Lattice Boltzmann Method Fluid Simulation

Schuyler Krawczuk
Phys 140, Fall 2017, San Jose State University

The Lattice Boltzmann Method is a technique used for solving problems in computational fluid dynamics. I use this method to create simulations of objects in a steady fluid flow.

INTRODUCTION

Computational fluid dynamics is a tool used to solve a variety of problems in engineering and science such as aerodynamics, weather science, structure airflow, and even more recently biotechnology. The method I used to solve my fluid dynamics problem is the Lattice Boltzmann Method. Contrary to the more common finite difference method that solves the Navier-Stokes equations that describe the motion of a fluid, this method takes a statistical approach. The motion of microscopic particles is described by a distribution function which in turn determines movement at a macroscopic scale. The advantage of using this method is each section can be solved locally, making it a good candidate for parallelization. For my project I use the lattice Boltzmann method to create simulations of two different static objects in a moving fluid, showing its behavior as it passes the object.

METHOD

The area of the fluid is broken down into a lattice; 200x100 in the case of my solution. At each lattice site, there are nine different options for movement by the fluid filling the site: the four cardinal directions, the four diagonal directions, and not moving. The probability of the fluid's movement is broken down into number density, or what portion of the fluid is making which movement. The equilibrium number density is initially determined by the Maxwell-Boltzmann distribution, which can be explained simply by saying the movements of farther distance have lower probability. Since the total number

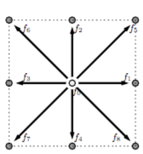

FIG. 1: The nine number densities for each site

density at each site sums to one, $\sum_k f_k = 1$. Not moving is the most likely case as it requires the least distance traveled, so $f_0 = \frac{4}{9}$. The cardinal directions are the next closest, so $f_{1-4} = \frac{1}{9}$. The diagonals are the furthest away from the lattice point, so their probabilities are lowest: $f_{5-8} = \frac{1}{36}$. The full undiscretized Lattice-Boltzmann equation is the following:

$$\frac{\partial f_k}{\partial t} + e_k \cdot \nabla f_k = -\frac{1}{\tau}(f_k - f_k^{EQ}) \tag{1}$$

f_k represents the nine directional number densities, f_k^{EQ} is the equilibrium values for the number densities, e_k are the nine velocity vectors corresponding to the number density at each lattice site, and τ is the relaxation parameter. When discretized, we create the main steps of the Lattice Boltzmann method: The collision step and the streaming step. In the collision step in equation (2), the number densities at each site are calculated based on their current relation to the equilibrium values. Equation (3) is the calculation of the equilibrium number densities. u is the total velocity of each site and w_k is the initial probability distribution of the number density.

$$f_k(x_k, t + \Delta t) = f_k(x_k, t) - \frac{\Delta t}{\tau}(f_k - f_k^{EQ}) \tag{2}$$

$$f_k^{EQ} = \rho w_k [1 + 3(e_k \cdot u) - \frac{3}{2}(u \cdot u) + \frac{9}{2}(e_k \cdot u)^2] \tag{3}$$

From the number density, we can calculate the macroscopic quantities of the fluid, such as density in equation (4) and momentum in equation (5).

$$\rho = \sum_{k=0}^{8} f_k \tag{4}$$

$$\rho u = \sum_{k=0}^{8} e_k f_k \tag{5}$$

In the streaming step, fluid is then moved into neighboring lattice sites respective of the number densities. Equation (6) shows how this is done. The boundary conditions are also addressed in this step. Instead of moving the fluid into boundaries, its velocity component orthogonal to the boundary is reversed.

$$f_k(x_k + e_k \Delta t, t + \Delta t) = f_k(x_k, t + \Delta t) \tag{6}$$

An important concept in fluid dynamics to address before reaching the results is the Reynolds number. Many computational fluid dynamics simulations are not quantified by time, velocity, and distance but rather by relative turbulence. The Reynolds number is a dimensionless quantity used to characterize this; higher Re meaning turbulent flow and lower Re meaning more laminar flow.

$$Re = \frac{UL}{\nu} \tag{7}$$

Reynolds number is the relationship of characteristic velocity U to characteristic length L and viscosity ν. In my simulations I use two relatively high Reynolds numbers since it allows them to be stable at the size they are. A laminar flow will have very little disturbance in it, while a turbulent flow around an object will cause perturbations known as vortex shedding. Examples of this are shown in my results.

