belong to the same equivalent class of the original space as a node in the quotient space.

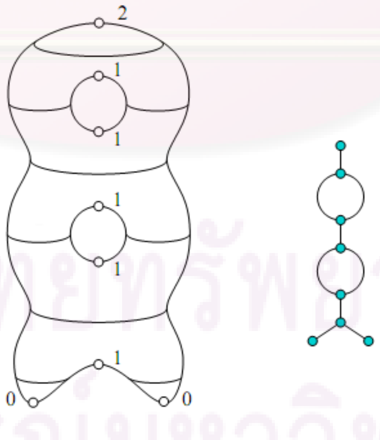


Figure 2-8 : Reeb graph representation [32]

The extension of an original Reeb Graph on a height function is made to hold not only topological information, but also geometrical information of an object. Its node has the coordinates of a critical point [33]. Its edge represents the connective components of an object and the graph has cross-sectional information of an object.

2.5 Adaptive Kernel Density Estimation

Adaptive kernel density estimation [18,19] is applied to the low-density regions. The basic idea is to construct an estimating consisting of a collection of kernels placed at the skeletal particles in order to allow the smoothing length of the kernel to vary form position to position. We simply use eq. (2.19).

$$\rho_i = \sum_j m_j W(\mathrm{x}_i - \mathrm{x}_j, h_0) \tag{2.19}$$

to estimate an initial density, where h_0 is chosen by reference to an initial distribution.

Then, local smoothing length, λ_i , are defined according to the relation

$$\lambda_i = k\left(\frac{\rho_i}{\vec{g}}\right)^{-\varepsilon} \tag{2.20}$$

where $\vec{g}$ is the geometric mean of the density. The value of $\vec{g}$ in eq. (2.20. is given by eq. (2.21).

$$\log \vec{g} = \frac{1}{N}\sum_{b=1}^{N} \log \rho_b \tag{2.21}$$

k is constant scaling factor and ε is the so-called sensitivity parameter defined in range $0 \le \varepsilon \le 1$. As a final step, the smoothing length of the kernel placed at the location i defined as $h_i = \lambda_i h_0$ and recalculating the density by replacing h_0 with h_i.

In order to ensure conservation of linear momentum and total energy, the actual kernel estimate which use to update the density, velocity of the particle modified by replaced h_i by the average mean as shown in eq. (2.23).

$$h_{ij} = \frac{1}{2}(h_i + h_j) \tag{2.22}$$

So that, the kernel employed to evolve the density and velocity of the particle is

$$W_{ij} = \frac{1}{2}\left[W\left(\mathrm{x}_i - \mathrm{x}_j, h_i\right) + W\left(\mathrm{x}_i - \mathrm{x}_j, h_j\right)\right] \tag{2.23}$$

CHAPTER III

LITERATURE REVIEWS

This chapter is separated into two parts. The first part describes the previous works both in non-physically based and physically-based method. These methods are divided into subsection each approach, and will mainly focus on SPH method. The second part then explains the previous works of controllable fluid which related to our research, including limitation discussion.

3.1 The Fluid Simulation Approaches

3.1.1 Non-physically based method

Early period, the fluid animation is approached by many ways of non-physically based method. Chen and da Victoria Lobo [34] proposed high field method for simulation by solving Navier-Stokes' in two-dimensional; then, push the high field up and down depending on the pressure to create the third dimension. Witting [35] also solved Naveir-Stoke's equations in two-dimensional system and created the third-dimension by using approximating scheme; forth order Runge-Kutta. The methods are easy to simulate, nevertheless, the methods do not support complex scene or complicated flow, such as turbulent flow.

3.1.2 Physically based method

The physically based methods became the alternative method for an animator to create realistic and complex fluid animations. Several researches use physically-based method for simulating the fluid phenomena. In this subsection, we give a brief review of the researches for solving the Navier-Stokes' equations in any viewpoints; Eulerian, Lagrangian and others.

3.1.2.1 Eulerian approach

Eulerian approach is the method which uses spatial coordinates to describe the system. The workspace is discretized equally as grid. The change in system is being tracked at fixed grid points.

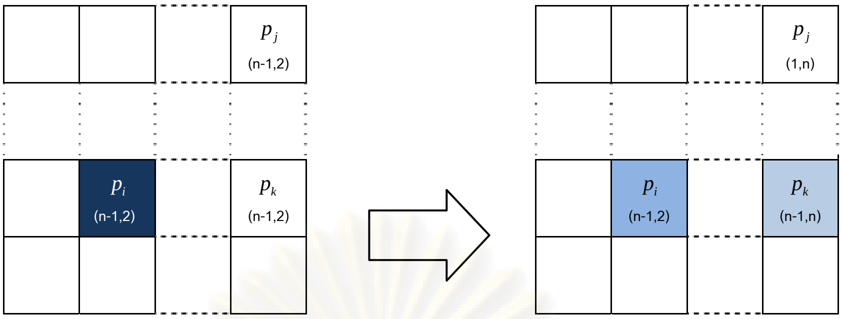


Figure 3-1 : the state of fluid simulation represented on spatial grid size nxn

As shown in the above figure, a supposed fix grid points are, p_i which its own property at the beginning is dark blue, p_j and p_k have white property. In the next time step; shown in the right-hand side, the result of the observation is that the property of p_i and p_k has been changed while p_j still has the same property as that in the previous time.

One of the most commonly used techniques is Maker-And-Cell method (MAC) proposed by Harlow and Welch [36]. This method has two major components. The first one is the cells which are formed with the uniform size of a cube, or voxel, storing two types of variables; a scalar value and a vector value. All scalar values are contained at the center of the cell while all vector values are stored on the cell faces describing components such as x, y, z. Another component is a large collection of maker particles in the fluid that mark which cells are filled with fluid and that carry velocity to the previous empty cells. A major strength of their method is that liquid is no longer constrained to be a height field, as demonstrate by their animations of pouring and splashing.

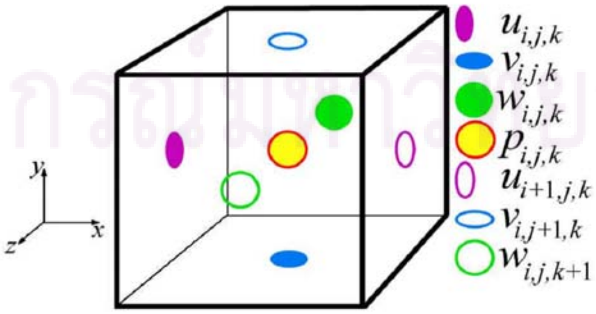


Figure 3- 2 : MAC grid representation [37]

Grid-based fluid simulation has been continuously developed in computer graphics community. One of the notable works was proposed by Stam [38]. An unconditionally stable model was introduced. They use a combination of a semi-Lagrangian scheme and implicit time integration. However, this model suffers from too much numerical dissipation which can cause visual artifact. The flow tends to dampen and vanish rapidly, thus, the small scale's detail is missing. To simulate visual of smoke, [39] proposed the method that exploits physical unique to smoke in order to design a numerical method that is both fast and efficient on the relatively coarse grids. Their model is not only stable, but also does not suffer from numerical dissipation.

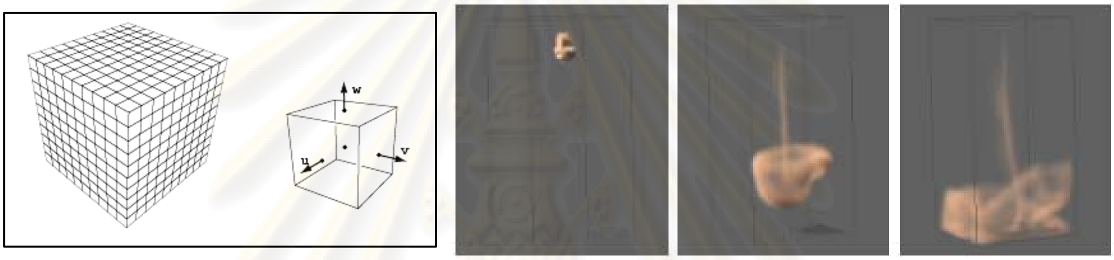


Figure 3-3 : (left) shown discretized domain and each voxel's components and (right) their result [39]

By using grid representation, the finer details are depending on grid size. Recently, many researchers proposed developed techniques based on grid-based approach to adjust the grid size, such as [40] who proposed the method called Octree for the simulation. [41] presented a new approach to simulate fluid flow that refines high-resolution data into fluid tiles in order to capture spatially localized fluid behavior. Their method is extremely fast and scalable to large domains. Similar to other model reduction for fluid simulation, this method cannot be used for simulating multi-phase flows; moreover, fluid with free surfaces cannot be properly handled.

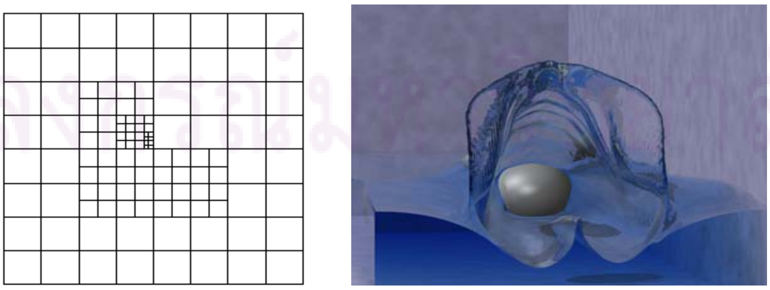


Figure 3-4 : (left) an example of initial quadtree refinement [41], (right) the exampled result by using Octree data structure [40]

Although grid based method is efficient and rather fast, this approach is still difficult to implement for control. Moreover, it is also difficult to simulate small scale of details due to its scale ability.

3.1.2.2 Lagrangian approach

The system which is described by Lagrangian viewpoints use material coordinates to introduce the change in system is being tracked by the position of each particle. In order to solve the equations by using Lagrangian approach, the continuum is treated as a particle system. Particle system was introduced by Reeves [42] in 1983. After that, this method has been widely used for modeling fuzzy objects such as fire, water, and clouds. The concept of this system is modeling an object as a large collection of simple geometric particles that define its volume and contain some attributes such as mass, density, velocity, and pressure. Over a period of time, particles are generated into the system; move and change the form within the system, and die from the system.

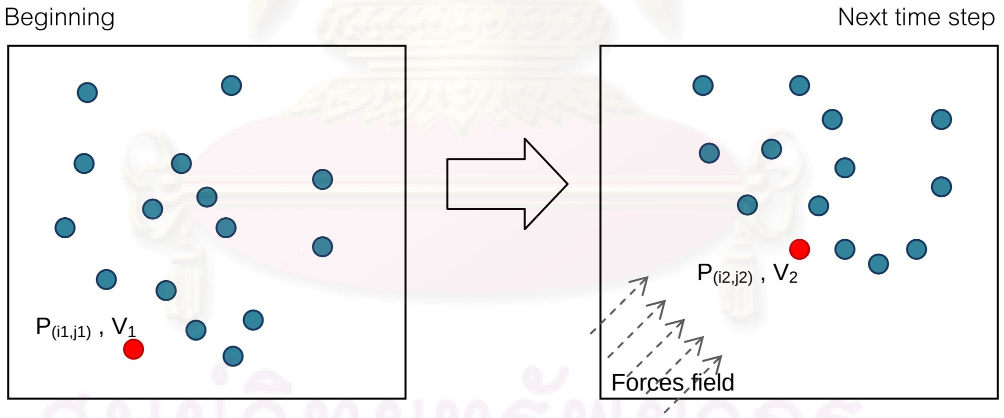


Figure 3-5 : Particle system, ● is tracked particle representing its position and its velocity

In the previous works, some typical particle methods or particle-like methods were introduced, e.g. Particle in Cell method (PIC) [43], Fluid in Cell (FLIC)[44], and Moving Particle Semi-Implicit (MPS) [45].

