REAL TIME FLOOD HAZARD SIMULATION

NURUL AIN BINTI MAT ARHAM @ ELHAM

This Report Is Submitted In Partial Fulfillment of Requirements For The Bachelor Degree Of Electronic Engineering (Computer Engineering)

Fakulti Kejuruteraan Elektronik Dan Kejuruteraan Computer
Universiti Teknikal Malaysia Melaka

June 2013

UNIVERSTI TEKNIKAL MALAYSIA MELAKA
FAKULTI KEJURUTERAAN ELEKTRONIK DAN KEJURUTERAAN KOMPUTER

BORANG PENGESAHAN STATUS LAPORAN
PROJEK SARJANA MUDA II

Tajuk Projek : REAL TIME FLOOD HAZARD SIMULATION

Sesi Pengajian : 1 | 2 | / | 1 | 3

Saya NURUL AIN BINTI MAT ARHAM B021010362

mengaku membenarkan Laporan Projek Sarjana Muda ini disimpan di Perpustakaan dengan syarat-syarat kegunaan seperti berikut:

1. Laporan adalah hakmilik Universiti Teknikal Malaysia Melaka.
2. Perpustakaan dibenarkan membuat salinan untuk tujuan pengajian sahaja.
3. Perpustakaan dibenarkan membuat salinan laporan ini sebagai bahan pertukaran antara institusi pengajian tinggi.
4. Sila tandakan (√) :

[ ] **SULIT*** — *(Mengandungi maklumat yang berdarjah keselamatan atau kepentingan Malaysia seperti yang termaktub di dalam AKTA RAHSIA RASMI 1972)

[ ] **TERHAD**** — **(Mengandungi maklumat terhad yang telah ditentukan oleh organisasi/badan di mana penyelidikan dijalankan)

[/] **TIDAK TERHAD**

Disahkan oleh:

(TANDATANGAN PENULIS)

(COP DAN TANDATANGAN PENYELIA)

HAMZAH ASYRANI BIN SULAIMAN
Pensyarah
Fakulti Kejuruteraan Elektronik dan Kejuruteraan Komputer
Universiti Teknikal Malaysia Melaka (UTeM)
Hang Tuah Jaya,
76100 Durian Tunggal, Melaka.

Tarikh: 06/06/2013

Tarikh: 06/06/2013

iii

"I hereby declare that this report is the result of my own work except for quotes as cited in the references."

Signature :

Author : NURUL AIN BINTI MAT ARHAM @ ELHAM

Date : 11 JUNE 2013

iv

"I hereby declare that I have read this project report and in my own opinion this project report is sufficient in terms of the scope and quality for the award of Bachelor of Electronic Engineering (Computer Engineering) With Honours."

Signature : ....

Name : EN. HAMZAH ASYRANI BIN SULAIMAN

Date : 06/06/2013

Special Dedicate:

To my beloved parents Mr. Mat Arham and Mrs. Jamaliah for their genuine love, prayers and encouragement. Then to my supervisor who guide and give moral support me and to all my friends for your help and support throughout my journey education.

ACKNOWLEDGEMENT

Alhamdulillah, Praised to Allah SWT, with His willing and blessing by giving me the opportunity for completing this Final Year Project which is title Real Time Flood Hazrd Simulation. This final year project report was prepared for Faculty of Electronic Engineering and Computer Engineering, Universiti Teknikal Malaysia Melaka (UTeM), basically for student in final year to complete the undergraduate program that leads to the degree of Bachelor of Electronic Engineering (Computer Engineering).

Firstly, I would like to express my deepest thanks to Mr. Hamzah Asyrani bin Sulaiman, as my supervisor for their guidance. Besides, constant supervise as well as providing some necessary information regarding to the project and also for their support in completing this project. I also would like to extend my sincerely thanks to the lecturers and staffs of FKEKK UTeM for their cooperation during I complete the final year project that had given valuable information, suggestions and guidance in the compilation and preparation this final year project report.

I would like to express my gratitude and appreciation toward my parents, family, and others for their cooperation, encouragement, constructive suggestion and full of support for the report completion, from the beginning till the end. Also thanks to all of my friends and everyone, that have been contributed by supporting my work and help myself during the final year project progress till it is fully completed.

