

# Simulation and modeling of natural processes

Week 5: Lattice Boltzmann modeling of  
fluid flow

Jonas Latt

# Week 5: Overview

- This week: computer simulation of fluid flow.
- Extension of discrete Cellular Automata to continuum lattice Boltzmann models.
- Part 1: General theory of fluid dynamics.
- Part 2: Write a Python program to simulate fluid flow with the Lattice Boltzmann method.

# Computational Fluid Dynamics: Overview

# What's a fluid?

A fluid ...

- ... is a substance that continually deforms under stresses.
- ... resists deformations only lightly, because of viscosity.
- ... can adopt the shape of any container into which it flows.

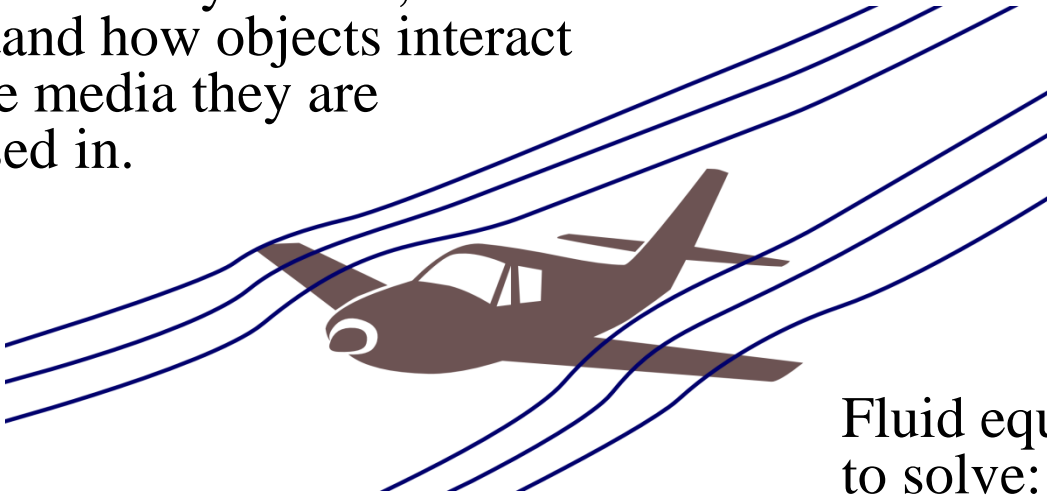
Fluids include:

- Liquids
- Gases
- Plasmas

Fluid = Continuum Model
-------------------------

# Why fluid dynamics?

Through fluid dynamics, we understand how objects interact with the media they are immersed in.



Fluid equations are difficult to solve: this is a case for numerical simulation.

# Fluid dynamics is everywhere

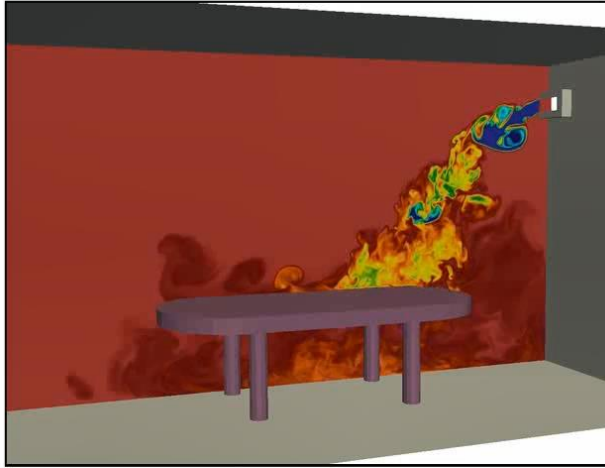
Important application areas are found

- In physics.
- In biology and medical physics.
- In chemistry.
- In geology.
- ... and many more.