RESULTS

After creating a lattice Boltzmann method program in python, I ran simulations of two different objects in a steady fluid flow, similar to a wind tunnel. I first did simulations of a simple straight line barrier at Reynolds numbers of 30,000 and 60,000, then did the same for a sphere. To produce the full simulations of about 9,000 steps, it took upwards of thirty minutes. On my current computer, each step, including the animation process, takes on average 89 ms which comes out to thirteen minutes for the length of my simulations. However, my previous laptop that I did all of the work for this project on was much slower. Images from these simulations can be seen in figures 4-7.

To test the validity of these simulations, I checked for the conservation of mass throughout the simulation. Initially, I found that there was a decrease in the total mass as the simulation went on, but this was due to the boundary conditions where the fluid flows from. The boundaries are periodic on the ends, and on the right edge the number densities are reset to reflect the initial velocity. Without the velocity being driven at this border, the mass is conserved. This comparison of mass conservation can be seen in figure (2) One observable feature of these simulations is the turbulence that appears in the fluid. This is called vortex shedding, and it is caused when a fluid passes over a non-aerodynamic object. The pressure difference between the fluid directly behind the object and the passing fluid causes low-pressure vortexes to come off alternating edges of the object. This vortex shedding can be quantified by the lateral force on the object, which I show in figure (3). We can see that with the higher Reynolds number, the more turbulent flow produced higher force and vortex shedding earlier in the simulation.

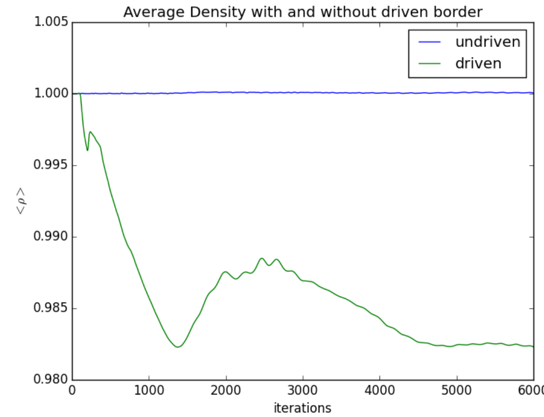

FIG. 2: Mass was conserved by the simulation where the periodic boundaries did not reset the number densities

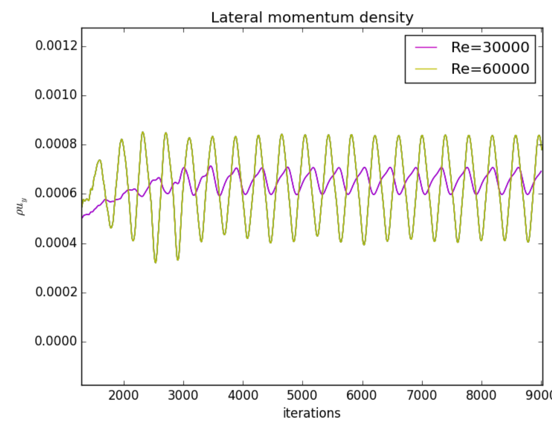

FIG. 3: The force perpendicular to the flow against the sphere for the two simulations

SUMMARY

To summarize, the lattice Boltzmann method is a technique in computational fluid dynamics that uses a statistical approach in determining microscopic behavior. I used this method to create simulations of a barrier and sphere in a fluid flow at different levels of turbulence.

[1] I. Mele. I. Tiselj. University of Ljubljana, Department of Mathematics and Physics, (2013)

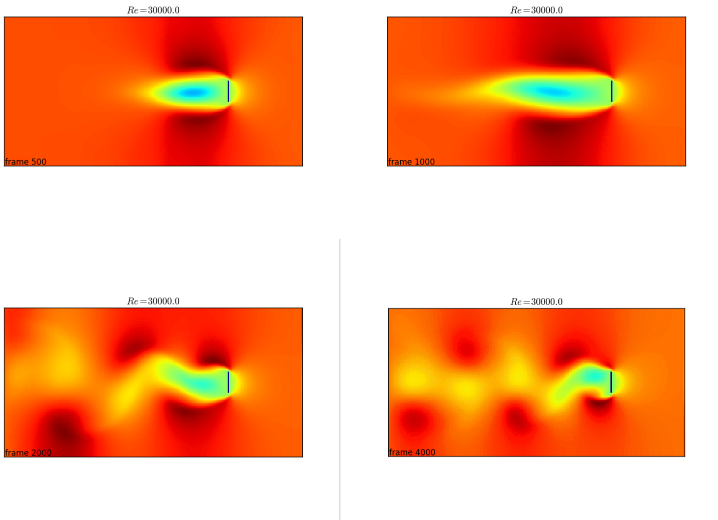

FIG. 4: Re=30,000 Simulation at frames 500, 1,000, 2,000, and 4,000. The flow is to the left and the colors represent speed to the left, hot colors being faster and cool being slower.

