Noah Mifsud
Modeling Natural Phenomena
TP7
CFD

Introduction

In this week's lab we were tasked with understanding, both theoretically and experimentally, how the Navier-Stokes (NS) equation for fluid dynamics can be approximated by the Lattice Boltzmann equation (LBE), and how the LBE can be implemented using the Lattice-Boltzmann method. We were provided a functioning code to implement a two-dimensional Lattice Boltzmann method.

The Lattice Boltzmann method approximates fluid flow using a set of cells, each of which represents a 'packet' of particles. These packets possess a multivariable state describing the combined average velocity of their particles at a given timestep. The system evolves by projecting the position of the packets forward, calculating the outcome of any collisions which occur between packets, then projecting the new velocities forward again. Constants are used to determine how quickly the fluid moves and how quickly it relaxes back to an equilibrium state following a perturbation. The method can support different collision parameters and can thus model both soft packet-packet collisions and hard packet-surface collisions.

Our provided code took as input every parameter required to describe a small 2D fluid system including x and y dimensions for the number of cells, constants to describe the fluid properties, number of iterations over which we wanted to evolve the system, and a delineation of hard obstacles off which the fluid would bounce. It output images of the system at intervals of 100 timesteps on a color-gradient representing the fluid flow. Our code came pre-calibrated to model a system in which fluid was injected from the left-hand side of the system and flowed from left to right before 'disappearing' out of the opposite side. The top and bottom edges had periodic boundary conditions. In the center was placed a hard, round circle around which our fluid flowed.

This essay will progress through an experimental and theoretical treatment of aspects of our Lattice-Boltzmann system before applying this knowledge to understand how the system could be used to model real-world scenarios.

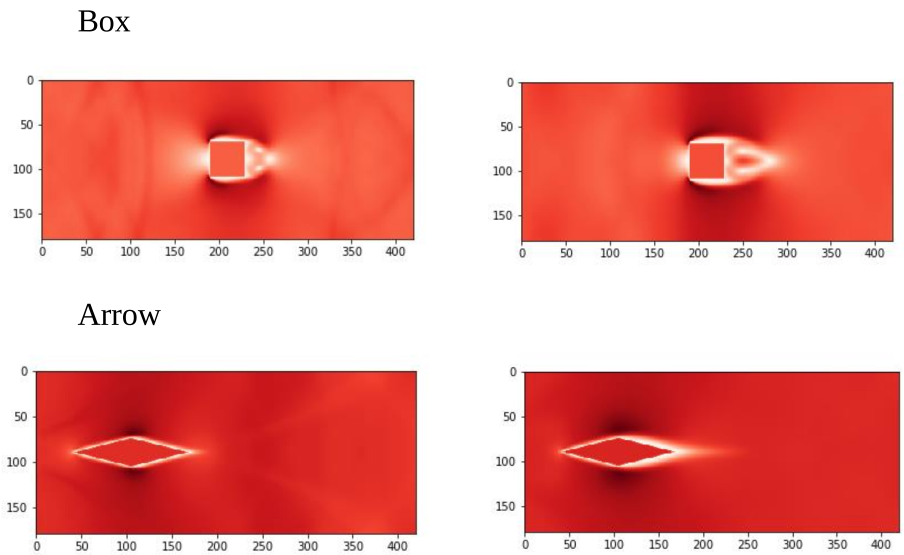
Understanding the system experimentally

Although it is possible to determine how various parameters should affect the system using theory, it is often quicker and more interesting to conduct simple experiments. Using this philosophy, the following section will cover a series of tests performed with goal of understanding how the fluid flow around objects is affected under different circumstances.

The first parameter we will analyze is the shape of the object placed to obstruct the fluid flow. This will give insight into the various types of flow we can observe and demonstrate some unexcepted behavior modeled by the system.

Figure 1 shows images of two objects and the fluid flows they created. The images were chosen out of the generated timestep-images for how well the represented the behavior and/or evolution of the flow. The labeled names correspond to the object shape and can also be found in the attached python code which outlines how each shape was encoded within the program.