Currently, the particle based method which is popular and widely used is Smoothed Particle Hydrodynamics (SPH). SPH was developed by Gingold and Monaghan [2] and Lucy [1] for the simulation of astrophysical problems and has been used more to study among other astrophysical topic. This method is a particle-based method which represents sample points that enable the approximation of the values and derivative of local physical quantities inside. It is general enough to be adopted to solve various problems. Also in computer graphics, many researches have been proposed by using SPH method to simulate various fluid flows. [10] proposed SPH based fluid simulation for interactive applications. They derived force field, pressure and viscous force, directly from the Navier-Stokes' equations and added a new term to model surface tension force for tracking and rendering the free surface of fluids. They also introduce special designed kernel to increase stability and speed. Although the simulation takes minutes or hours per frame; by given the off-line simulation, the results are quite promising. [11] proposed a new technique to model fluid-fluid interaction based on SPH method. It makes possible the simulation of boiling water, trapped air and dynamics of a lava lamp, but, the limitation of SPH approach for single particle lead to lose small details.

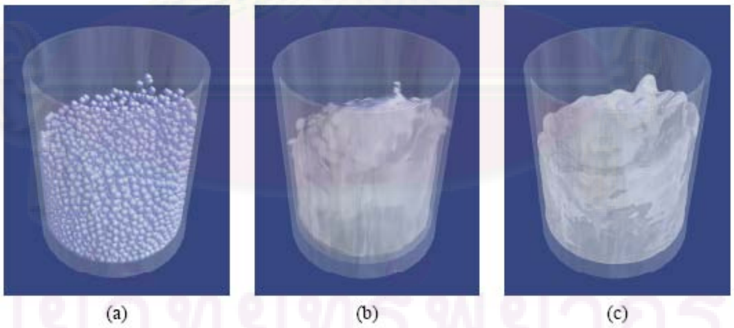


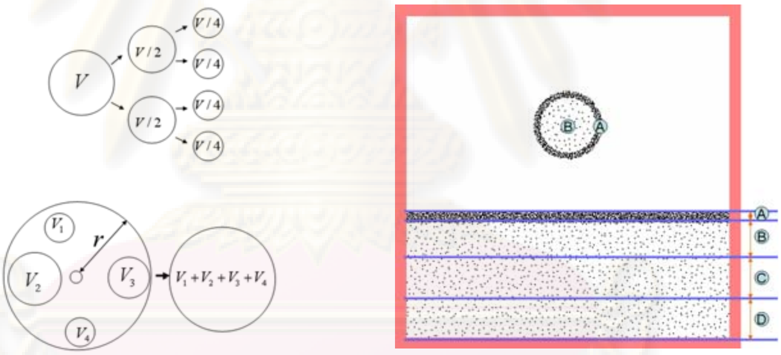
Figure 3-6 : [9] (a) shows the particles, (b) its surface using point spattering, and (c) the iso-surfaces triangulated via marching cube

Some researches use SPH to simulate other types of substance which is similar to fluids such as sand. [46] proposed the work which simulating sand as fluids. They represented sand as a cloud of particles and turned the water simulator into a sand simulator by adding the account for inter-grain and boundary friction.

A. SPH Enhancement

In order to improve SPH method which suffers from unsuitable size of particle or fixed kernel length, many researches introduced the technique for enhancing the method. [47] proposed the technique for improving the accuracy of particle approximation by considering a shape function which constructed by using Moving Least-Squares (MLS) and Redial Basis Functions.

As being introduce in the first chapter, there are many researches proposed the techniques called Adaptive SPH, for adjusting the size of particle. Size adaptation leads particle to be fit with the density of its region. The main idea of this technique is consisted of splitting and merging. Splitting technique allows larger usage of particles in an appropriate area in order to represent the fluid surface more accurately. Merging is mainly performed away from the surface in the regions of low turbulence for computational reduction.



(a) (b)

Figure 3-7 : (a) splitting and merging particles,

(b) defined layers for splitting and merging [14]

Because of the concept of splitting and merging, the researches which proposed the adaptive SPH have to introduce the considered constraints. [14] proposed a technique for simulating incompressible fluid that integrates adaptive refinement of particle sampling. Particles can be split into several particles in highly deformable areas for a detailed representation; similarly, in regions of smooth flow, nearby particles can be merged into a single particle. They used the criteria in order to determine the level of particle sampling needed in a region of fluid. They quantify what they call the "*deformablity*" of each region of the fluid and then decide which particle splitting and merging on this measure. To find that quantity, they consider depth or distance from the surface and the local Reynolds number. Depth is determined by constructing a signed distance field from the fluid interface as determined by particle’s positions. The simulation domain; however, has to be divided into a set of layers based on the distance from surface in order to define the level of detail, it is not suitable for the flow modeled by guiding object.

[13] proposed an adaptive sampling algorithm by introducing a sampling condition based on geometric local feature size. The main contributions are decreasing computational resources and reducing the number of particles deep inside. They also proposed a novel fluid surface definition based on approximate particle-to-surface distances which the distance field constructed by using medial axis. Similarly, [15] proposed the method for adjusting the size of particles by considering these 3 constraints; geometric complexity, physical complexity, and complementary conditions. Since the hypothesis, which is dense particle, is usually in the middle of the flow, geometric complexity is a condition that measures how far the particles are from the surface. Secondly, physical complexity is a condition based on the concept that turbulent flow has high pressure; therefore, the particles in the regions have to be small size. The last condition; complementary condition, is given from the assumption that finer details are needed in low density regions.

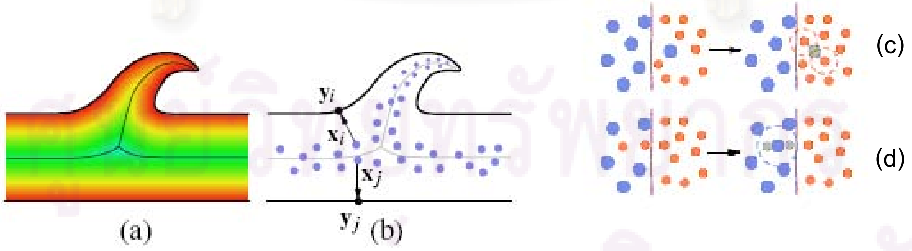


Figure 3-8 : (a) and (b) shows distances fields and approximated particles of medial axis, (c) and (d) shows how to determine splitting and merging [13]

The other direction for preserving detail in low density region which caused by SPH limitation, some previous works proposed the method of adaptive SPH by adjusting the kernel length. In physical fluid dynamics field, [18, 19] provided an alternative technique; called Adaptive Kernel Density Estimation (AKDE), in order to improve the performance of numerical solutions of dynamical problems. The technique provided for representing functions and derivations with adaptive parameters which can be automatically adjusted to optimal values according to the location of the particles. To couple the adaptive kernel method with particles method which is based on SPH, the considerable parameter is the smoothing length or influence radius; h . Due to the key advance of SPH which is associated with each particle is a smoothing length representing the finite spatial extent of the particle, the smoothing length (or radius) can differ in value for separate particle, as well as vary in time [17]. By varying the kernel radius, it is possible to achieve significant improvements over the fixed kernel radius approaches. Substantial improvement on the approximation of functions and derivatives are obtained by allowing the kernel parameters to dynamically accommodate, both globally and locally, to the all given points. In addition to adjusting symmetric radius of kernel, [48] presented the algorithm with anisotropic smoothing kernel which resolves much better than standard SPH whenever anisotropic collapses or expansion occurs.

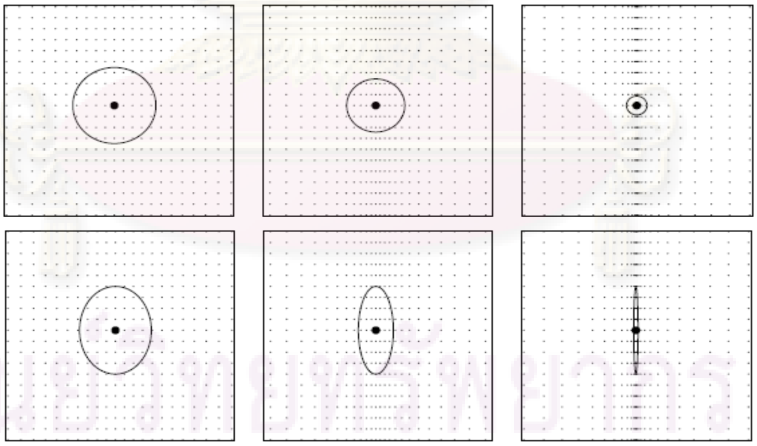


Figure 3-9 : the above shows anisotropic contraction with standard SPH (symmetric radius of kernel) whereas the bottom shows the contraction with adaptive SPH

However, either the adaptive kernel density estimation method or adaptive smoothed particle hydrodynamics also consumes expensive cost of computation. To adjust the kernel length; for instance, the local size of kernel has to be computed before adjusting the global length. In computer animation; however, it might be exceeding complexity. To apply the method in computer graphics, we can simplify it by adding some constraints. More details are described in Chapter IV.

3.1.2.3 Other approaches

Besides those two main approaches, there are a lot of proposed methods for simulating fluid phenomena. Cellular Automata is a method usually used for simulating gas [49]. Similarly, Lattice Boltzmann Method is also widely used such in [50], [51] for non-Newtonian fluid flow. A spectral method is also used for turbulent flow simulation [52, 53]; however, this method cannot easily represent complex boundaries of free surfaces. Finite elements method [54] is also computationally expensive and complex.

Nevertheless, recent researches tend to use coupling method to simulate the flow of fluid. According to the different features of those principal approaches, researchers are willing to combine their advantages while avoiding the raising limitations.

The particle level set [55] was created to compensate some of the inaccuracies while advecting level sets by adding Lagrangian particles to the simulation and letting them correct the first order error with the inherent high resolution detail near the front where the particles contained. Particles were used both inside and outside the surface of fluids; called interface, in order to correct the error in surface representation. In 2004, [56] proposed a method for directable animation of photorealistic liquid by using the particle level-set method. The main concept of this particular work is to apply different level of velocity calculation to each region which is divided by user's specification. [57] proposed a two-way couple simulation framework that uses particle-level set method to efficiently model the simulation. The novel method allows dense areas modeled with incompressible Navier-Stokes' equations; based on grid representation, while SPH methods are used for diffuse areas. The main limitation of the approach is that using FLIP method which introduces unwanted noise. Another limitation occurs where SPH particles do not have neighbors on all surrounding sides, so particle density estimation is unreliable near the air/liquid interface.

Figure 3-10 : control fluid with particle-level set method [56]

3.2 Controllable Fluid

According to user requirements, the methods for controlling fluid movement have been proposed. The first developed technique has been proposed to control the fluid simulation by [31]. By using the concept of an embedded controller, animators can specify and control the fluid animation without knowledge of the underlying equations. The limitations of the method occur when large time step size is used. Other problems are that the object should be simple compared with gird size and the method cannot deal with transparency. Thereafter, the developed techniques for control fluid have been continuously proposed.

[58] described a method for controlling smoke simulation through user-specified key frames. This method become computationally prohibitive for large problems with fine grained control, and get caught in local minima. [24] also proposed a method for efficiently controlling animated smoke. In order to achieve, new terms are added to the standard flow equation; Navier-Stokes equation. These terms are driving force term and a smoke gathering term. Therefore, complex smoke can be controlled and easily extended to support the usual external forces. Nevertheless, this method cannot be used to control the fluid motion by user-specified as for flow-path.