ABSTRACT

This project is to constructing and developing complex fluid simulation. Fluid simulation is practical method for the representation of liquids in interactive application like serious game simulation and computer games. Fluid simulation has been various unique of behavior such as stirring liquids, blending differently colored fluid, mixing insoluble, rain falling and flowing terrain and fluid interacting. Hence, all this behaviors are useful for many applications. In recent years fluid simulation has becoming one of the common trends in computer visualization. This thesis will explain how to do fluid simulation such as water and liquid and then allow it rendering with the real-time simulation. Besides, this project will describe a study using OpenGL with Visual C++ programming software to implement the 3D computer graphic and complex fluid simulation techniques. The main goal was to study and examine the stability and effectiveness of this approach as a solution tool for environment studies.

ABSTRAK

Projek ini adalah untuk membina dan membangunkan simulasi cecair kompleks. Simulasi Bendalir adalah kaedah praktikal untuk perwakilan cecair dalam aplikasi interaktif seperti simulasi permainan serius dan permainan komputer. Simulasi Bendalir telah unik pelbagai tingkah laku seperti kacau cecair, mengadun berbeza cecair berwarna, pergaulan tidak larut, hujan jatuh dan mengalir kawasan dan berinteraksi cecair. Oleh itu, semua tingkah laku ini adalah berguna bagi banyak aplikasi. Dalam tahun-tahun kebelakangan ini simulasi bendalir telah menjadi salah satu trend biasa dalam visual komputer. Tesis ini akan menerangkan bagaimana untuk melakukan simulasi cecair seperti air dan cecair dan kemudian membenarkan ia memberikan dengan simulasi masa nyata. Selain itu, projek ini akan menggambarkan satu kajian menggunakan OpenGL dengan Visual C++ pengaturcaraan perisian untuk melaksanakan simulasi komputer 3D grafik dan kompleks teknik cecair. Matlamat utama adalah untuk mengkaji dan memeriksa kestabilan dan keberkesanan pendekatan ini sebagai alat penyelesaian untuk kajian alam sekitar.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF ABBREVIATION

SPH	-	Smoothed Particles Hydrodynamic
GUI	-	Graphic User Interface
FLIP	-	Fluid Implicit Particle
3D	-	3 Dimension

CHAPTER I

INTRODUCTION

1.1 Introduction

Fluid simulation has becoming one of the common trends in computer visualization. Instead of this trend, it became a tool for generating realistic animation such as water, smoke, fire and other fluid object. In computer graphic visualization, fluid simulation is in range of complexity simulation development where it extremely need long time consuming to produce a high quality of visual effect used in modern games and also movie animation. Because of it, research on fluid simulation methods has become an important research direction in the field. The requirements for liquid simulations in computer graphics are slightly different from those in other fields. What is important is not physical accuracy but physical plausibility and the appearance.

The expertise in computer graphics basically able to modified the physical-based formula to define fluid behavior and created several various algorithms to develop any behavior of fluid movement. The common algorithm used are Eulerian grid-based methods, Smoothed Particle Hydrodynamics (SPH) methods, Vorticity-based methods and Lattice Boltzmann methods. These algorithms were used to create fluid objects behavior and mostly were applied in movie industries and modern game for the visual effects.

Computer graphics in real time is allow users to convert the real world data into a computer program and then allow the user for a better observation and visualization. Such as the ocean wave in 3D simulation viewing. This approach is tend as best useful of environmental studies and so that can be extended. For all types of virtual realities like serious games and training or computer games is always become demanding to cover an aspect of world needed. Since many different aspects of real time particle-based fluid simulation where covered in a couple of papers from authors around the world. This thesis gives an overview on the topic, as it discusses my implementation of a particle-based fluid simulation and a suitable water rendering.

1.2 Problems Statement

It was known as an environmental study on flood. The increasing flash flood hazards in major cities have caused a tremendous damage to the society and this requires more essential countermeasures to be implemented. Various kinds of software and computer system have been developed by foreign experts to provide a better decision making and superior analysis for flood mitigation. Most of the software is based on hydrological and hydraulic modelling. However the attempt on using computer graphical method is still considered as new. Solving fluid simulation is an issue in handling it in virtual reality application.

