Lattice Boltzmann Methods

BY MARTINO FERRARI

1 Introduction to Lattice Boltzmann methods

The **LBM** is an evolution of the lattice gas automata (LGA) method and it permits to simulate very accuratly and more easly on modern computer (as it use float numbers instead of integer only) the beahvior of a fluid.

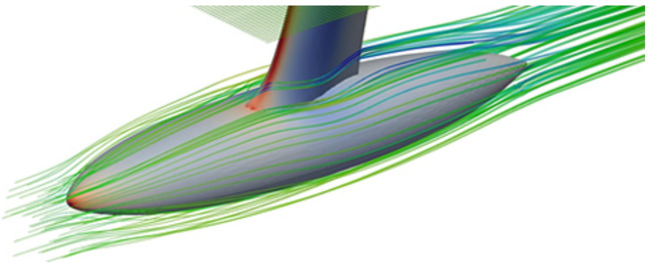

Figure 1. simulation of the hydro-dynamics of a keel

The application are multiples, from the study of the fluido-dynamics of a veichle (as in figure 1) to the simulation of the biological fluids in the body.

This is done solving the Navier-Stokes equation by the simulation of the streaming and collision process of the particles.

The Navier-Stokes equations

However this simulation has many limitation:

- sub-sonic speed only
- uncomprimible fluids only
- eating and chemical processes are not taken in account
- and many others

The use of floating point data permits to represent the density of the fluid and not the number of particles as in the LGA, in this way is possible to reach better results using smaller meshes.

To avoid isotropy problem the LBM use 9 direction for 2D simulations, the 8 neighbours and the cell itself.

The simulation as told before is divided in 2 main step:

1. collsion step: where the particles collide inside the same cell
2. streaming step: where the particles propagate from a cell to its neighbours

2 Implementation

The implementation in python 3 and is based on both the given template and the code shown in class.

Each iteration is divided in outflow and inflow condition, equilibrium condition, collision step and streaming step.

3 Results

With the initial configuration the results was very stable and no trubolence were produced (file :*vel_re0010.gif*):

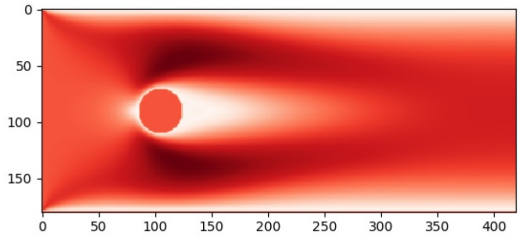

Figure 2. simulation with *Reynolds number = 10*

To be able to have some turbolence in the fluid I had to change Reynolds number to a much higher value (file: *vel_re1000.gif*).

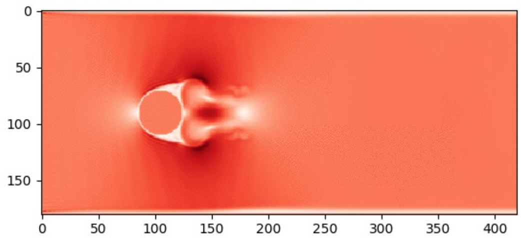

Figure 3. Example of turbolence with *Reynolds number = 1000*

Adding more obstacles gave interesting results:

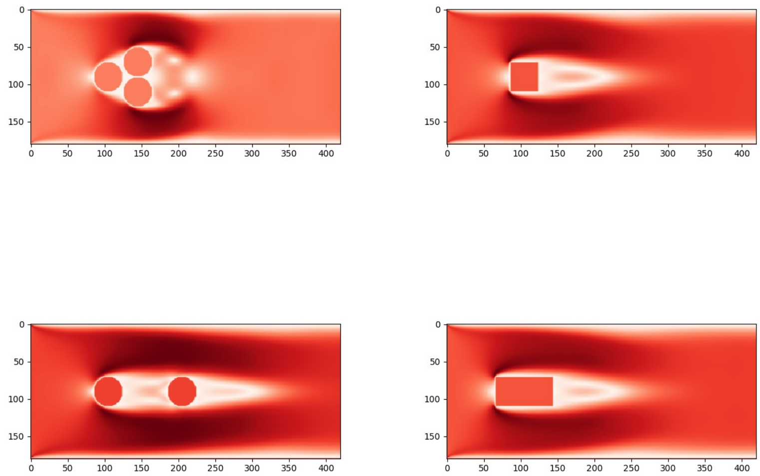

Figure 4. multiple obstalce (and Re = 50)

3.1 Questions:

1. adding new obstacle create more complex turbolences and patterns as shown in figure 4
2. the speed and the viscosity of the fluid are the variable that affect the more the simulation
3. to add the walls is very simple and all previous images are taken with the walls on:

```
obstacle[:,0] = True
obstacle[:,-1] = True
```