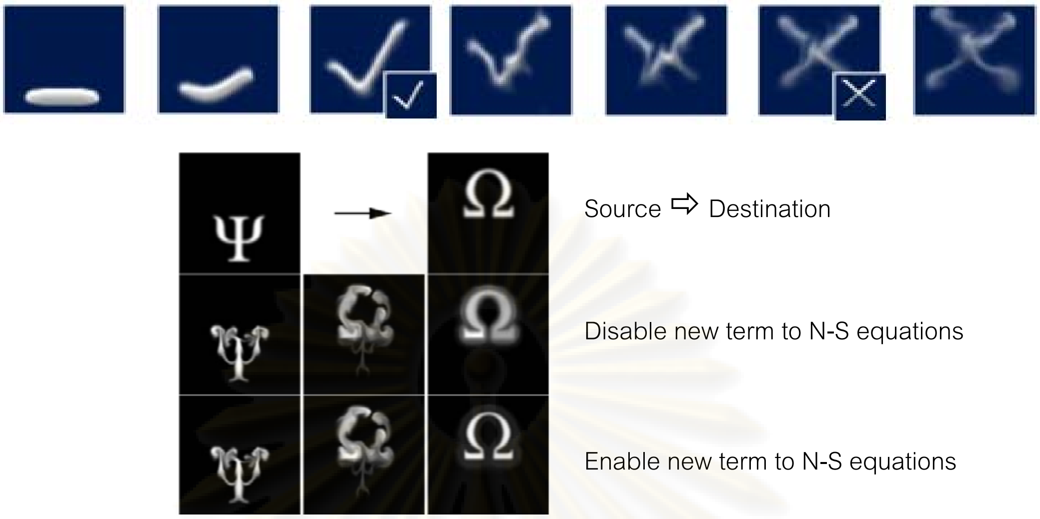
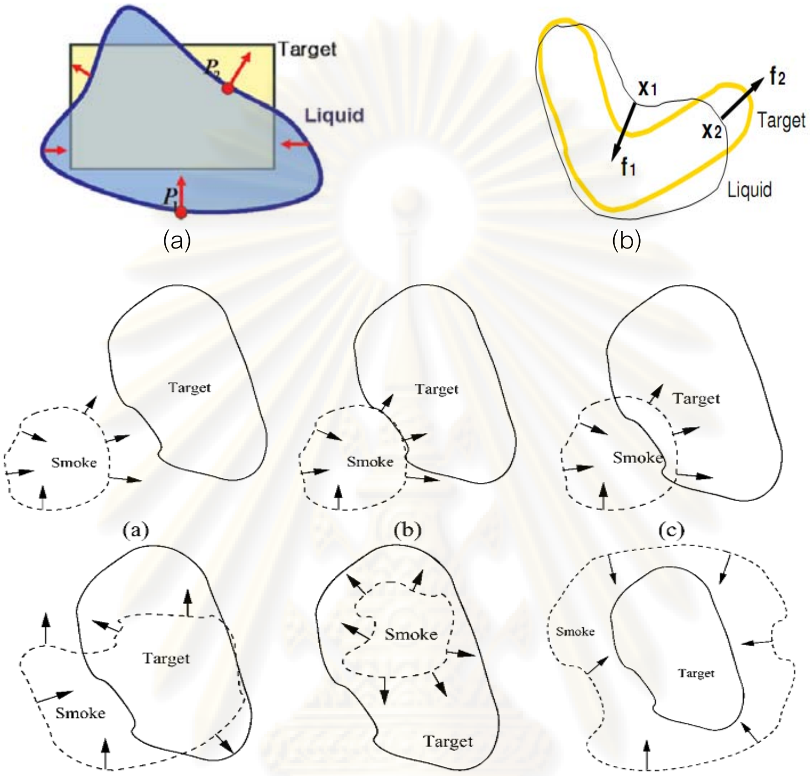


Figure 3-11 : above image shows controlling smoke through user-specified key-frames, bottom image shows animated smoke control by adding new term to N-S

A way to use key-framing method of particle motion to enhance the visual effects and user controllability of physically based particle systems is present in [20]. They introduced three general types of key-framing; position to position, density to density, and boundary to boundary, which are applicable to existing systems and able to combine to a variety of effects.

Other way for controlling fluid flow is such proposed in [59]. They used Adjoint method to control fluid. By using this method, derivations can be computed efficiently and can easily handle huge control vectors. Nevertheless, discontinuities in the water simulator make control more difficult. Moreover, it is less accurate for smoke

In order to preserve a geometric of the target-object, Jeong-mo Hong and Chang-Bun Kim [25] presented a new fluid control technique that uses a geometrically induced potential filed. A potential added as an extra dimension to the simulation space which forces the fluid to inform the target shape. [26] is also their proposed method for controlling liquids flow into a target shape by using a concept of shape feedback. The force is determined from the magnitude of pressure jump. Pressure jump; which is also used in [27], provides the force to make the fluid assuming the target shape. Instead of adding the force to an external term, the force is added at the projection step. The drawback of this method is that the forces may cause visual artifact.



(a) (b)

(c)

Figure 3-12 : the control techniques which are based on objects guiding,

(a), (b), (c) presented in [25], [26], [24] respectively.

[23] proposed the method which uses imposing velocity constraints. The smoke shapes are created in resemblance to user-specified objects and, both smoke region and the target object are represented by implicit function. The velocities constraints are derived from shape matching process. The advantages of this method trade of between controllability and smoke appearance. Another proposed technique by [30] can obtain an efficient and effective solution by applying two different external force fields. It makes use of continuous sequence of frames. However, it does not support splashing.

[60], [61] decomposed the velocities field into coarse-and fine-scale components and only apply control forces to the low-frequency part. They use low-pass

filter to adapt to the influence kernels of the control particles. However, dependence on force-based, user has to experiment for weighted constraints.

A new technique was developed by [62] based on the Advected Radial Basis Function; therefore, the local properties of the fluid are modeled by time-varying kernel. Advected particles are served as center of Radial Basis Function and then used as a calculation point in SPH. However, the limitation depends on a fitting finite number of particles and fixed radius. [21] proposed the novel technique to model a controllable fluid simulation by coupling Reeb Graph and Radial Basis Function (RBF). In addition, SPH is used to approximate flow dynamics, and the concept of skeletal particles is used for controlling. However, there are visual artifacts because of averaging velocities of fluid particles with skeletal particle. Another problem which occurs in this method is that it cannot automatically choose the number and destination of skeletal particles.

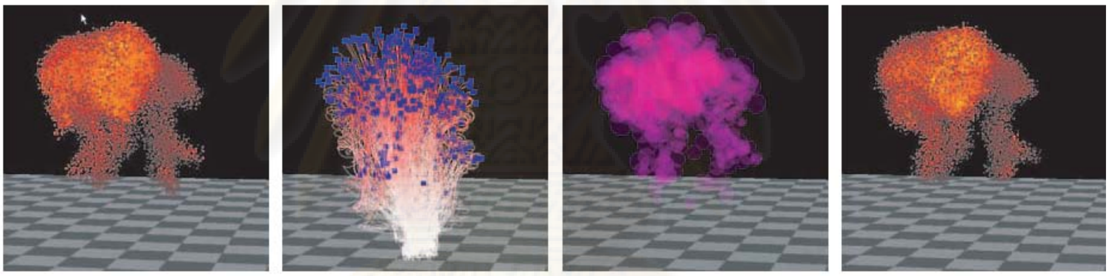


Figure 3-13 : The fluid simulation using Advected Radial Basis Function (ARBF)[62]

Figure 3-14 : keyfram control fluid simulation using skeletal particles[21]

In the field of particle control method, [22] uses particles to sample and control implicit surfaces. A simple constraint locks a set of particles onto a surface while the particles and the surface move. They implemented control points for direct manipulation by specifying particle motions, then solving for surface motion that maintains the constraints.

Despite the fact that the control techniques for supporting user's requirements are continuously developed; due to the tradeoff between physical correctness and controllability, there are still limitations and challenges for improvement.

3.3 Limitations and Challenges

3.3.1 Controllable Fluid

Our interested challenge of controllable fluid method is how to preserve the details of fluid while it can be flown along a defined movement as users require. As shown in the previous works, there are many techniques that supporting fine controllability, but trading off more visual artifacts. In this research, we will propose the method which can be controlled easily while preserve realism. Our interested method for control is skeletal control method which proposed by [21] because this method can be use to create imaginary movement of realistic phenomena while conserve object topology. Nevertheless, it is still got suffer from fix kernel length approximation; the visual artifact will occur by average velocity in low density regions.

3.3.2 SPH Improvements

Because of the problem in low density regions, we have to improve the method for both preserving small details and not increasing the cost of computation. Adaptive SPH by adjusting the size of kernel length is our alternative choice. With the advantage of object contour obtained from Reeb Graph construction, it can be profitably applied to be the constraint for the kernel length adaptation.

CHAPTER IV

PROPOSED ENHANCING METHOD

For this section, we introduce our proposed method for enhancing the previous method used in [21]. Our enhanced methods consist of skeletal extraction by using Reeb Graph and kernel length adaptation.

Firstly, we introduce overall of our procedure by following flow chart. After that, we describe more details of each step, especially skeletal extraction and SPH enhancement.

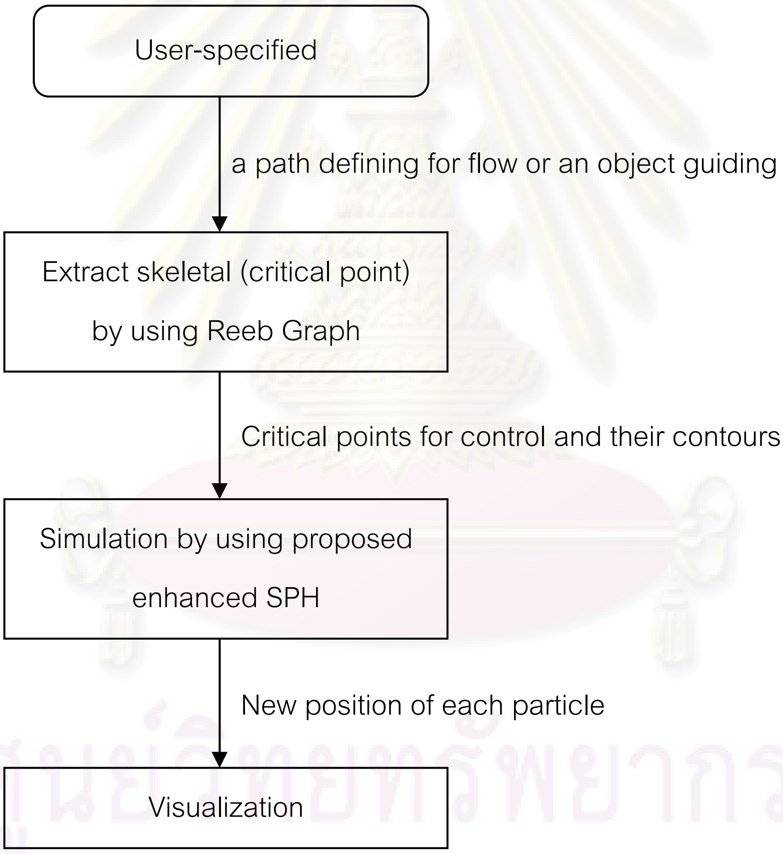


Figure 4-1 : The process of our simulation

4.1 User Specified Information

In this research, we propose the method which supports both path defining control and object guiding control. User has to define the path to represent the direction of the fluid flow. Object guiding is another type of input which defined by key-frames. The fluid movement will go along within the boundary of the model.

