Jordan Varghese 11/06/2015

Final Project Proposal PH235

**Section I**

*Project Title:* Computational Analysis of Pipe Flow for Turbulent & Reactive Fluids

*Members:* Jordan Varghese

Role: Research, Programming, and Animation.

**Section II**

Computational fluid is a highly researched field and much groundwork in terms of programming such models and analysis has already been done. I wish to bring this analysis into the scope of python programming using the methods we have investigated. I want to eventually create an animation for the fluid flow simulation, thus requiring a heavy use of the pylab and matplotlib modules. Furthermore, much of computational fluid dynamics is the time-step solution of various differential equations such as Navier-Stokes, the Boltzmann Equation, and the Reynolds-averaged Navier–Stokes equations. These equations can be solved via the various numerical approximation methods that have been studied thus far. Of particular use, Runge-Kutta methods or Euler methods could be used. Furthermore, I am interested in the implementation of Direct simulation Monte Carlo (DSMC) method, which uses probabilistic simulation to solve fluid equations. I would like to attempt to code up this method in the analysis of a fluids system.

While the onset of this project would be first applied to simple, laminar flow, once the framework of the code has been done, I would like to try applying and optimizing the models with turbulent fluid mechanics, and perhaps even more interestingly reactive flows (combustion, synthesis, reaction design/optimization). Much simulation research has been on done on these topics, and methods have been implemented to relatively successful accuracies, particularly the field of Large Eddy Simulation (LES) and Direct Numerical Simulation (DNS). Optimization of the final code could be studied for increasing accuracy, decreasing computational cost, and increasing sensitivity. Comparisons of the python simulation could be made to other CFD simulation software to see how the python code compares.

**Section III**

The final deliverable would be various simulations of pipe flow fluid dynamics. The several components would increase in complexity, the first essentially being a simple laminar flow, the next being turbulent flow, and the final being reactive flow (with and without turbulence). I would hope to have animations that show the effects of turbulence by showing the mixing of fluid layers that occur as turbulent flow occurs. Reactive flow would be further complex and will require further study with Professors Davis and Sidebotham.

**Section IV**

The metrics upon which this project can be analyzed and evaluated are its accuracy, computational speed/pragmatism, applicability, and complexity. Furthermore, its performance in comparison to other CFD simulation software could be used as a metric for the success of the project.

**Section V**

I believe that the solution of laminar single component fluid flow will be somewhat trivial and perhaps the easiest portion of this project. The challenges I forsee coming up is the simulation of turbulent flow can be a highly volatile and unpredictable phenomena. Furthermore, the overall speed of the program will be mitigated as the systems become more and more complex. Reactive flows may face a similar issue. If turbulent or reactive flows become too complex to simulate, I could perhaps study laminar flow more in depth, perhaps a multi-component fluid flow or fluid flow with some sort of accumulation.

**Section VI**

By *November 23rd* I would like to have most of the laminar flow simulation done, with a two component fluid (this will be useful for reactive flows later)

By *December 4th* I would like to have animations for turbulent flow done, to exhibit the chaotic and unpredictable nature of this type of flow

By *December 11th* the reactive flow simulation, as well optimized versions of the turbulent flow and laminar flow simulation should be complete.

**References**

https://web.stanford.edu/group/ctr/ctrsp00/harvey.pdf

http://www.cd-adapco.com/webinar/cfd-chemical-process-industry-reactive-flows

http://pythonprogramming.net/monte-carlo-dice-function/?completed=/monte-carlo- simulator-python/

http://catdir.loc.gov/catdir/samples/cam032/99059887.pdf

http://www.geo.unibe.ch/people/alt-epping/research.php?PID=92485227

http://www.wiley.com/WileyCDA/WileyTitle/productCd-0471261793.html

http://www.aem.umn.edu/~kmahesh/publpdf/journal/jeff\_jcp\_algo.pdf

http://ebooks.cambridge.org/chapter.jsf?bid=CBO9780511610103&cid=CBO978051161 0103A008&tabName=Chapter&imageExtract=

http://www.sciencedirect.com/science/article/pii/S0376042108000559

http://link.springer.com/article/10.1007%2FBF01082555#page-1

http://users.monash.edu.au/~bburn/pdf/csiro99\_pipe.pdf

http://www.researchgate.net/profile/B\_Armaly/publication/232005183\_Experimental\_an d\_theoretical\_investigation\_of\_backward- facing\_step\_flow/links/0c960528cd88caf896000000.pdf