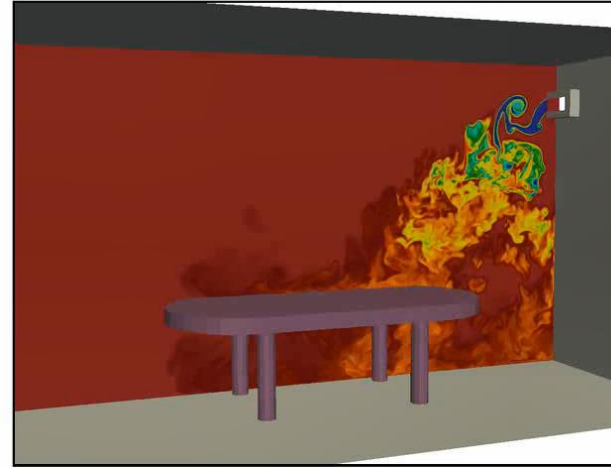
# Example 1: Heat convection

Air-conditioner in a meeting room

 [palabos.org](http://palabos.org)



Fixed air-conditioner



Sweeping air-conditioner

# Example 2: Blood flow in an artery



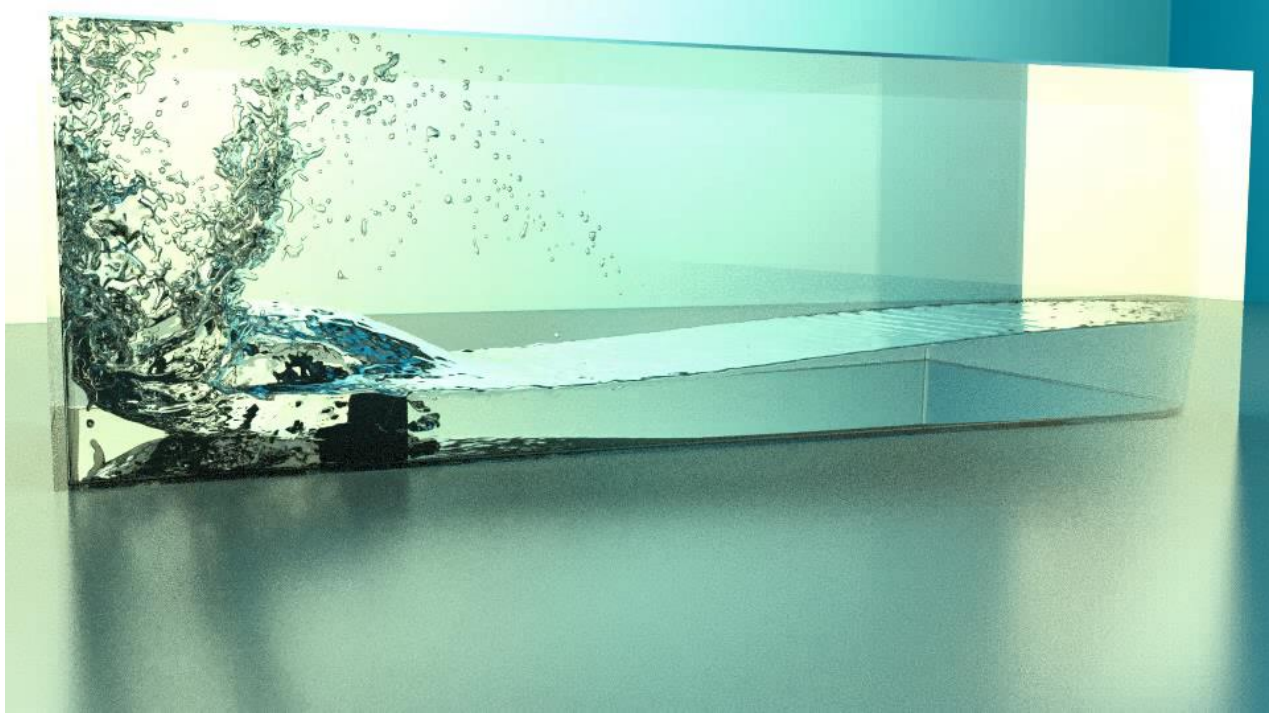
Segment of a human artery with an aneurysm (balloon-like bulge).