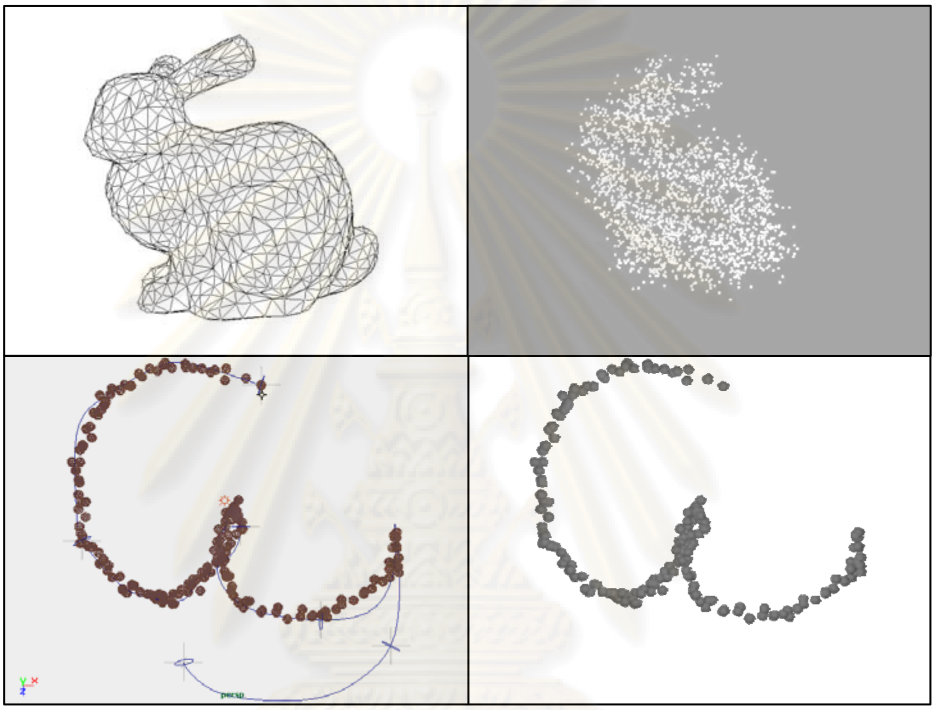


Figure 4-2 : Examples of input (user-defined information);
upper is object guiding control, and lower is path defining control

4.2 Skeletal Extraction by Using Reeb Graph

In particle control method, we have to get a set of control points in order to lead their neighboring particles movement. As we introduced in [21]; therefore, we use the contribution of using skeletal particle approach to control fluid particles. Skeletal particles are a set of control particles generated from geometrical information of user-specified key-frames in the first step.

To get the set of path-specified guiding particles for control the mechanism of the fluid movement, in our method, we extract the structure of an object or path-line defined by user by using Reeb Graph. As a result of this process, we obtained critical points which represent the topology of the object. Hereafter, we will describe the usage of the method for extraction and how to use the skeletal particles to control the fluid movement.

In our study, we use the extension of an original Reeb Graph on a distance function; modified Reeb Graph [33], which also employs in the study of translation, rotation, and scale invariant skeletal graph. The algorithm is summarized as follow.

```
ALGORITHM : Extended Reeb Graph Algorithm [33]
     DATA : Number of Level $K$, Threshold of each level (user defined)
   INTPUT : $M$ = Input model information (including vertexes and edges)
   OUTPUT : Critical points, level of contours
   METHOD :  1  Find the centroid of the surface $M$
             2  Find the maximum distance, $d_{\max}$ from the centriod to $M$
             3  Given $K$, define:
                    $r_k = k_{\frac{d\max}{K}}$ , $k = 1, 2, ....., K$
             4  Generate the spheres $S_1, S_2$ with radii $R = r_1$ and
                $R = r_2$ respectively
             5  Find the part of $M$ that lies between $S_1$ and $S_2$
                    $\tilde{M}_p = M \cap (\lfloor S_1 \rfloor \cap \lceil S_2 \rceil$
                where $\lfloor . \rfloor$ and $\lceil . \rceil$ identify the interior and exterior of a
                closed surface
             6  assign a node $N_{M_p}$ to each connected component $M_p$
                and $\tilde{M}_p$
             7  For $k = 3$ to $K$
                -  generate the sphere $S_k$ with radius $R = r_k$
                -  find the portion of $M$ that lies in between $S_{k-1}$ and $S_k$
                -            $\tilde{M}_c = M \cap (\lfloor S_{k-1} \rfloor \cap \lceil S_k \rceil$
                -  find the connected components $M_c$ of $\tilde{M}_c$
                -  for each $M_c \in \tilde{M}_c$ do
```

```
        ▪ assign a node $N_{M_c}$ at the centroid of $M_c$
        ▪ find the connected region $M_p \in \tilde{M}_p$ such
          that $M_c \cup M_p$ is a single connect region.
          Add an edge between $N_{M_c}$ and $N_{M_p}$
        ▪ end for
        ▪ $\tilde{M}_p = \tilde{M}_c$
8   end for
```

Table 4-1 : Extended Reeb Graph algorithm

However; in our experiment, we only need to show the readers how to adjust the kernel length. Thus, we implement simple Reeb Graph to our demonstration. Let M be a manifold or object's information, $m(x,y,z) \in M$. Suppose z axis is a vertical axis, and K is number of levels. Given L_k defines a set of a level k; $k = 0,...,K-1$. We define function f as a high function which given by eq. (4.1).

$$k = f(m) = \left\lfloor \frac{|z - z_{\min}|}{d} \right\rfloor \quad where \;\; d = \left\lfloor \frac{|z_{\max} - z_{\min}|}{K} \right\rfloor \tag{4.1}$$

Then, we find the critical point of each L_k by average mean value of $m(x,y,z) \in L_k$. Our algorithm is summarized as follow,

ALGORITHM	: Reeb Graph Extraction
DATA	: Number of Level; K, Threshold of each level (user defined); L_k.
INTPUT	: Input model information (including vertexes and edges); M
OUTPUT	: Critical points; c_k, contours; L_k

METHOD :

```
1   suppose $z$ axis is a vertical axis, Find $z_{\max}, z_{\min}$ of $M$
2   Given $K$, define:
        $d = \left\lfloor \frac{|z_{\max} - z_{\min}|}{K} \right\rfloor$
3   For all $m(x,y,z) \in M$, assign the level number; $k$, by
        $L_k = \{ m(x,y,z) \mid \left\lfloor \frac{|z - z_{\min}|}{d} \right\rfloor = k \}$
4   for all $k = 0,...,K-1$
```

```
        - Find the centroid of each $L_k$; $c_k(x, y, z)$, by average
          mean values,
              $(x, y, z) = \frac{1}{N}(\sum_{i=1}^{N} x_i, \sum_{i=1}^{N} y_i, \sum_{i=1}^{N} z_i);\ N = size\,of\ L_k$
    end for
```

Table 4-2 : Reeb Graph Extraction algorithm

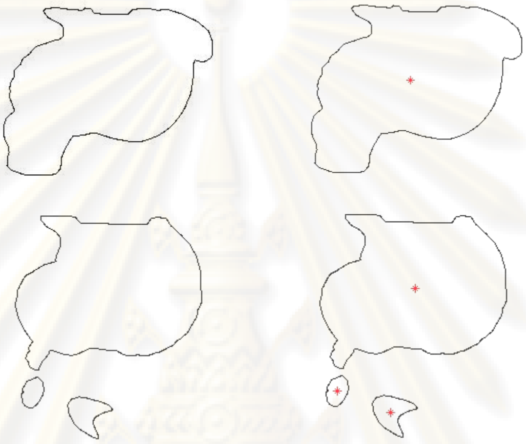


Figure 4-3 : examples of contours of the model (top view)

4.3 Generating Controlling Particles

We define initial controlling particles into 2 types:

1. *Path-line control*: First, the controlling particles are randomly chosen from the initial particles generated from an emitter. Then, the particles with the shortest distance from each critical point are changed to be representative or control particle. They will be changed in every time-step.
2. *Object guiding control*: In case that the particles formed as model or object from the beginning, we define the closest particles of critical points to be representatives.

4.4 Kernel Length Adaptation for SPH enhancement

SPH is an efficient method for fluid simulation; however, the fineness of the simulation depends on the resolution of system, i.e. the size of particles, the number of particles or the length of kernel. The more detailed simulation which requires a lot of particles or narrow kernel length, leads to the more expensive computation. Hence, many researches proposed the methods to adaptively adjust the number or size of particles; otherwise, adapt the length of kernel.

There are many situations where the fixed kernel estimation leads to poor estimates, because small details will be lost. Insufficient numbers of particles in low density region for approximation in required function causes the particle method becoming unstable. Proper kernel length modification can solve this problem; therefore, Adaptive Kernel Density Estimation is applied to the low-density regions. Kernel function plays an important role in SPH method; thus, if the kernel length is not appropriate, this will affect the properties' value distribution. We propose an enhanced method for adjusting the length of kernel by using 2 attributes; contours from Reeb Graph and density of the regions.

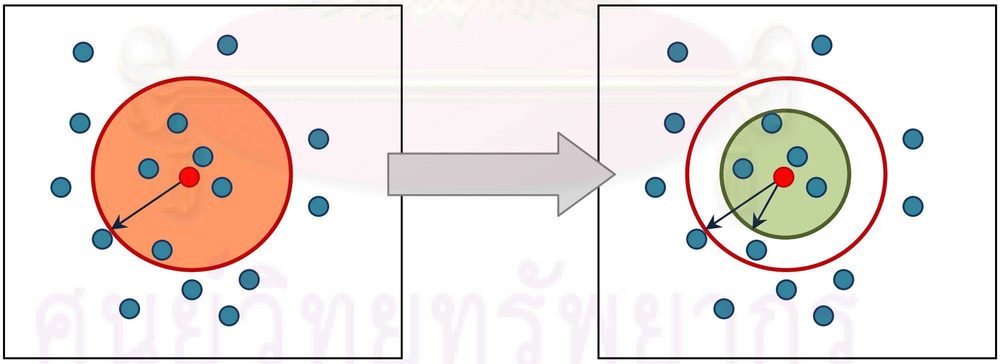


Figure 4-4 : An illustration of the adaptive kernel length

We summarize the algorithm for calculation as follow;

ALGORITHM	: Kernel Length Adaptation
DATA	: contours, density, a set of control particles
INTPUT	: length of kernel in previous time step
OUTPUT	: length of kernel in current time step
METHOD	:

```
For each time step
    Find control particle (nearest particle of critical point)
    For each control particle
        Calculate the coefficient/weight; $w_r$ , for
        kernel length adjustment using 1st constraint
        Calculate the coefficient/weight; $w_d$ , for
        kernel length adjustment using 2nd constraint
        Adjust the length of kernel
        Calculate the attributes' value using SPH
        Adjust velocity
    End for
End for
```

Table 4-3 : Kernel Length Adaptation Algorithm

4.4.1 The weight from contour condition

First, we define a weight for adjusting kernel length obtained from the constraint of Reeb graph contour as w_r . According to the concept, a large contour shows us that dense particles area; middle area, should be large. Conversely, a narrow contour shows us that dense particles area should be small. We use this concept as one of the constraints for calculating weight in order to adjust the length of kernel to be suitable for the region. Our constrained assumption is shown in figure 4-5.

