OPhO Invitational: Fluid Dynamics

August 1, 2022

1 Introduction

Fluid dynamics is an important area in physics and engineering, and being able to simulate fluids properly is a challenging task that is applicable for rocketry, airplanes, cars, and even in designing pipes for air conditioning! The most popular technique is **computational fluid dynamics** (CFD), which numerically solves the Navier-Stokes equation. However, it is very difficult to set up and is very slow.

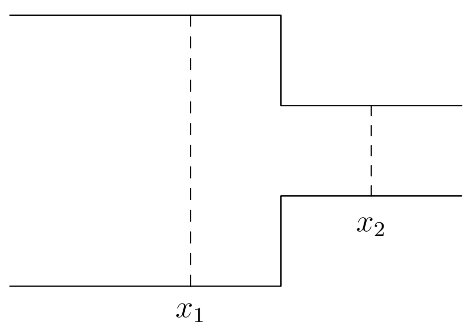
In this experiment, we will use a very simple simulation software that simulates fluids in two dimensions through the Lattice-Boltzmann algorithm (and is thus, *not* CFD), which you can find here.

Please take time to read the description on the page. **Make sure to keep all settings as default.** By dragging your cursor around, you can create different types of barriers, and clicking start will run the simulation. The first dropdown menu tells us the number of pixels. While a higher resolution will lead to more accurate results, it will be more strenous for the computer and harder to create boundaries. We recommend you to draw the barriers on a lower resolution but run the simulation on a higher resolution.

On the last row, we recommend checking **Flowlines** and **Sensor**. Enabling sensor will result in a cursor to appear on the screen that you can drag around, where the density ρ, horizontal speed u_x, and vertical speed u_y will all be shown. If for some reason the simulation is unstable, you may also check the **Data** button, which will record the data on the cursor as a function of time.

2 Mass Flow Conservation (15 pts)

We will first investigate if this implementation of the Lattice-Boltzmann algorithm correctly simulates the conservation of mass.



1. Create a tube that either opens up or closes down very abruptly such as in the diagram above. Pick two locations x_1 and x_2 such that one is in the wider section and the other is in the more narrow section. Compute the flow $\frac{dm}{dt}$ past both these points. (10 pts)
2. Do the two mass flow rates agree with each other? If not, suggest a possible reason why. (5 pts)

Note:

- You are free to pick the shape and dimensions of the tube and it does not need to be perfect.
- Please report your figures with uncertainties and make clear how you estimated and/or obtained them! You do not need to show your computations.
- Provide a screenshot of the boundaries and the parameters.

3 Law of the Wall (25 pts)

The Law of the Wall provides an emperical relationship between the average velocity of a turbulent flow at some point and the distance to a wall (or fluid boundary). Namely,

$$u_x = \frac{u_T}{\kappa} \ln\left(\frac{\rho u_T}{\mu} y\right) + C,$$

where

- u_x is the component of velocity parallel to the wall
- u_T is the shear velocity defined by $u_T = \sqrt{\tau_W/\rho}$,
- ρ is the density
- μ is the viscosity (specifically, the dynamic viscosity)
- κ, C are constants.

Note that $\tau_w = \mu \frac{\partial u_x}{\partial y}\Big|_{y=0}$ is the wall shear stress.

Studies have shown that κ ranges from 0.35 to 0.42. Using a straight line to model a wall, answer the following questions:

1. Is the relationship between u_x and y logarithmic? (10 pts)
2. What are the values of κ and C? (10 pts)
3. Does the value of κ agree with experiment? If it does not, suggest a possible reason why. (5 pts)

You are encouraged to add graphs.

Note:

- You are free to pick the length and direction of the boundary.
- Please report your figures with uncertainties and make clear how you estimated and/or obtained them! You do not need to show your computations.
- Provide a screenshot of the boundaries and the parameters.