Figure 1



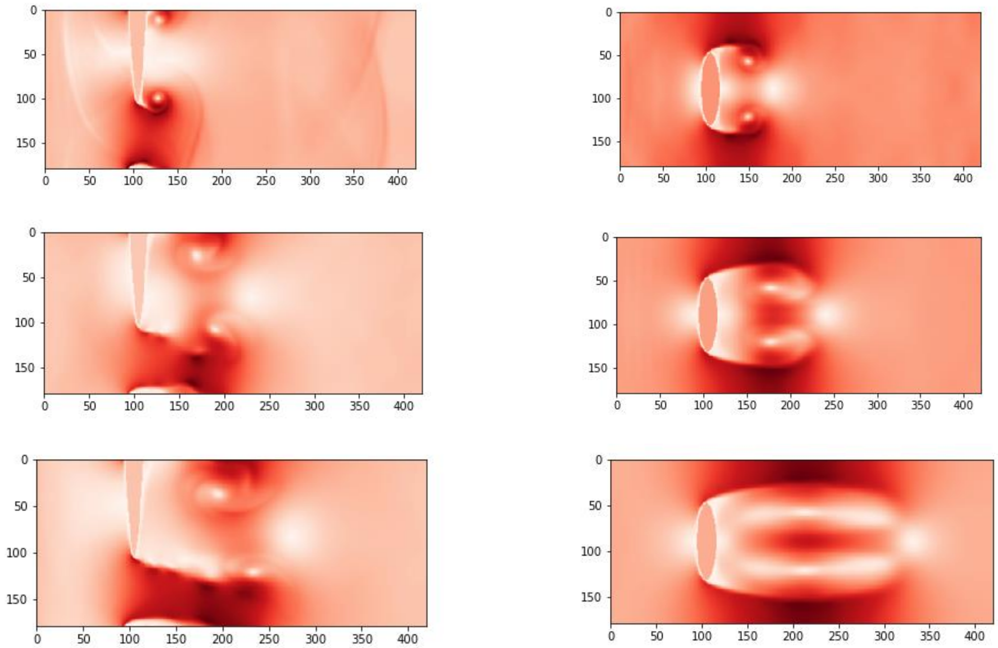
In figure 1 we observe how the box and arrow shapes changed the fluid flow. In these images (and all subsequent ones) the darker red represents faster flow. Since both shapes were symmetric and we used perfectly symmetric flow conditions, the resulting images have symmetry across the horizontal center axis.

We can see the flow impacting the front of the box creating an area of low flow, while the sharp corners abruptly create high velocity flows around the sides. Also evident is the turbulent area behind the box as the fluid attempts to 'fill' the empty space and swirls in strange elongated eddies. In contrast, the arrow disrupts the flow much less. The leading point slices through the fluid, parting it into slightly faster moving areas around the shape before it is rejoined behind the trailing point. Here we don't see any turbulent flow. We also note the difference in color contrast between the box and arrow flows. The general flow around the box is much slower than the high flow along its edges, while the arrow has a more compact color gradient (notice how the box image is overall lighter with select darker spots). These differences result from the arrow being a more aerodynamically efficient shape.

In figure 2 we see another set of images representing fluid flows around two objects. In this case these objects were chosen for their ability to create turbulent flows.