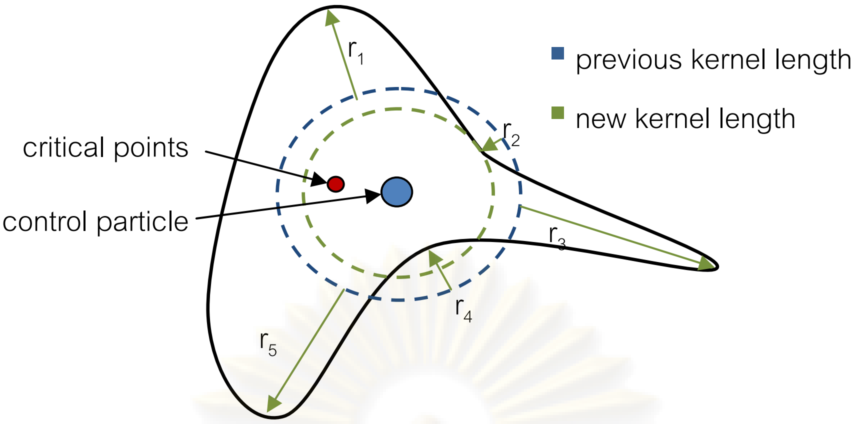


Figure 4-5 : An illustration of the constraint of Kernel length modification obtained from Reeb graph contour

Let r_i be sign distance values between circumference of kernel and critical point or perpendicular point on the contour of the object. From the assumption, we yield eq. (4.2).

$$w_r = \frac{1}{h}\left(\frac{1}{\|P\|}\sum_{i\in P}^{P} r_i + \frac{1}{\|N\|}\sum_{i\in N}^{N} r_i\right) \tag{4.2}$$

when h is kernel length in the previous step, P is a set of positive distance while N is a set of negative distance.

4.4.2 The weight from density condition

Based on the assumption that nearby particles which have similar density should be in the same region, we will find the proportion between local and global density to adjust the kernel length.

To estimate an initial density, where h_0 is chosen by reference to an initial distribution as in [19], we simply use

$$\rho_i = \sum_j m_j W(\mathrm{x}_i - \mathrm{x}_j, h_0) \tag{4.3}$$

Then, we give the weight depending on density, w_d, defined according to the relation as

$$w_d = k\left(\frac{\rho_i}{\rho_s}\right)^{-\varepsilon} \tag{4.4}$$

where ρ_s is mean of the density estimates given by

$$\rho_s = \frac{1}{\|N\|}\sum_{b=1}^{N}\rho_b \tag{4.5}$$

when N is a set of neighboring of control particle i, k is constant scaling factor and ε is the so-called sensitivity parameter defined in range $0 \le \varepsilon \le 1$.

Finally, we calculate a new kernel length by

$$h_{new} = (w_r + w_d)h \tag{4.6}$$

4.5 Process Algorithm

The following is summary of the process in each time step which is proposed.

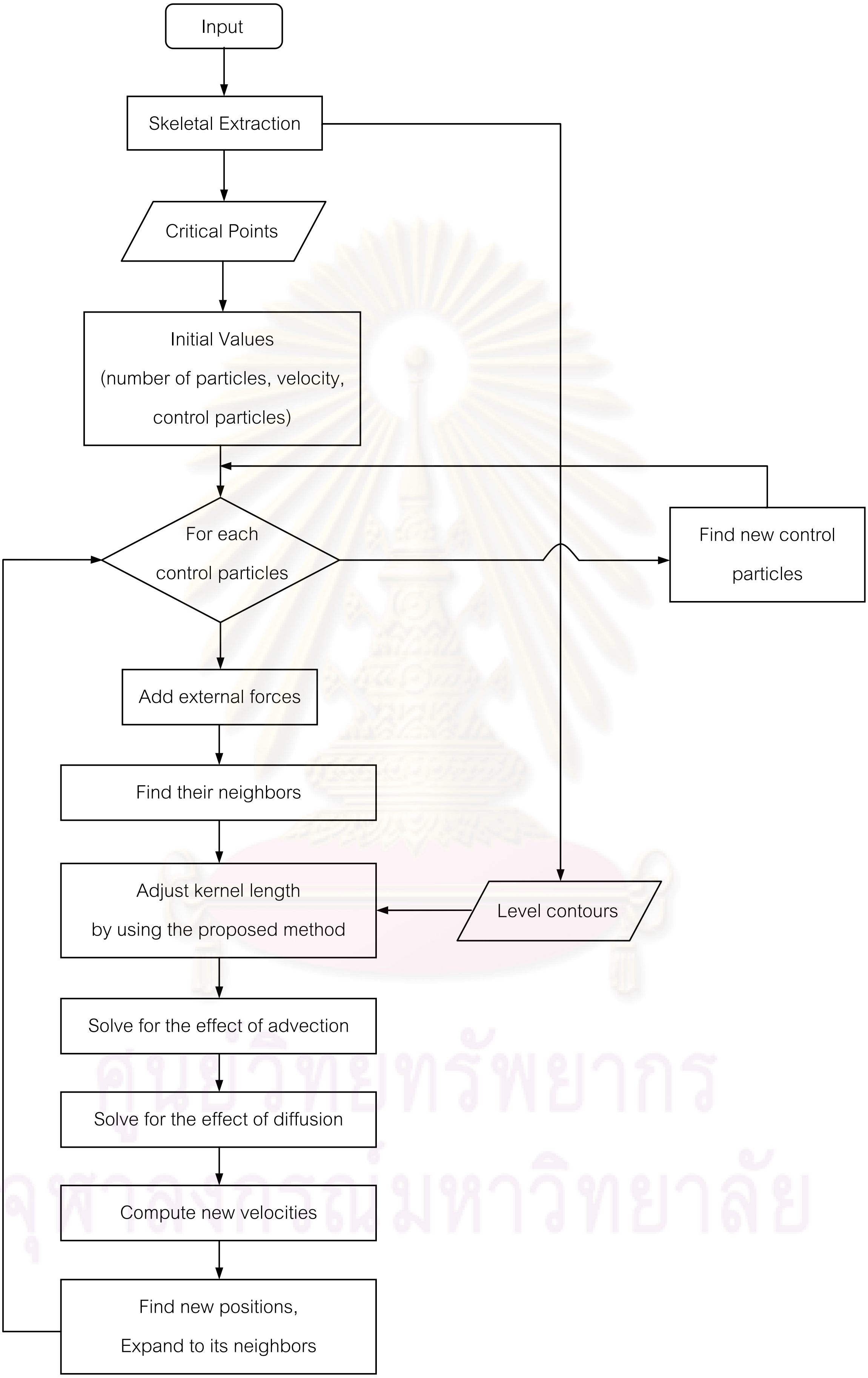


Figure 4-6 : An algorithm of entire process of each step

CHAPTER V

IMPLEMENTATION, RESULTS AND DISCUSSION

This section shows the results of the experiments from the proposed enhanced method. The constraints and testing environment are also provided. Finally, the discussion of our experimental results is given in the last section.

5.1 Constraints and Testing Environment

The objective of this research is to show that our proposed enhancing method can preserve more details in low density regions in the simulation while provide the easy control for user. The experiments should be separated into 2 tests as follow:

1. Testing for control ability
2. Testing for preserving details in low density region

For the control ability; however, because of using skeletal particle for control in this research, the method is obviously controllable. Therefore, we only test for the efficiency of preservation. For visual quality, it depends on each decision; thus, the experiment is just shown by visualization.

The enhanced method which is proposed in this research is implemented by using MATLAB 2009b and MAYA 2010. MATLAB is used to calculate the length of kernel while MAYA is used for visualization. In order to extract the Reeb Graph; in this experiment, MeshLab is a convenient tool for demonstration. The testing system is Intel® Core™ Duo Processor T2300 @1.66 GHz with 2.00 Gb, 980 MHz of RAM

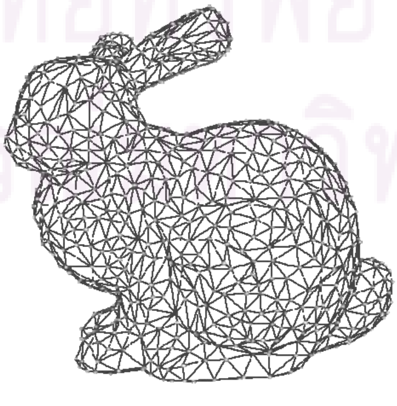


Figure 5-1: Testing model

5.2 The detail preservation test

In this test, the experiment is separated into 2 processes; generating level of contour and adapting the length of kernel. To generate the level contour, we use the algorithm as shown in Table 4-2 to extract the contour of each level and find their centroids. Figure 5-2 shows the level contour and its centroid obtained from skeletal extraction step. Each centroid or critical points are assumed to be the control node of the fluid movements.

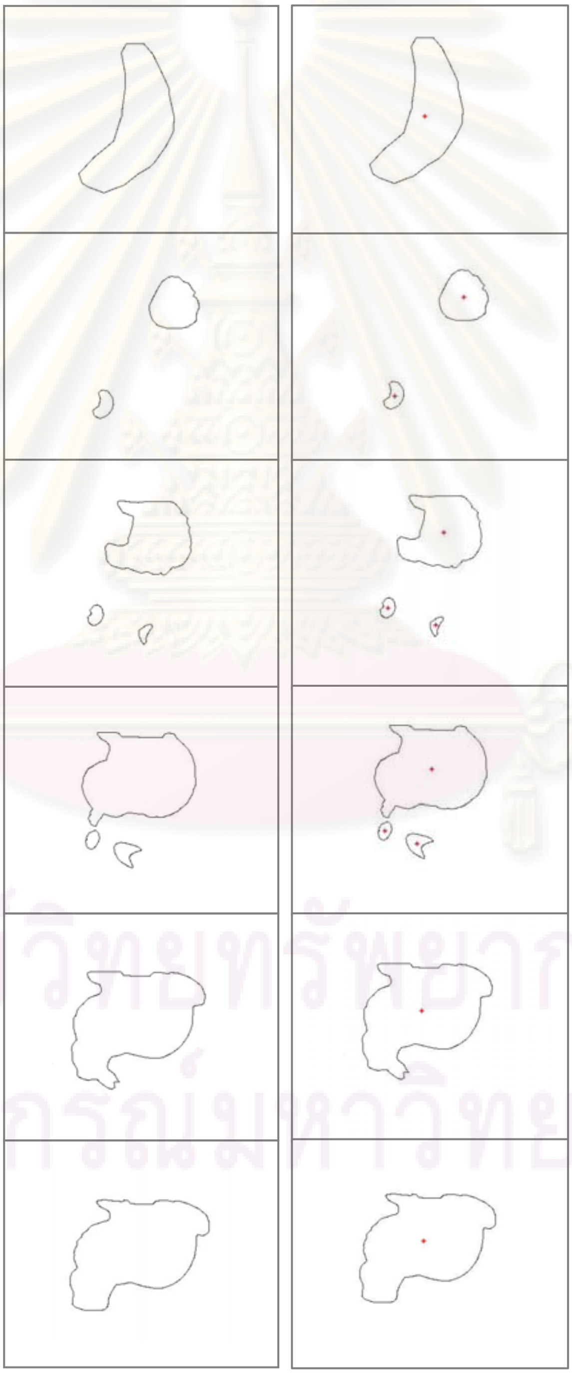


Figure 5-2 : Some of level contours and critical points in this experiment

Then, the initial particles are placed into the system. In this experiment, 2500 particles are distributed by using Gussian kernel from the critical points. A fitting curve is used to suppose the initial kernel length. That is the larger and smaller lengths are supposed to be initial length of kernel.