Simulation:

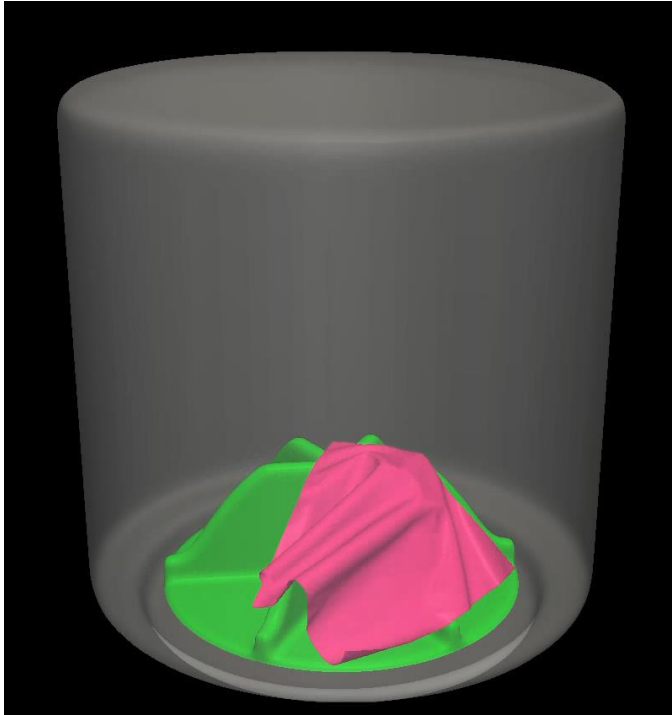
- Fluid dynamics of pulsatile blood flow.
- Embedded red blood cells.
- Blood clotting inside aneurysm, with change of geometry.



# Example 3: Collapse of a water column



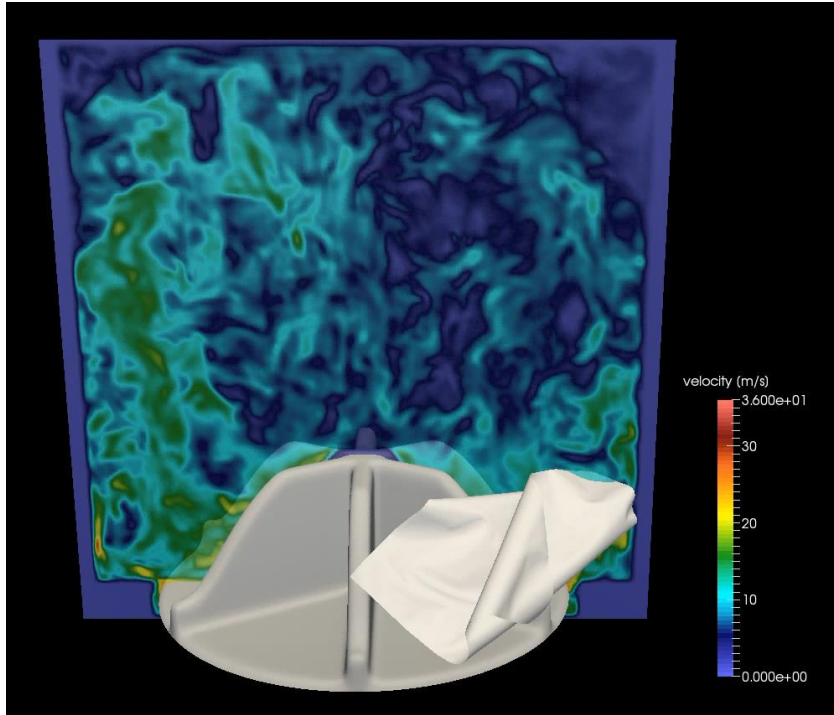
# Example 4: washing machine



## Simulation:

- Water fluid flow.
- Rotating agitator.
- Embedded flexible tissue, 2-way coupled with the water.