Figure 2

Parabola Swirl



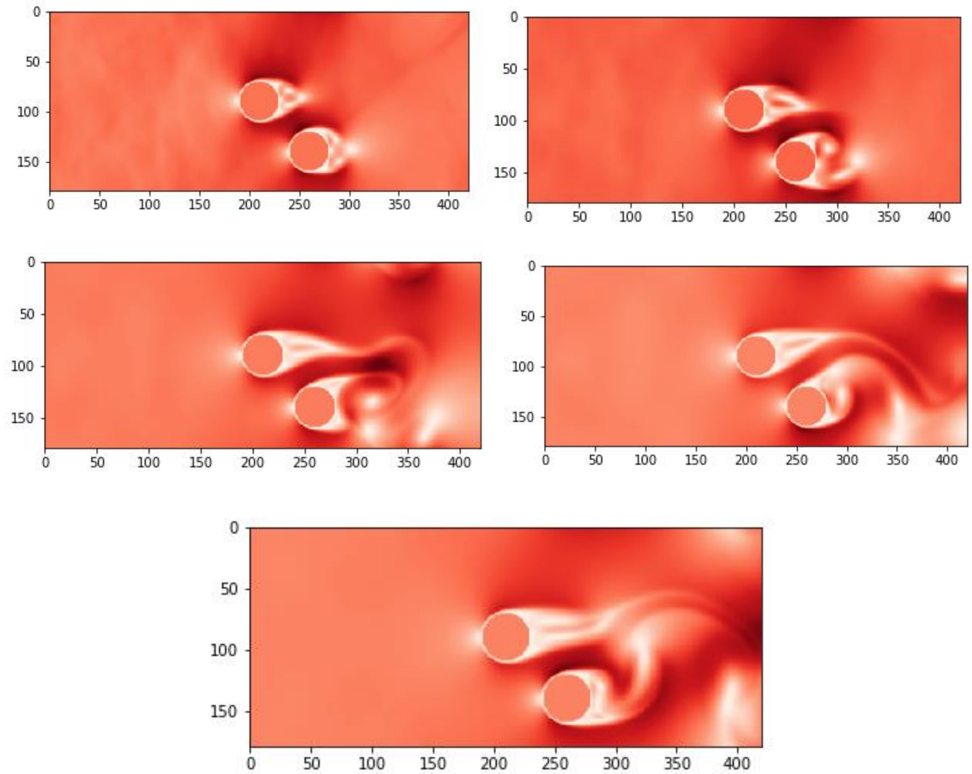
On the right side of figure 2 are three stages of the flow evolution around a tall flat oval. We observe in the first image the creation of symmetric vortexes swirling off the top and bottom edges. These are like the eddies created behind the box and indicate that having large open areas behind shapes results in the creation of turbulent flow. The subsequent images show the created vortexes being stretched in the horizontal direction in a stable way. This contrasts the right images which feature an asymmetric shape.

On the right side we observe the flow created by a parabola shape. Here we first note the influence of our periodic vertical boundary conditions; the edge of the shape which impacts the top of the image disrupts the flow at the bottom of the image. We also observe very unstable, asymmetric flow created by the tip of the parabola, particularly in the bottom-most image where there are clear uneven wobbles following the initial vortex. These wobbles are perhaps a result of interactions between the two asymmetric edges. We also note that instead of creating a signal vortex which was horizontally stretched, like in the right-side case, a vortex was created which maintained its shape and was horizontally translated.

Through the images in figures 1 and 2 we have analyzed several different object shapes and the flows they generate. We now examine interactions between the flows of more than one object.

Figure three shows five steps of the flow patterns created by starting with an initial circle then adding a second one translated behind and below the first.

Figure 3



In this setup we observe an unstable, asymmetric and almost random flow. The wake of the first ball stays relatively consistent in it shape but is strongly affected by the swirling chaotic behavior of the flow around the second ball. The second ball, being influenced by the wake of the first, creates no consistent wake, instead we observe a wildly swirling and wiggling mess. This experiment indicates that stable flows are very dependent on the state of the incoming fluid, and the disruption caused by an object can quickly become chaotic when the flow is asymmetric.