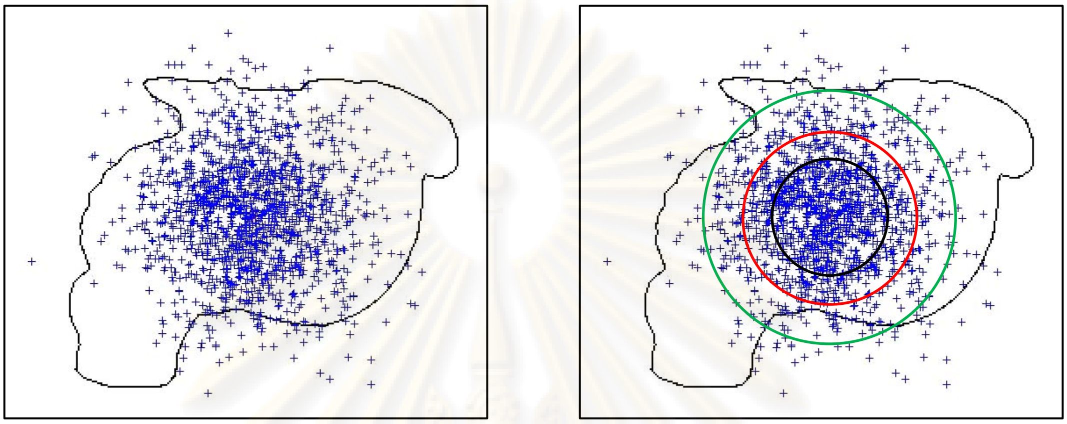


Figure 5-3 : (left) One of the results of the particle distributed from critical point (right) a red circle is a fitting curve, the black and green circles are the initial influence radius

Firstly, we demonstrate the effect of the weights which are propose to adjust the length of kernel; w_r and w_d. To get w_r, we find the difference of distance between radius and contour edge and use eq. 4.2 for calculation. Another weight; w_d, which depends on density, local density; ρ_s, is the summation of particles' density being in the radius. The following figures show the difference between the kernel lengths which is modified by w_r and w_d.

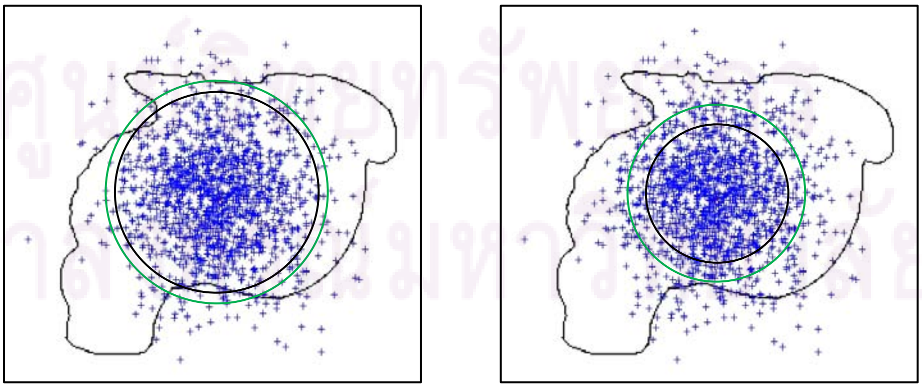


Figure 5-4 : display a kernel length of an initial radius which are adopted by w_r (left) and w_d (right)

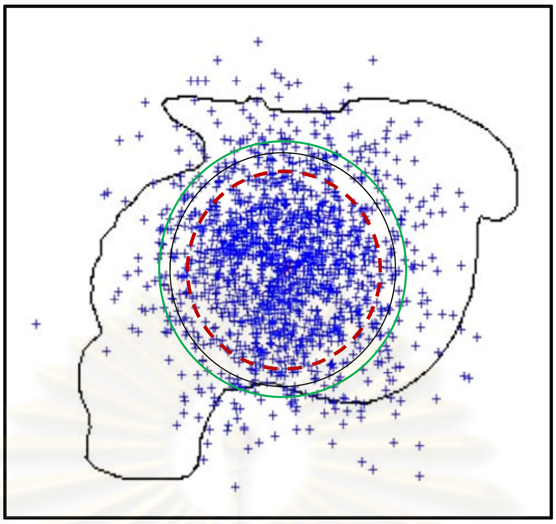
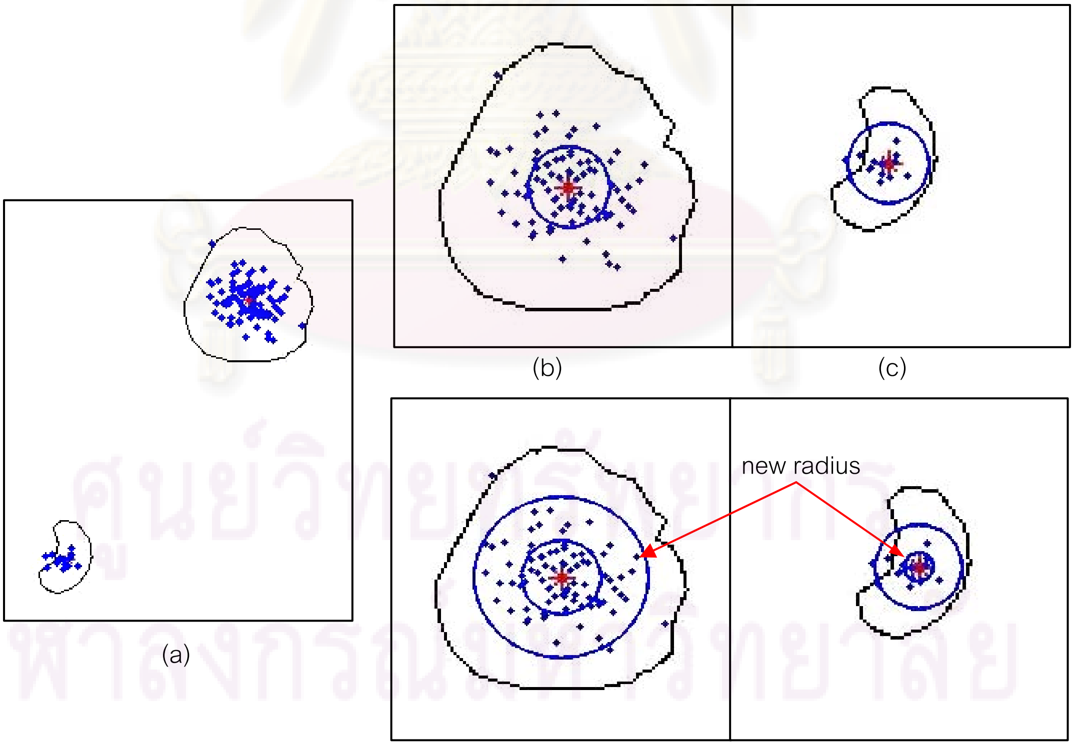


Figure 5-5 : new kernel length and its influence area

Next, to demonstrate the efficiency of our proposed method using in low density area, we suppose small amount of particles into the system. 100 particles are initially expanded from the centroid by using Gaussian kernel with h_0 =25. Then, the results of influenced particles in the kernel both initial and modified lengths are shown in Figure 5-6.



(a) (b) (c) (d) (e)

Figure 5-6 : One of the results of particles in low density area, (a) shows the particle expended in the two region, (b)and(c) show each region with initial radius, (d)and(e) show the results of new radius

From the results, fig. 5-6(a) shows the initial particles in the experiment and fig.5-6 (b),(c) represent the radius of initial kernel. Consider the radius of kernel length in (b), the spread of outside particles is similar to the internal particles. Moreover, the region of the contour is large, we imply that the length of kernel should be enlarged. On the other hand; as shown in (c), there are some particles away from the group and also some particles outside the contour. We suggest that the length of kernel should be decreased in order to preserve the detail remaining in the simulation.

After the kernel length modification, the results are shown in fig.5-6(d) and (e). They show that our proposed method is able to expand the influence radius where dense area while preserve the detail in the small and low density area.

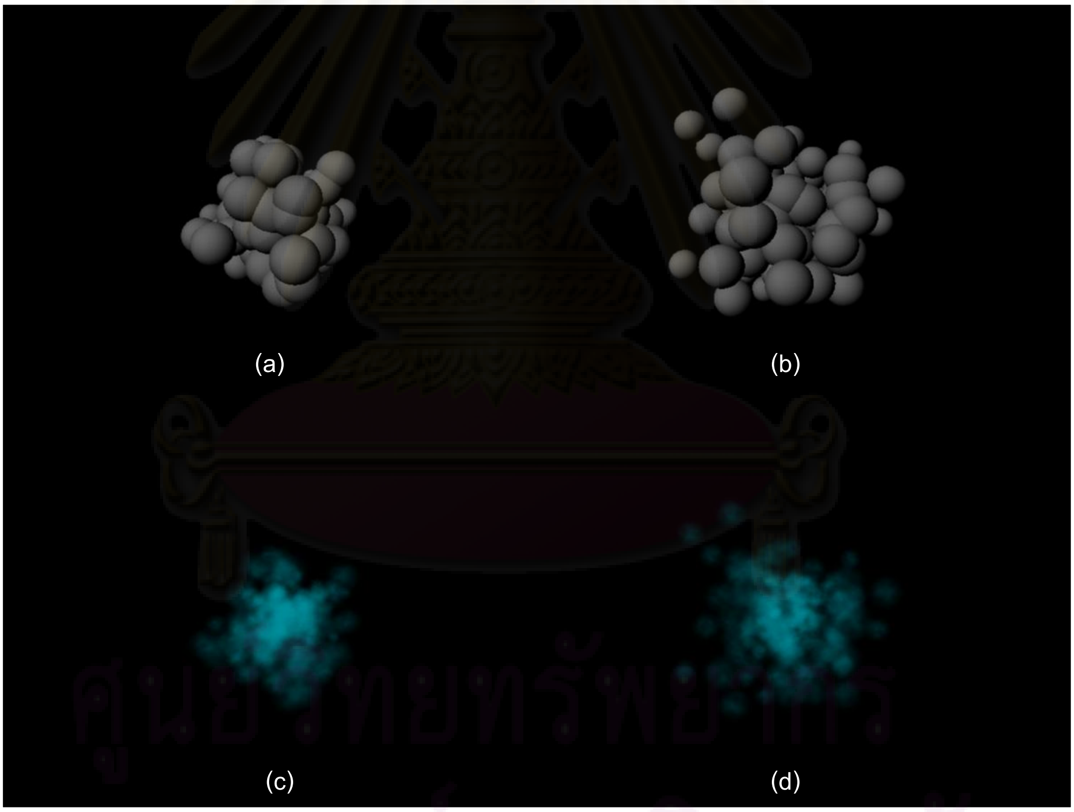


Figure 5-7 : (a),(b) are illustrated the particles in the testing simulation while (c),(d) are shown their visualization of fixed kernel length and adopted kernel length obtained from our proposed method, respectively.

Figure 5-7 shows the visualization of the movement of 50 particles using fixed kernel length and adaptive kernel length method. The simulations with a fixed kernel length with large size are shown in fig.5-7(a) and (c). The particles gather into a group and flow together. Unlike the results in fig.5-7(b) and (d) which display the results of our proposed enhancing method, they show that the outside particles splash all the time. Therefore, the method can preserve the small details on the surface of fluid such as splashing water.

5.3 Discussion

In this proposed method, fig. 5-4 display the length of kernel which is adopted by using the weighting values; w_r and w_d . If there are enough of particles for calculation, the adopted kernel length which is affected by w_d has a little change compared with the initial length. Unlike the adaptation in the low density area, the length has significant change compared with the initial length; the result is shown in the following figure.