1.3 Objectives

The main goal was to study and examine the stability and effectiveness of this approach as a solution tool for environment studies.
The objectives for this research study are as follows:

1. To develop fluid simulation based on the reality behavior of fluid.

2. To implement fluid simulation based on random terrains that generated by the computer.
3. To implemented SPH fluid simulator which can import a model represented by boundaries.

1.4 Scope of Project

The scope of work for the project will focus on the following areas:

1. Concentrating on development fluid simulation
   - Focusing on development of fluid simulation in random terrain. Determine the position texture, velocity texture and etc.

2. Using Microsoft Visual Studio C++ 2010
   - Using the Microsoft visual Studio C++ to perform the complex fluid simulation in random terrain. The water or fluid simulation will be done using particle-based method for showing the fluid movement. The idea was to be determined the process in real-time basis.

1.5 Report Organization

The contents of this report are divided into several chapters which followed the type of contents. There are five chapters are inserted in this report which are introduction, literature review, methodology, discussion and results and conclusion and recommendations.

In chapter one, the overall information about this project are explained briefly. This chapter included the introduction of project, objectives, problem statement and scope of project. All of them are explained detail. The methodology is also explained in this chapter briefly.

Chapter two in this report is explained about the literature review about this project. All the theories and basic principles are inserted in this chapter. For this project, literature review of the basic review which contains information gathered to gain knowledge and ideas in completing the project.

Methodology of this project is explained in chapter three. This part will explain more on the project development and the project path, how it is done and method on how it has been made. This part also includes software development in systematic ways.

Experimental results and analysis is presented in Chapter 4. Finally, this thesis ends with Chapter 5 that concludes the project followed by a number of recommendations for future research.

CHAPTER II

LITERATURE REVIEW

2.1 Introduction

This chapter describes the theoretical information leading to the development of the project. The chapter begins with the description of the fluid simulation and the algorithm. There are lots of project being implement and develop instead of make human life easier and more systemic. Since there are many different aspects need to explore to implement this project such as fluid simulation algorithm, technique being used, fluid behaviors, fluid mechanism and others. Thereafter, this chapter will describes the implementation of simulation.

2.2 Fluid Simulation Algorithm

In general, fluid simulation algorithms can be divided into a several categories:

1. The Navier-Stokes Equations

   The Navier-Stokes equations are a set of non-linear equations that describe the motion of a fluid at any point within the flow. The full equations can be manipulated into different forms based on the characteristics of the fluid. For a viscous, incompressible fluid the equations are the momentum equation

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

(2.1)

And the mass conservation equation,

$$\nabla \cdot \mathbf{u} = 0$$

(2.2)

Here the equations are given in vector form, and they are equally valid for both two and three dimensions. The fluid is described by two continuous fields, a velocity field u and a pressure field p. The equations describe how these two fields change over time.

The momentum equation (Equation 2.1) determines how the fluid accelerates due to the forces acting on it. Each term of the equation can be thought of as acceleration due to a force. The easiest term to understand is the last one, f. This term accounts for acceleration due to all external forces acting on the fluid. Generally, gravity is the only external force specified; however, external forces can also include things like wind blowing on a particular spot of the fluid surface or user specified forces. The second term,$\frac{1}{\rho}\nabla p$ gives us the acceleration due to the gradient of the pressure. The term ρ is fluid density, mass per unit volume. If the pressure at a particular point is equal on all sides, the net force will be zero. If, on the other hand, the pressure is higher on one side than on the other, a force will push from the higher pressure towards the lower pressure. The third term,$\nu\nabla^2\mathbf{u}$ accounts for acceleration due to viscosity. Intuitively, this can be thought of as each point in the fluid trying to move at the same velocity as nearby surrounding points. If the fluid on one side of a point is flowing faster, and the fluid on the other side of the point is flowing slower, the point will try to adjust its velocity towards the average of the velocities. The viscosity constant ν determines how strongly the point favours the surrounding velocities over its own velocity.

The first term of the momentum equation, - (**u** · ∇) **u**, is the hardest to understand. It accounts for the advection of the fluid. Advection is how any property of the fluid gets moved along according to the flow of the fluid. As an example, think of a point in space that has a temperature, *T*, associated with it. The temperature at that point could change in one of two ways. It could change simply because the point itself is heating up or cooling down, or it could change because the surrounding fluid flow causes hot air to be replaced by cold air. The latter case describes advection. In this particular term, it is the fluid momentum which is being advected.