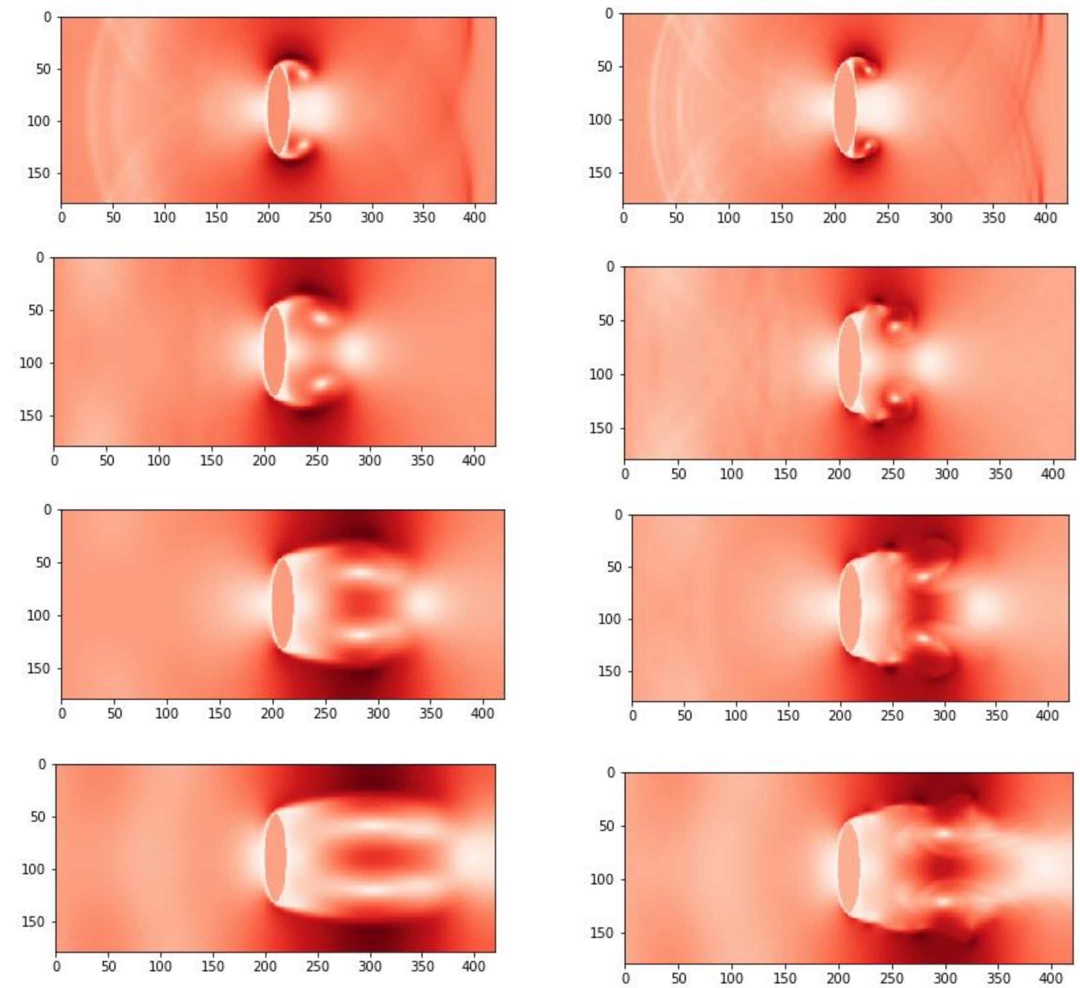
Having experimented with different effects which can be created by altering the shape, position and number of objects in the flow, we now turn to understanding how the flow can be affected by altering the properties of the fluid itself. To do so, several more experiments where conducted. In these tests, all but one parameter were maintained and the changes in the fluid flow were observed for variations of that single parameter To have an interesting but understandable flow, the 'swirl' object from figure 2 was chosen for these tests.

The parameters which could be varied in our provided code where: size and shape of the domain, Reynold's number, location and shape of the hard objects, velocity of the fluid and number of time iterations. Number of time iterations was not varied for obvious reasons, location and shape have been tested above and size/shape of the domain merely change the scale of the simulation. Thus, only velocity of the fluid and Reynold's number were tested for their effects on the fluid flow.

Figure 4 shows four images of flow pattern for the ‘swirl’ shape with the value of the Reynolds number at 50 and 200. For comparison, the value had been set at 100 for all previous images.