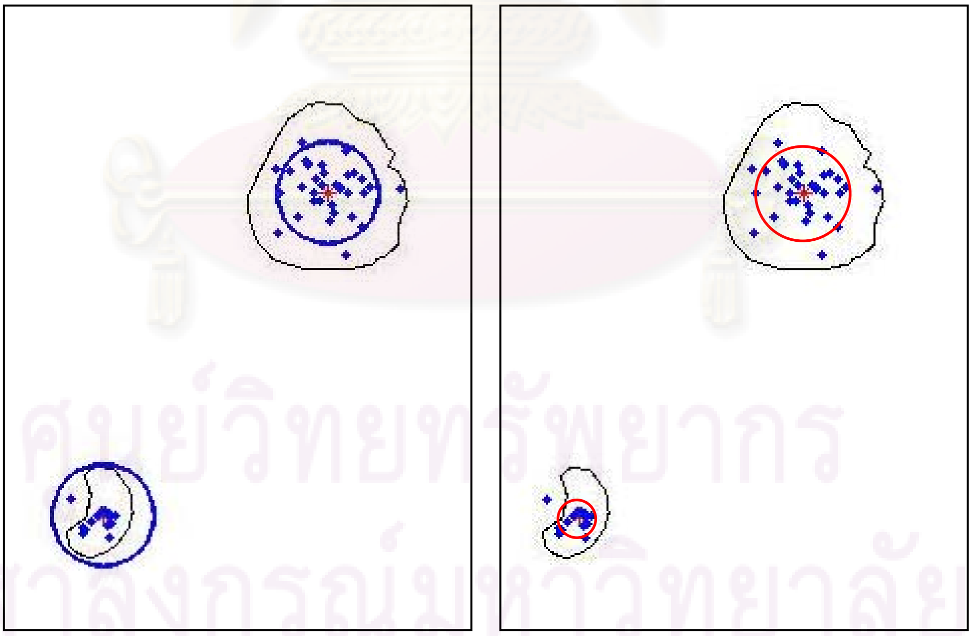


Figure 5-8: display a kernel length of an initial radius $h = 25$ (left) and the adopted length in low density area by using w_d (right)

For the weighting value which obtained from the contour's constraint; w_r , the result shows that the change of kernel length which depends on the shape of contour is significant when being used in small region while being used in large area probably leads to get more artifacts.

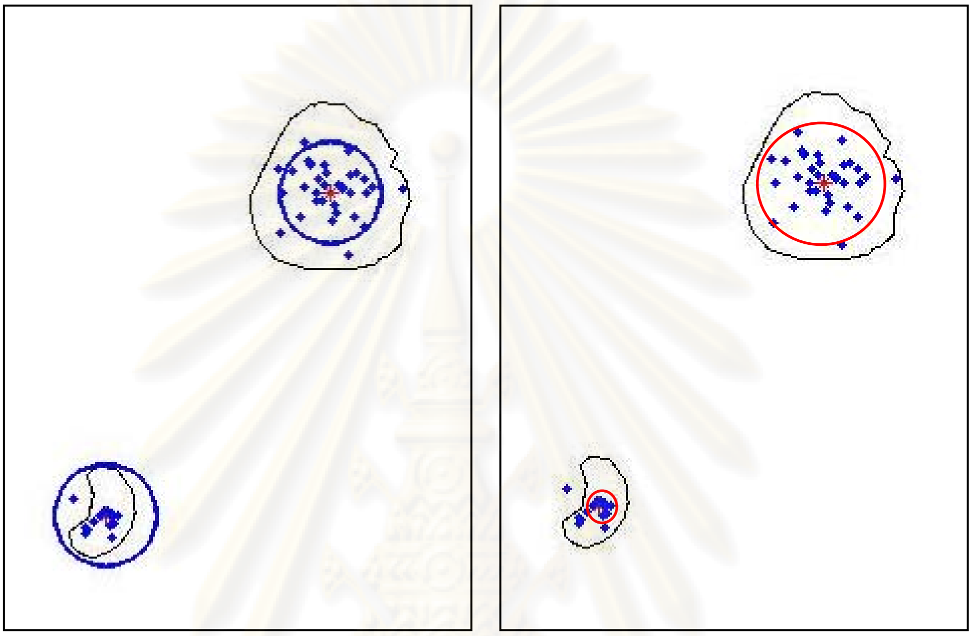


Figure 5-9: (left) display a kernel length of an initial radius $h = 25$,

(right) displays the adopted length in large area (above)

and small area (bottom) by using w_r

From the experiments; therefore, we found that the method is suitable for using in small area which has low density. The new kernel length preserves the particles which stay away from the group of particles in dense area. The errors of unstable mean values of particles being in the influence radius are decreased. However, in the large area with a number of particles, this method cannot perform efficiently because it has to spend a lot of time for calculation.

CHAPTER VI

SUMMARY AND FUTURE WORK

This section concludes over this entire thesis. We also present remaining problems and the future work.

6.1 Conclusion

The idea that is proposed in this research is based on the particle control which is called skeletal particles. By using this approach, animator can also control the fluid simulation easily without the knowledge of the underlying equation, and specify the desired movement. In addition, the control techniques for animation such as key-framing and path-line are supported. We apply Reeb graph in the skeletal extraction step. By using Reeb Graph to extract the geometry information of the user-specified object (or path-line), not only their topological structure is represented as the skeleton of the objects, but also input objects are divided into several sets having same properties such as distance from control points. At the same time, we can obtain the control points from this step.

The skeleton (or contour) is used as trajectories for control the resulting fluid animation. Then, the fluid movements are simulated based on SPH. The skeletal particles are representative of the neighboring particles within their influence radius.

The skeletal particle method is effective to model fluid simulation which can be controlled by user. However, the fixed smoothing length will be the cause of the visual artifacts in the resulting animation of the former method. In order to preserve the small details and local fluid movement, we apply the adaptive kernel estimation method into the process to calculate a new smoothing length in every time step. The weight values which are used for enhancement are obtained by considering 2 attributes; contours from Reeb Graph and density of the regions. Therefore, the smoothing length is dynamically adapted to all given points to be appropriate with density of their regions. As a result, the errors of unstable mean values of particles in low density regions are decreased.

6.2 Future works

As shown in the chapter V, we found that using adaptive kernel length method leads to more computation if has no defined constraints. The proposed method is suitable for control fluid which coupled with skeletal extraction by using Reeb graph or other similar methods. Nevertheless, controlled fluid animation is usually specified the movements by a set of key frames, it is possible to use them to define the constraints for the simulation.

Another consideration, even the kernel which adjusted the length is capable for preserve the small details in low density region, there is a problem when the contour is concave, see Fig. 6-1 for examples.

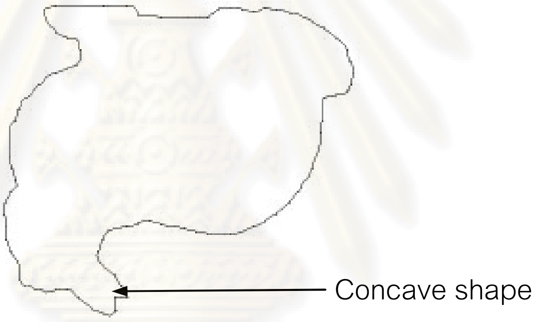


Figure 6-1: the contour which is concave

As shown in the above image, the particles are rarely in the area. In order to solve that recommended problem; in the future work, we will take another experiment by extracting Reeb graph of each level of contour; after that, we use the critical points to be minor control points.

Moreover, it would be better if we can specify the region where its kernel length has to adjust or not. The proposed condition for consideration is the distance between radius of kernel and the edge of contour. With the concept that short distance means it nearly surface; therefore, the kernel length of the particles in the area should be adapted. Long distance; on the other hand, it means far from surface; the lengths do not probably change in order to simplify the calculation.

Another additional term which will be adding to our proposed method is the term of attractive coefficient of contour in order to fit the particles to the model. In this research, we discard this constraint because the point of our proposed enhancement is preserving details in small region. However, the term of attraction can be applied to control forces or defined to be a constraint for the kernel length adaptation.

Finally, by using our method with the concept of split and merge, the particles which are in the same influence radius can be grouped together in order to let users roughly preview the results before work on the final animation.

References

[1] Lucy, L.B. A numerical approach to the testing of the fission hypothesis. Astrophysical Journal, 82 (1977): 1013-1024.

[2] Gingold, R.A. and Monaghan, J.J. Smoothed particle Hydrodynamics. Monthly Notices of the Royal Astronomical Society. 181 (1977): 375-389.

[3] Monaghan, J.J. Particle methods for Hydrodynamics. Computer Physics reports. 3(2) (1985): 71-124.

[4] Monaghan, J.J. Smoothed particle hydrodynamics. Annual review of astronomy and astrophysics. 30 (A93-25826 09-90)(1990): 543-574.

[5] Monaghan, J.J. Simulating Free Surface Flows with SPH. Journal of Computational Physics. 110(2)(February, 1994): 399-406.

[6] Sigalotti, L.Di.G., Daza, J., and Donoso, A. Modelling free surface flows with Smoothed Particle Hydrodynamics. Condensed Matter Physics. 9(2) (2006): 359-366.

[7] Desburn, M. and Gascuel, M.P. Smoothed Particles: A new paradigm for animating highly deformable bodies. In Proceedings of EG Workshop on Animation and Simulation 1996, pp. 61-76, Springer-Verlag, 1996.

[8] Premoze, S., Tasdizen, T., Bigler, J., Lefohn, A., and Whitaker, R.T. Particle-Based Simulation of Fluids. In Proceeding of Eurographics 2003, pp. 401-410, Computer Graph forum, 2003.

[9] Enright, D.P., Marschner, S.R., and Fedkiw, R.P. Animation and rendering of complex water surfaces. In Huges, J. (ed.), SIGGRAPH 2002 Conference Proceeding, Annual Conference Series, pp. 736-744, ACM Press/ACM SIGGRAPH, 2002.

[10] Müller, M., Charypar, D., and Gross, M. Particle-based fluid simulation for interactive applications. In Breen, D. and Lin, M., (eds.), Proceedings of the 2003 ACM SIGGRAPH/Eurographics symposium on Computer animation, pp. 154-159, San Diego, California : Eurographics Association, 2003.

[11] Müller, M., Solenthaler, B., Particle-based fluid-fluid interaction. In Proceedings of the 2005 ACM SIGGRAPH/Eurographics symposium on Computer animation. pp. 237-244, Los Angeles, California : ACM, 2005.

[12] Horvath, P. and Illes, D., SPH-Based Fluid Simulation for Special Effects, In The 11th Central European Seminar on Computer Graphics, pp. 23-25, April 2007 Budmerice, Slovakia. 2007.

[13] Adams, B., Pauly, M., Keiser, R., and Guibas, L.J. Adaptively sampled particle fluids. In International Conference on Computer Graphics and Interactive Techniques, ACM SIGGRAPH 2007, pp. 48-1-48-7. San Diego, California: ACM. 2007.

[14] Hong, W., House, D.H., and Keyser, J. Adaptive particles for incompressible fluid simulation. The Visual Computer: International Journal of Computer Graphics, 24(7) (2008): 535-543.

[15] Yan, H. Wang, Z., He, J., Chen, X., Wang, C., and Peng, Q. Real-time fluid simulation with adaptive SPH. Computer Animation and Virtual Worlds, 20 (2009): 417-426.

[16] Lastiwka, M., Quinlan, N., and Basa M. Adaptive particle distribution for Smoothed Particle Hydrodynamics. International Journal for Numerical Methods in Fluids 47(2005): 1403-1409.

[17] Attwood, R.E., Goodwin, S.P., and Whiteworth, A.P. Adaptive smoothing lengths in SPH. Astronomy & Astrohysics no. Acoustic 464(2) (March 2007): 447-450.