The mass conservation equation (Equation 2.2) is responsible for enforcing in-compressibility of the fluid. It states that the divergence of the fluid must be zero everywhere in the fluid. One way to think about this is to visualize a small chunk of fluid volume. Because the density of the fluid is constant, the total mass within the volume must also remain constant. When the fluid is sloshing around, some fluid will be entering and leaving the chunk. Thus, in order for the mass to remain constant, the rate of fluid entering the chunk must be exactly equal to the rate of the fluid leaving. Also, since the fluid is incompressible everywhere, this equation holds for all individual chunks of fluid and thus everywhere in the fluid.

2. Eulerian

Eulerian algorithms are widely used in fluid dynamics. Here, the computational mesh is fixed and the continuum moves with respect to the grid. In the Eulerian description, large distortions in the continuum motion can be handled with relative ease, but generally at the expense of precise interface definition and the resolution of flow details.

The Eulerian specification of the flow field is a way of looking at fluid motion that focuses on specific locations in the space through which the fluid flows as time passes.This can be visualized by sitting on the bank of a river and watching the water pass the fixed location.as the simply the explanationis by divides a cuboid of space into cells. Each cell contains a velocity vector and other information, such as density and temperature.

3. Lagrangian

Lagrangian algorithms, in which each individual node of the computational mesh follows the associated material particle during motion, are mainly used in structural mechanics. The Lagrangian description allows an easy tracking of free surfaces and interfaces between different materials. It also facilitates the treatment of materials with history-dependent constitutive relations. Its weakness is its inability to follow large distortions of the computational domain without recourse to frequent re-meshing operations. Particle system is use a cloud of primitive particles to define the volume of an object. The particles appearance and motion are controlled by procedural equations. Each particle is initialized with attributes such as position, velocity, colour and lifespan.

The Lagrangian specification of the flow field is a way of looking at fluid motion where the observer follows an individual fluid parcel as it moves through space and time. Plotting the position of an individual parcel through time gives the path line of the parcel. This can be visualized as sitting in a boat and drifting down a river. Lagrangian is actually particle-based physics, not as effective as Eulerian Grid for modelling “swirly”. However, particles based are much better for expansive regions, since they aren’t restricted to a grid.

4. Hybrid

For large worlds that have specific regions where swirlies are desirable, use Lagrangian everywhere, but also place Eulerian grids in the regions of interest. When particles enter those regions, they become influenced by the grid’s velocity vectors.

One of the earliest numerical fluid simulation methods combining the Eulerian and Lagrangian viewpoints were the Particle-In-Cell (PIC) method [4]. This method was designed to simulate compressible fluids. The simulation area is represented by a grid of equally sized cells, and the fluid is represented by particles. Fluid velocities and temperatures are stored on the grid, while fluid mass is stored in the particles. At the beginning of a time step, the density in each cell is calculated from the number and mass of particles in the cell. A state equation for the fluid is used to determine the pressure in a cell based on its density and temperature. Next, the acceleration due to the pressure gradient is computed, and this value is used to update the grid velocity to tentative values. The individual particles are then moved according to a velocity that is linearly interpolated from the nearest stored grid values. Then, the final velocity and temperature values for each cell are calculated. If no new particles have entered the cell, the tentative values are accepted as the final values. Otherwise, a weighted average of the values from the cell and the values from surrounding cells from which the new particles have come is computed. The PIC method is successful in modelling highly distorted fluid flow, but some numerical viscosity is caused by transferring the momentum back and forth from the grid to the particles. The Fluid Implicit Particle (FLIP) method extends PIC to use an adaptive grid and attempts to reduce some of its numerical dissipation [13]. More information is stored in the particles, namely mass, velocity and internal energy. The algorithm14 proceeds similarly to PIC. At each time step, the particle quantities are transferred to a grid. The momentum equation is solved on the grid resulting in a new velocity field.

2.3 How to Simulate Fluids

Based on paper written by Stefan Auer, Real time Particle Based Fluid Simulation, in the 19th century Claude Navier and George Stoke created the fundamentals of modern fluid dynamics as they formulated the well-known Navier-stokes equations. These equations will describe about the conversion of momentum with two additional equations for mass and energy conversion. As the formula tends