Figure 4

Reynolds Number: 50 200



In this figure we observe how a higher Reynold’s number affects the flow. The images on the right show a markedly more unstable and turbulent flow than those on the left. They feature a sharper formation of vortices in the first step and, although these vortices do evolve in a similar way in both cases, on the right side they are perturbed consistently by turbulences, resulting a very different state by the final images. We see that while on the left the vortices seem to stretch along with the flow, the right-side vortices maintain a more rounded shape and warp the surrounding flows into their spiral. The left side images appear soft and smooth in comparison to the right. Thus, we observe that a higher Reynold’s number results in more turbulent and unsteady flow.

A similar experiment was conducted by changing the velocity of the flow while maintaining a Reynold's number of 100, however, the only notable change between low and high velocity was the length of the stretched vortices and thus no further analysis was conducted.

Based on the experiments conducted above we can conclude that changing the shape, position and number of objects can create unique and varied flow patterns, and the unstable, turbulent aspects of these flow patterns will be more pronounced with a higher Reynolds number.

With this experimental understanding of the system gained, we now turn to a theoretical analysis of the NS and LB equations.

Understanding the system theoretically

We begin with the Navier Stokes equation for conversation of fluid flow

$$\frac{\partial u}{\partial t} + (u \cdot \nabla)u = \frac{1}{Re}\nabla^2 u + g$$

Where u is the velocity field, Re is the Reynolds number and g is the acceleration due to gravity. In words, this equation reads 'the rate of change of u with respect to time plus the dot product of u with the gradient of itself is equal to one over the Reynolds number times the Laplacian of u, plus the acceleration due to gravity.' To unpack this, we consider each piece individually.

The rate of change of u with respect to t is relatively straightforward, simply representing how the velocity field changes over time.

The dot product of u with the gradient of itself is in this case called the advection term. Advection is the flow of a fluid from one place to another. We can see how this would be determined using this formula. By taking the gradient of u we are calculating a sort of vector derivative. The result of the gradient on a field would be a vector field where the vector at each point indicates the direction over which the field was changing, i.e a sort of 'flow'. By then taking the dot product of this with all the vectors of u we get the advection of the system, a field representing the direction of the system flow at every point.

On the opposite side of the equation, g is trivial, but the second term is not. The Laplacian of u is called the diffusion term, which follows because the Laplacian calculates the net flux of a vector in every direction, i.e. the way the field is diffusing.

Thus, the righthand side represents the fluid flow summed with the change in the filed over time and is equated to the left which represents the diffusion of the fluid.

We can then see why a high Reynolds number resulted in a more turbulent flow, because the higher we value the Reynolds the less influence the diffusion term has on the behavior of the system and thus the less the system will diffuse back to equilibrium during its flow. High Reynolds number means the system is dominated by the flow itself.

With this understanding of how the NS conservation of flow equation works, we move forward to understand the LBE.

The Lattice Boltzmann equation written in terms of discrete time and length steps looks like so

$$f_i(x + e_i\delta_t\,, t + \delta_t) = f_i(x,t) + \frac{1}{\tau_f}(f_i^{eq} - f_i)$$

where $f_i(x,t)$ is the state of the system at time t and position x. Here the left-hand side is the state after one timestep. The right-hand side represents the state at the previous timestep plus an evolution term.

To understand how the system evolves we consider this evolution term. The f^{eq} function represents the equilibrium state for the fluid where it is at a constant velocity and pressure. Thus, the evolution term is taking this state and subtracting from it the current state of the cell at that time, then multiplying the result by a factor one over tau. Finally, it adds this term back to the initial state. The total effect of this action is to take a state and push it towards equilibrium by a certain factor before it evolves again. A state already at equilibrium will not be affected as the evolution term will be zero but a state not at equilibrium will be drawn towards it at a rate one over tau.

To illustrate how a state could leave equilibrium we imagine a cell near some hard surface being modeled using bounce-back mechanics. We notice that the left-hand side of the equation moves a given packet to a neighboring cell with some speed e_i in direction i. Thus, a cell near a hard surface at some time could have cells being pushed into from behind by the flow, and cells bouncing off the surface being pushed into it from in front. Thus, before the collision step it would be far out of equilibrium and would only relax back toward it through the collision. It would not reach equilibrium during this step unless tau equaled 1.