[18] Leonardo Di G. S., Hender L., Donoso A. Sira E., and Klapp J. A shock-capturing SPH Scheme based on adaptive kernel estimation. Journal of computational Physics 212 (2005): 124-149.

[19] Leonardo Di G. S., Hender L. Adaptive kernel estimation and SPH tensile instability. Computers and Mathematics with Applications 55 (2008): 23-50.

[20] Brent, M.D. and John, K. Keyframing particles of physically based systems. In Theory and Practice of Computer Graphics 2005 : Eurographics UK Chapter Proceedings, pp. 11-18. University of Kent, Canterbury, United Kingdom: Eurographics. 2005.

[21] Arisara S. , Pizzanu K. Keyframe control of fluid simulation using skeletal particles. In Proceedings of The International Conference of Computational Methods 2007, Hiroshima, Japan: International Conference Center, 2007.

[22] Witkin, A.P. and Heckbert, P.S. Using particles to sample and control implicit surfaces. In Proceedings of the 21st annual conference on Computer graphics and interactive techniques, pp. 269 – 277. 1994.

[23] Shi, L. and Yu, Y. Controllable smoke animation with guiding objects. ACM Transaction on Graphics 24 (2005): 140-164.

[24] Fattal, R. and Lischinski, D. Target-driven smoke animation. In International Conference on Computer Graphics and Interactive Techniques ACM SIGGRAPH 2004, pp.441-448, 2004.

[25] Seung-Ho, S. and Chan, H.K. Controlling fluid animation with geometric potential. Computer Animation and Virtual Worlds 15(3-4) (2004): 147-157.

[26] Shin, S.H. and Kim, C.H. Target-driven liquid animation with interfacial discontinuities. Computer Animation and Virtual Worlds 18 (2007): 447-453.

[27] Shin, S.H., Lee, J., Kim, S.J., and Kim, C.H. Controlling liquids using pressure jump. In International Conference on Computer Graphics and Interactive Techniques ACM SIGGRAPH 2006 Sketches, Article No.: 62, 2006.

[28] Harris, M.J. Fast fluid dynamics simulation on the GPU. GPU Gems: Programming Techniques, Tips, and Tricks for Real-Time Graphics, pp. 637-665.

[29] Tan J. and Yang, X.B. Physically-based fluid animation : A survey. Science in China Series F: Information Science. 52(5) (2009): 723-740.

[30] Shi, L. and Yu, Y. Taming liquid for rapidly changing targets. In Eurographics/ACM SIGGRAPH Symposium on Computer Animation 2005, pp.229-237, Jul. 2005.

[31] Foster, N. and Metaxas, D. Controlling fluid animation. In Proceedings of CGI'97, pp.178-188, 1997.

[32] McLaughlin, K.C., Edelsbrunner, H., Harer, J., Natarajan, V., and Pascucci, V. Loops in Reeb graphs of 2-manifolds. Discrete & Computational Geometry 32(2) (July 2004): 231 – 244.

[33] Baloch, S., Krim, H., Kogan, I, and Zenkov, D. Rotation invariant topology coding of 2D and 3D objects using morse theory. IEEE Transactions on Image Processing 19(2) (February 2010): 306-321.

[34] Chen, J.X. and da Vitoria Lobo, N. Toward interactive-rate simulation of fluids with moving obstacles using Navier-Stokes's equations. Graphical Models and Image Processing 57 (March 1995): 107-116.

[35] Witting, P. Computatinal fluid dynamics in a traditional animation envieronment. In Proceedings of SIGGRAPH 99, Computer Graphics Proceedings, Annual Conference Series, pp. 129-136, Aug. 1999.

[36] Welch, J.E., Harlow, F.H., Shannon, J.P., and Daly, B.J. THE MAC METHOD: a computing technique for solving viscous, incompressible, transient fluid-flow problems involving free surfaces. Report LA-3425, Los Alamos Scientific Laboratory, 1965.

[37] Mark Thomas Carlson. Rigid, melting, and flowing fluid. Doctoral dissertation, College of Computing Geogia Institue of Technology, 2004.

[38] Stam, J. Stable fluids. In Proceedings of SIGGRAPH 99, Computer Graphics Proceedings, Annual Conference Series, pp. 121-128. Aug. 1999.

[39] Fedkiw, R., Stam J., and Jensen, H.W. Visual simulation of smoke. In Proceedings of the 28th annual conference on Computer graphics and interactive techniques, pp. 15-22. ACM. 2001.

[40] Losasso, F., Gibou, F., and Fedkiw, R. Simulating water and smoke with an octree data structure. ACM Transactions on Graphics 23 (August 2004): 457-462.

[41] Shi, L. and Yu, Y. Visual smoke simulation with adaptive octree refinement. Tech. Rep. UIUCDCS-R-2002-2311, University of Illinois at Urbana-Champaign, Dec. 2002.

[42] Reeves, W.T. Particle system - a technique for modeling a class of fuzzy objects. ACM Transaction on Graphics 2(2) (April 1983): 91-108.

[43] Harlow, F.H. The particle-in-cell computing method for fluid dynamics. Methods in Computational Physics 3 (1964): 319–343.

[44] Gentry, R.A., Martin, R.E., and Daly, B.J. An Eulerian differencing method for unsteady compressible flow problems. Journal of Computational Physics 1(1) (August 1966): 87-118.

[45] Koshizuka, S., Nobe, A., and Oka, Y. Numerical Analysis of Breaking Waves Using the Moving Particle Semi-implicit Method. International Journal of Numerical Methods of Fluid 26 (1998): 751-769.

[46] Y. Zhu and R. Bridson. Animating sand as a fluid. In SIGGRAPH '05: ACM SIGGRAPH 2005 Papers, pp. 965–972. New York, NY, USA : ACM Press. 2005.

[47] Brownlee, R., Houston, P., Levesley, J., and Rosswog, S. Enhancing SPH using Moving Least-Squares and radial basis function. Algorithm for Approximation, pp. 103-112. Springer Berlin Heidelberg. 2007.

[48] Martel, H. and Shapiro, P.R. Cosmological Simulation with Adaptive Smoothed Particle Hydrodynamics. In Makino and Hut P. (eds.) Astrophysical Supercomputing Using Particles IAU Symposium 2001.

[49] D'humières, D. and Lallemand, P. Lattice gas automata for fluid mechanics. Physica A: Statistical Mechanics and its Applications 140(1-2) (December 1986): 326-335.

[50] Wolf-Gladrow, D. Lattice-Gas Cellular Automata and Lattice Boltzmann Models. Springer Verlag. 2000.

[51] Succi, S. The Lattice Boltzmann Equation for Fluid Dynamics and Beyond. Oxford University Press. 2001.

[52] Kontomaris, K., Hanratty, T.J., and McLaughlin J.B. An algorithm for tracking fluid particle in spectral simulation of turbulent channel flow. Journal of Computation Physics 103(2) (December 1992): 231-242.

[53] Besnard, D.C., F.H. Harlow, R. Rauenzahn and C. Zemach. Spectral Transport Model for Turbulence. Los Alamos National Laboratory Report LA-UR-92-1666, 1992.

[54] Zienkiewicz, O.C.; Taylor, R.L.; Nithiarasu, P. Finite Element Method for Fluid Dynamics, 6th Edition, Elsevier, 2005.

[55] Hieber, S.E. and Koumoutsakos, P. A Lagrangian particle level set method. Journal of Computational Physics 210(1) (November 2005): 342-367.

[56] Ramussen, N., Enright, D., Nguyen, D., Marino, S., Summer, N., Geiger, W., Hoon, S., and Fedkiw, R. Directable photorealistic liquids. In Eurographics/ACM SIGGRAPH Symposium on Computer Animation, pp. 193-202. Aug. 2004.

[57] Losasso, F., Talton, J.O., Kwatra, N., and Fedkiw, R. Two-way coupled SPH and particle level set fluid simulation. IEEE Transactions on Visualization and Computer Graphics, 14(4) (2008): 797-804.

[58] Treuille, A., Mcnamara, A., Popović, Z., and Stam, J. Keyframe control of smoke simulations. ACM Transactions on Graphics 22 (July 2003): 716-723.

[59] McNamara, A., Treuille, A., Popović, Z., and Stam, J. Fluid control using the adjoint method. ACM Transactions on Graphics 23 (August 2004): 449-456.

[60] Thürey N., Keiser R., Pauly M., and Rüde U. Detail-preserving fluid control. In Cani M.P. and O'Brien J. (eds), Proceedings of the 2006 ACM SIGGRAPH/Eurographics symposium on Computer animation. pp. 7-12. 2006.

[61] Thürey N., Keiser R., Pauly M., and Rüde U. Detail-preserving fluid control. Graphical Models (2009):

[62] Pigin F. Cohen J. M., and Shah M. Modeling and Editing Flows Using Advected Radial Basis Functions. In Eurographics/ACM SIGGRAPH Symposium on Computer Animation, pp. 223-232. 2004.

Bibliography

[1] Doraiswamy, H. and Nataajan, V. Efficient algorithms for computing Reeb graphs. Computational Geometry 42 (2009): 606-616.

[2] Enright, D., Losasso, F., and Fedkiw, R. A fast and accurate semi-Lagrangian particle level set method. Computers and Structures 83 (2003): 479-490.

[3] Enright, D., Fedkiw, R., Ferziger, J., and Mitchell, I. A hybrid particle level set method for improved interface capturing. Journal of Computational Physics 183(1) (November 2002): 83-116.

[4] Foster, N. and Fedkiw, R. Practical Animation of Liquids. In Proceedings of the 28th annual conference on Computer graphics and interactive techniques, pp. 23-30. New York: ACM. 2001.

[5] Li, S. and Liu, W.K. Meshfree and particle methods and their applications. Applied Mechanics Reviews 55(1) (January 2002)

[6] Liu, M.B., Liu, G.R., and Lam, K.Y. Adaptive smoothed particle hydrodynamics for high strain hydrodynamics with material strength. Shock Wave 15(1) (2006): 21-29.

[7] Losasso, F., Fedkiw, R., and Osher, S. Spatially adaptive techniques for level set methods and incompressible flow. Computer & fluids 35(10) (2006): 995-1010.

[8] LÓpez H. and Donoso A. Adaptive Kernel Methods to Simulate Quantum phase space flow. Condensed Matter Physics 9 (2005): 351-358.

[9] Oger G., Doring M., Alessandrini B., and Ferrant P. An improved SPH method : Towards higher order convergence. Journal of Computational Physics 225 (2007): 1472-1492.

[10] Owen, J.M., Villumsen, J.V., Shapiro, P.R., and Martel, H. Adaptive smoothed particle hydrodynamics: Methodology. II. The Astrophysical journal supplement series 116 (June 1998): 155-209.

[11] Pelupessy, F.I., Schaap, W.E., and Weygaert, R. ven de. Density estimators in particle hydrodynamics: DTFE versus SPH. Astronomy and Astrophysics. 403(2003): 389-398.

[12] Poulin, P., Beaudoin, P., and Clavet, S. Particle-based viscoelastic fluid simulation. In Symposium on Computer Animation:Proceeding of the 2005 ACM SIGGRAPH/Eurohraphics symposium on Computer animation, pp. 219-228. New York : ACM Press. 2005.

[13] Takahashi, T. et al. Realistic animation of fluid with splash and foam. Computer Graphics Forum 22(3) (November 2003): 391-400.

BIOGRAPHY

Miss Saithip Limtrakul was born on November 9, 1983 in Khonkaen. She studied at Demonstration School, Khonkaen University. In 2006, she graduated from the department of Mathematics Science, Faculty of Science, Khonkaen University.