# Example 4: washing machine



Washing machine:  
velocity field.

# Credits

I'd like to thank

- **Orestis Malaspinas** (University of Geneva) for the blood flow example.
- **Dimitrios Kontaxakis** (FlowKit Ltd, Lausanne) for the example of a washing machine.
- **Andrea Parmigiani** (University of Geneva) and **Andrea Di Blasio** (FlowKit Ltd, Lausanne) for water column example.

*End of module*

Computational Fluid Dynamics: Overview

*Coming next*

Computational Fluid Dynamics: Equations and  
challenges

# Computational Fluid Dynamics: Equations and challenges

# The Navier-Stokes equations

The basis: Navier-Stokes equations for incompressible flow.  
They don't account for

- Fluid compressibility in gases.
- Thermal effects.
- Chemical reactions.
- ... many more.

And yet, they are very frequently and successfully used, for both liquids and gases.

# The Navier-Stokes equations

$$\partial_t \mathbf{u} + (\mathbf{u} \cdot \nabla) \mathbf{u} = -\frac{1}{\rho_0} \nabla p + \nu \Delta \mathbf{u}$$

Flow acceleration

Convective acceleration

$$\nabla \cdot \mathbf{u} = 0$$

Pressure Gradient

Continuity equation:  
Fluid incompressibility

$\mathbf{u}(\mathbf{x}, t)$	Velocity
$p(\mathbf{x}, t)$	Pressure
$\nu$	Viscosity

Viscous dissipation



# Dimensionless formulation

$$\partial_t \mathbf{u} + (\mathbf{u} \cdot \nabla) \mathbf{u} = -\frac{1}{\rho_0} \nabla p + \nu \Delta \mathbf{u}$$



Reynolds number:

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

Choose a...

