Isobel McNeany  
Department of Computer Science and Creative Technologies   
University of the West of England   
Coldharbour Lane Bristol, UK   
email.address@live.uwe.ac.uk

Abstract (*What will the project achieve, how so, and why.)* not included in wordcount

My project is about fluid dynamics and how it can be combined with rigidbodies to simulate the effects of wind on a rigidbodies. The wind effect would be modelled by using fluid dynamics

Keywords - not included in wordcount

1 Introduction (~400 words)  
*This is a brief literature review. Using a number of key sources, map out the professional context for your project.*

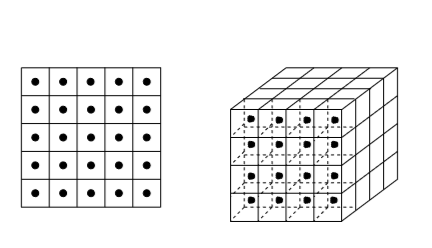
Due to the modern demand for graphics in films and games to look as realistic as possible certain techniques have had to have been developed to keep up with the demand.

Figure 1- Grid with center point that holds the data

These techniques include fluid dynamics as a way of modelling liquids and gases through a scene. The fluid dynamics need to be produced in real time so that it can be updated to and react to the objects placed within the scene. A lot of work into fluid dynamics has been done by Stam (1999) who created a semi-lagrangian method to produce a grid (figure 1). Within the grid each cell holds data.

The grid – based system allows the fluid to flow through the cells by checking the data in the other cells around. Stam’s Grid-based system was produced on the central processing unit (CPU), which limited the extent of the project due to the computational power needed. Further work has then been done by (Liu et al, 2004) who produced the system on the GPU, allowing it to be calculated quicker. The fluid dynamics system has also been applied in various context’s such as for image manipulation (Bertalmio et al, 2001), applications in weather (Mason, 2006), as well as modelling forces that are on and around objects (Gou et al).

The further application of my project will be to add objects into the flow, this will then either, push the object or flow around the object (depending on mass etc..). A lot of research has been done into the movement of flow around objects that are solid and fixed in the scene (Liu et all 2007), (Gou et al).

The project will be a simulation on modelling the effects of wind on an object, using fluid dynamics. The simulation will be in 2D, which could be expanded to 3D, however the 2D scope of the project will be enough to show off the simulation, and it is a large enough scope, while reducing the computational time compared that that of the project if it was in 3D.

2 Research methods ~200 words  
***List (!)*** *your key research questions and the research methods you use(d) to answer them*

Specifically, to be researched are the different options of how to approach a fluid dynamics system. Stam (1999) produced a paper which details his methodology, by using a grid-based system. After reading more papers, books as well as searching online, it was discovered that there are two main elements to the fluid dynamics system, a density solver and a velocity solver. A density solver will calculate the movement of the density in and out of each cell. The velocity solver will then be called after the density solver to recalculate the velocity of the grid due to the change in the density. The final part of the project is to add rigidbodies into the system and have the fluid and object react to each other.

3 Research findings ~800 words  
*discuss what exactly you have discovered. Insert tables, images and short snippets of code if necessary.*

To create a simple but effective fluid dynamics system, as used by Stam (1999) a grid will need to be created, first a 2D grid which can then later be converted into a 3D grid. Each of the grid sections will hold centralised data which Stam(1999) and Stam(..) favors as the method used to create the dynamic flow by controlling all the different variables within the grid individually. This allows the flow through each section of grid to be controlled. This also allows extra variables to be added into the sections of the grid. The placement of objects into the scene can be shown and allow the flow to adjust in accordance e.g. a block placed in the section of the grid will mark it as containing an object and wouldn’t allow any flow into the section. The sections around would then adjust to deal with the extra flow through their blocks.

Calculations around how the flow will move around into the other grid cells is solved using a diffusion solver. There are two different types; implicit and explicit. (Sharcnet,2009) Goes into more detail about the specific differences between implicit and explicit, the website does suggest that explicit is the better of the two to use, however (Stam, 1999) disagrees and believes that implicit is the better solver to use as the explicit solver will reach a certain point where it can no longer handle the values. The project will use the implicit one as it is better at handling any of the values without getting to big or out of control.

With the method of solving density decided upon, Caretto (2010) states that density – based solvers are traditionally used for compressible flows, as the simulation being produced allows the flow to be compressed a density solver is appropriate to use. Within the density solver there are several steps that need to be followed as seen below (figure2).

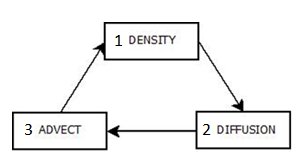


Figure 2 Density solver iterations

The Second step of a fluid dynamics system can give a few options on which direction to go in to solve it. One option is to use a velocity solver, which will follow the same format as the density solver, with using the velocity, calculating the new velocity before applying it into the project. The other option is to use a pressure solver, which again follows the same suit as the previously mentioned two. The final option which seems to be a popular one as used by (Carlson et al, 2004). The combination of a velocity and pressure solver is based on bernoulli's principle (Bernoulli, 1738). Bernoulli’s principle has a disadvantage as it makes several assumptions, the first is that the fluid is incompressible and that all energy is conserved, also that the viscous effect is nonexistent. This however makes a velocity and pressure solver easier to solve as the system doesn’t need to account for a loss in energy or the effect of viscosity.

For this project however, a velocity solver is a good option for the moment as with wind pressure only varies as the altitude rises, therefore with the scope of the project a velocity solver should be enough to demonstrate what is needed.

The final step of the project is the behavior of the rigidbodies. From the unity documentation (Unity, 2018) the rigidbody controller allows forces to be applied onto objects, which can change the rigidbody’s position, the AddForceAtPosition() function allows forces to be added on the rigidbody at specific positions. Work has been done before with the flow of fluid around solid objects (Liu et all 2007), (Gou et al). This can give me an idea of how to start adding objects into the fluid. The idea is that rigidbodies will move with the fluid flow, some research has been done into the movement (Camarena, 2011), however Camarena did have trouble predicting the movement of the rigidbodies, which may mean a flaw in the system, this system does produce the movement of a rigidbody in a flow.

4 Conclusion and Recommendations (Planned research implementation) ~100 words  
***One paragraph only*** *that tells us how your project will develop from here, based on your research.*

For my project I will first work towards producing a 2D fluid dynamics model, which will use the method of using a density solver and a velocity solver. Then a rigidbody will be added into the fluid dynamics model. The unity rigidbodies system will then be used to an advantage where the forces will be applied by using the AddForceAtPosition function.

References - not included in wordcount

Author, A. (2009) *A Book About Student Projects*. Location Publisher.

Author, B (2008) ‘Journal Article’, *Digital Media Journal*, Vol 1/13, pp 13-23

University of the West of England (2009) *UWE Library Services:Study skills - The Harvard System* [Online] Available from [18 September 2009]

[https://www.researchgate.net/figure/Density-solver-steps\_fig1\_289666107 - figure 2](https://www.researchgate.net/figure/Density-solver-steps_fig1_289666107%20-%20figure%202)

<https://www.sharcnet.ca/Software/Fluent6/html/ug/node988.htm> - FLUENT 6.3 User's Guide - 25.1.2 Density-Based Solver

Bibliography - not included in wordcount

Appendixes: - not included in wordcount

A Log sheets (mandatory)  
covering your research and development phase (September to December)

B Further documentation (optional)  
If applicable, insert further evidence of research completed, such as interview questions, competitor research, longer code sections or other relevant materials in here. This appendix does not come under the wordcount. Nevertheless only insert useful materials here, please don’t add bulk.