Using our analysis of the NS equation we can understand how these terms result in macroscopic effects. The equilibrium function is a sort of diffusion term, constantly trying to distribute the packets back into equilibrium, while the non-equilibrium parts of the BE are the flow terms, moving the particles from one cell to the next.

We now consider a way to understand our flow system in the physical world.

[1]

Physical representation

First we consider how to translate the values in our modeled system into standard units of time and distance. Given a system with N cells and knowledge that out cylinder has a diameter of L centimeters and our fluid is air at room temperature, we can find the duration of a single timestep.

---

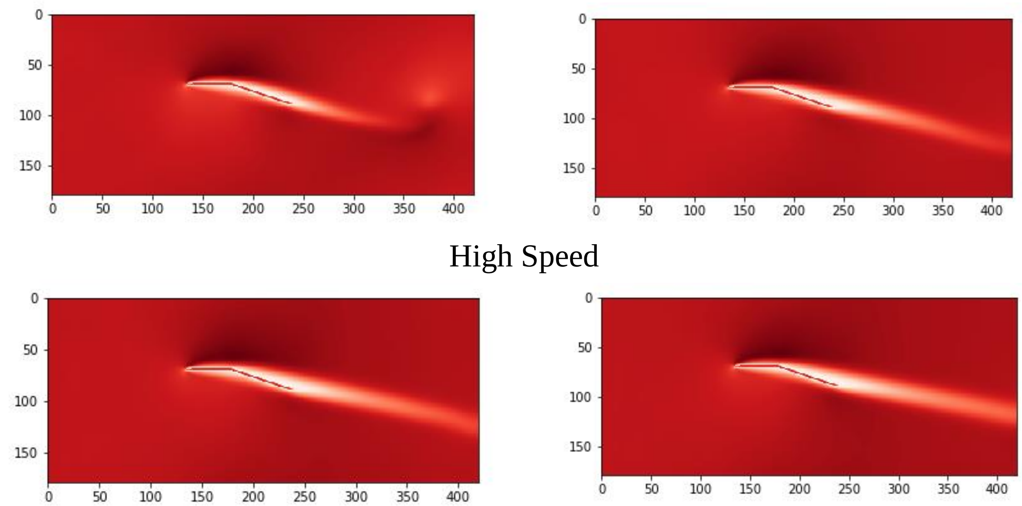
[1] Here is where the answer to question 5 would have gone, however, after completing the above analysis and creating the model as described, I could not find any meaningful way to relate it to my previous considerations. My model with the boundaries behaved exactly like the model without, save for the additional walls with slight wakes. Because of this and time constraints I omitted it.

Since the fluid is air at room temperature, we know the Mach constant Cs is the classical speed of sound. Having knowledge of both the number of cells and the diameter of our cylinder we can convert the distance from one cell to the next into centimeters. With these two calculations complete we can relate the Mach constant and the distance between cells to the time between timesteps using the equilibrium condition and thus find the time between timesteps in physical units.

This would enable us to use our system to model real world scenarios and calculate distances and times based on our in-system measurements. For example, figure 5 shows a simple model of an areofoil as on a plane wing at two different flow speeds.

Figure 5

Normal speed



Using our model, we could convert this system to a real would scale then determine at what speed the foil generates the most lift, calculate the length of the foil's wake, and perform other necessary analysis.

Conclusion

In this week's lab were provided with a working code to perform 2D computational fluid dynamics modeling using the Lattice Boltzmann method. This code enabled the modeling of the flow around various objects for a fluid with properties determined by editable constants. This essay began by outlining experiments with the model and analyzing how different sized and shaped objects induced different flows. It continued by recounting testing to determine how altering the fluid properties changed the flow behavior. With this understanding of how the system behaved in practice, this essay provided analysis of the Navier-Stokes and Lattice-Boltzmann equations theoretically, outlining how they worked to represented macroscopic properties. Finally, it gave a brief discussion of real world applications for the 2D CFD system.