Characteristic  
velocity:  $U$

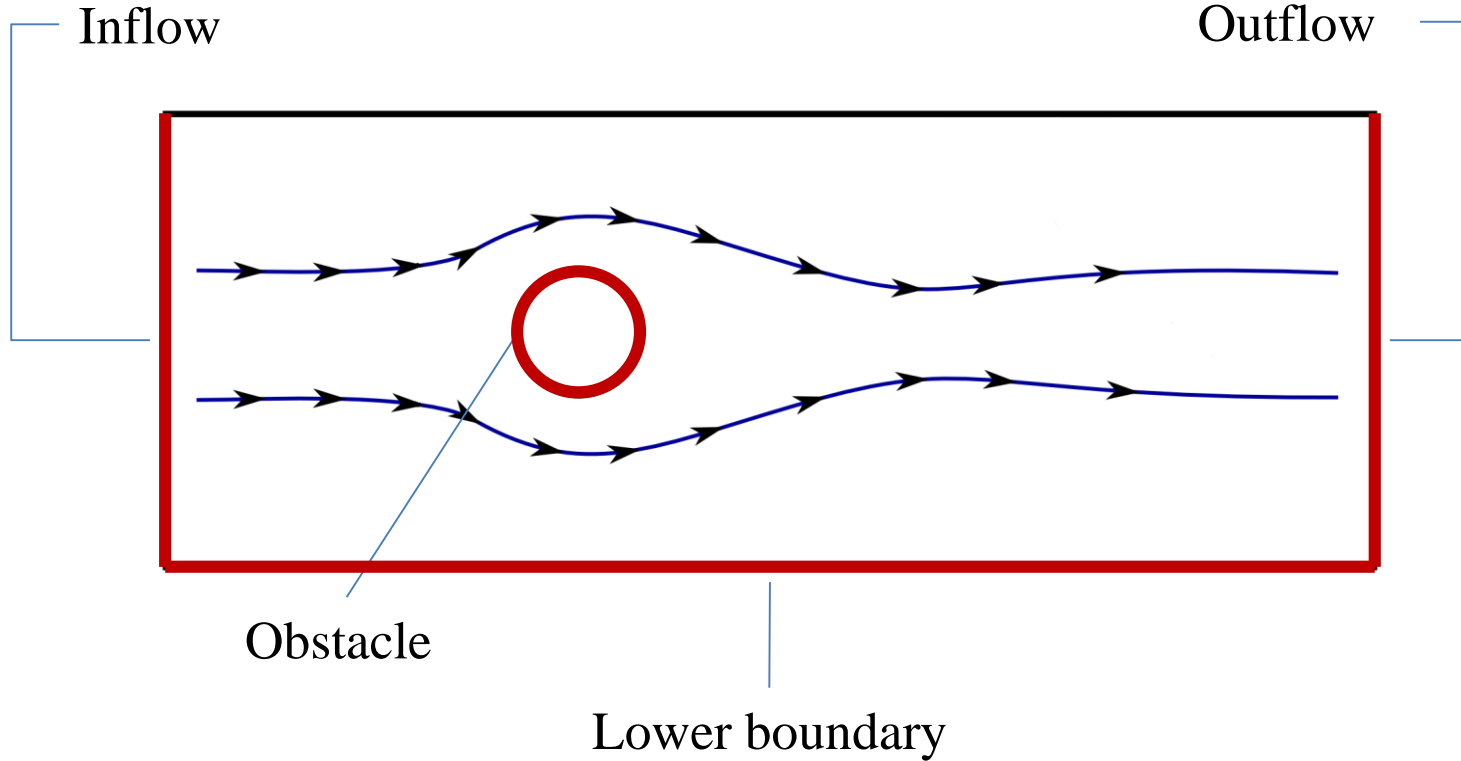
Characteristic  
length:  $L$

$$\begin{aligned} \vec{u}^* &= \frac{\vec{u}}{U} & \partial_t^* &= \frac{L}{U} \partial_t \\ p^* &= \frac{p}{\rho_0 U^2} & \vec{\nabla}^* &= L \vec{\nabla} \end{aligned}$$



$$\partial_t^* \mathbf{u}^* + (\mathbf{u}^* \cdot \nabla^*) \mathbf{u}^* = -\nabla p^* + \frac{\nu}{UL} \Delta \mathbf{u}^*$$

# Boundary Conditions



*End of module*

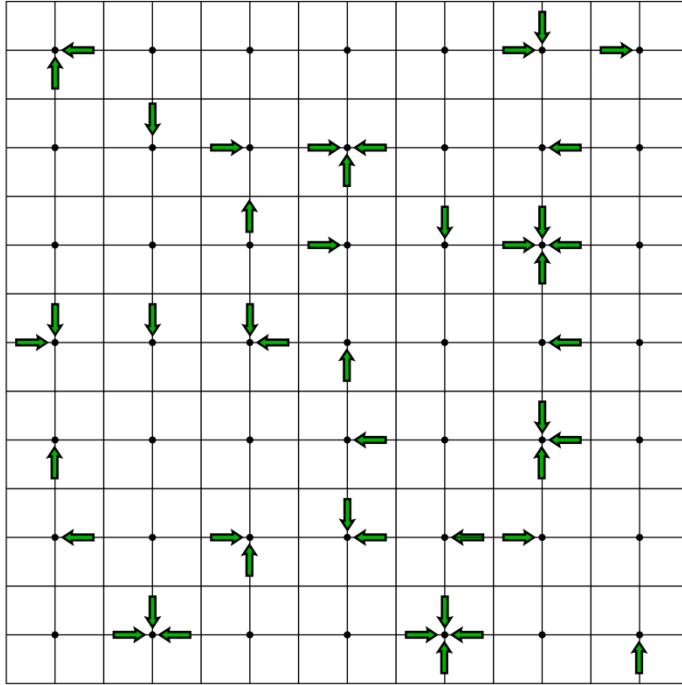
Computational Fluid Dynamics: Equations and  
challenges