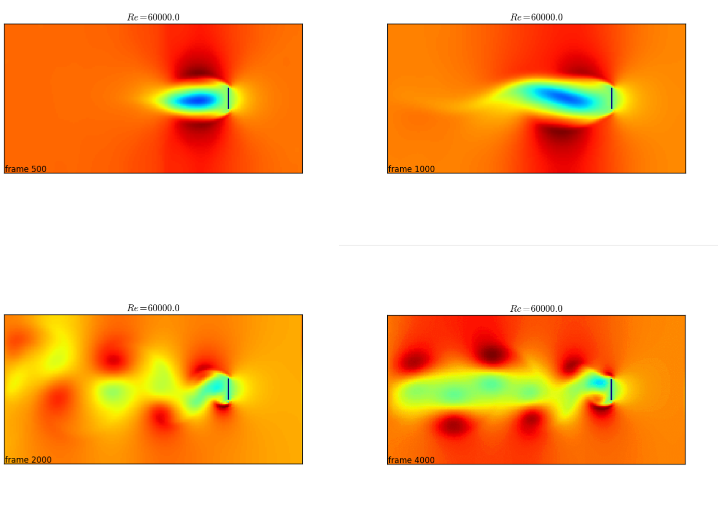

FIG. 5: Re=60,000 Simulation at frames 500, 1,000, 2,000, and 4,000. The flow is to the left and the colors represent speed to the left, hot colors being faster and cool being slower.

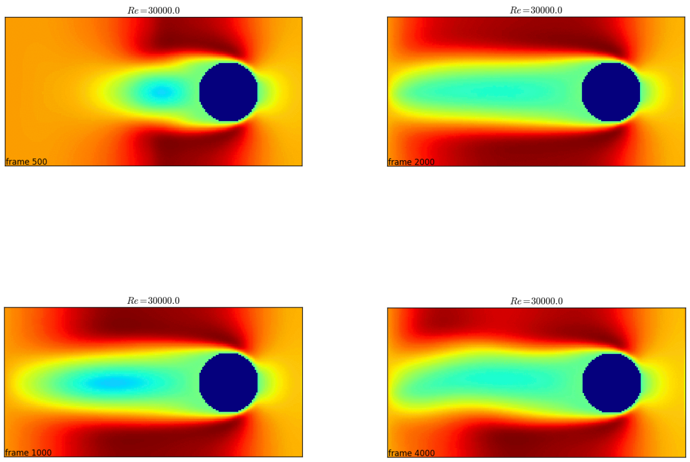

FIG. 6: Re=30,000 Simulation of a sphere in the fluid flow. Color also represents leftward speed in this simulation

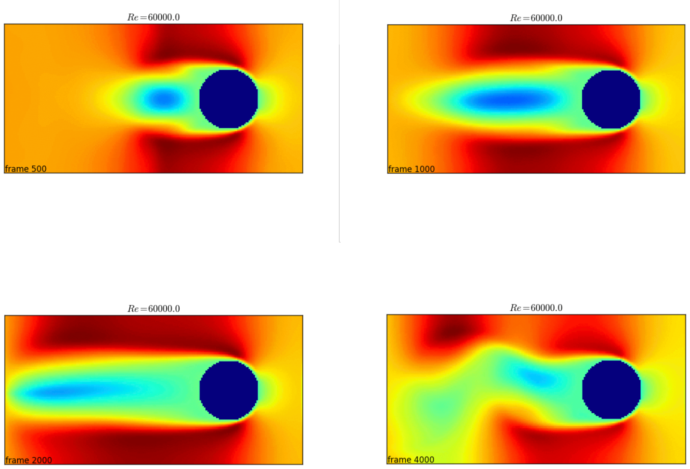

FIG. 7: Re=60,000 Simulation of a sphere in the fluid flow.