*Coming next*

From Lattice Gas to Lattice Boltzmann

# From Lattice Gas to Lattice Boltzmann

# Lattice Gas Automata: Reminder



## The lattice gas automaton HPP

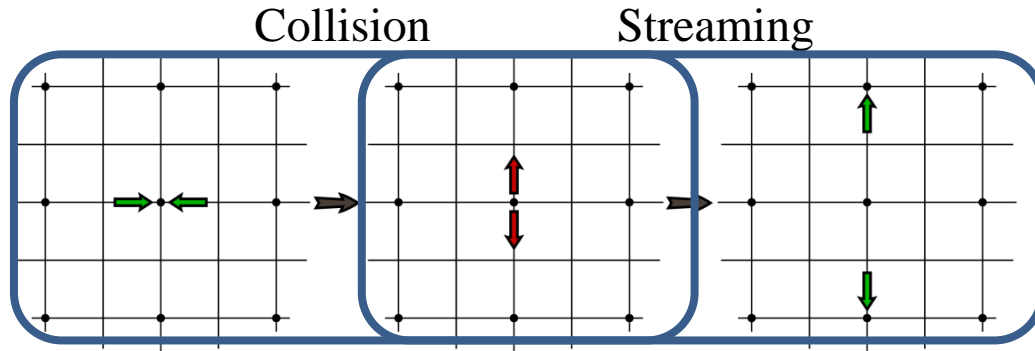
- Discrete Boolean states.
- Each cell is populated by four particles at most.
- Each of the four storage locations corresponds to a direction of propagation of the particle.

## Comment

- This mesoscopic model describes the dynamics of a gas, not a liquid.

# Lattice Gas Automata: Reminder

- When several particles meet on a node, they collide: all physics takes place here.
- After collision, particles are streamed to a nearest neighbor.

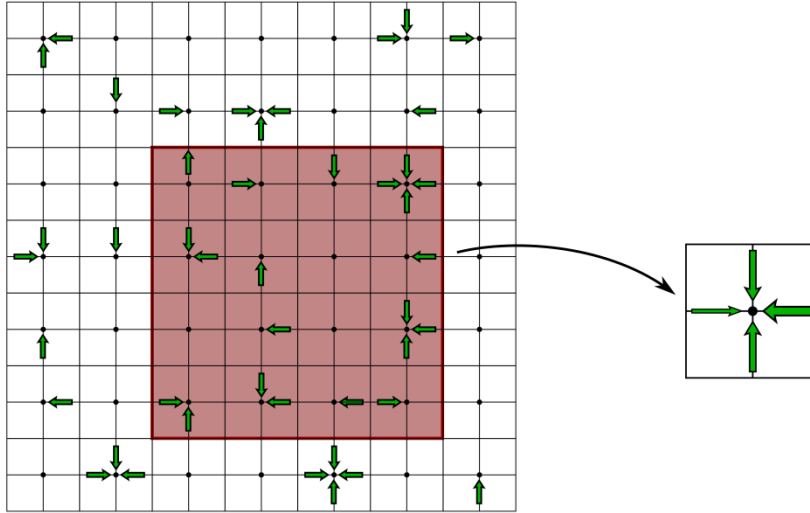


Our color scheme:

→ Green: pre-collision

→ Red: post-collision

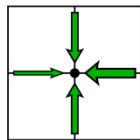
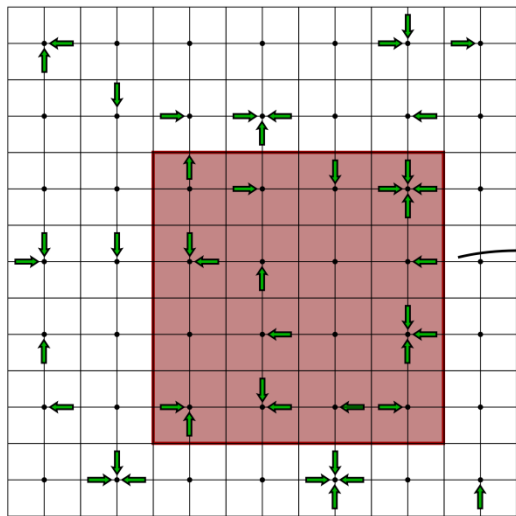
# From discrete to continuum variables



## **In a lattice gas automaton:**

- Run the simulation with discrete variables.
- At the end, extract macroscopic variables by taking averages.

# Lattice Boltzmann Method: Idea

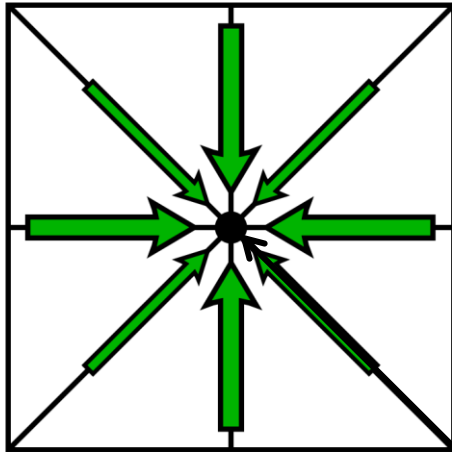


- Run simulation with continuum variables right away, to save space.
- Continuum model is derived from discrete method by means of statistics.

- Particle density: real-valued.
- Arrow thickness stands for particle density.



# Lattice Boltzmann Method: details

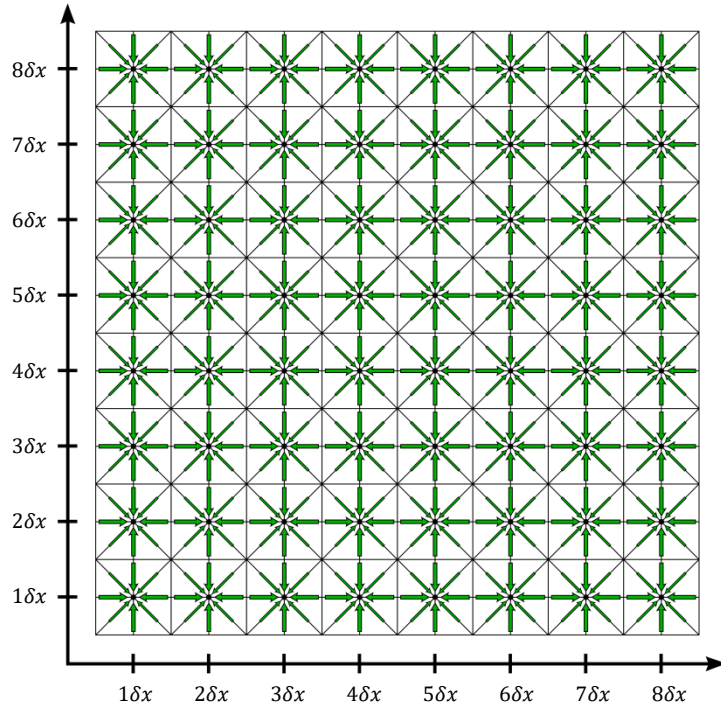


Lattice Boltzmann 2D model D2Q9:

- Four directions aren't quite enough.
- Nine directions: eight connections to nearest neighbors + «rest population».
- Variables representing particle densities are called «populations».

«Rest population» particles stay on a cell, don't travel to neighbors during streaming.

# Lattice Boltzmann Variables



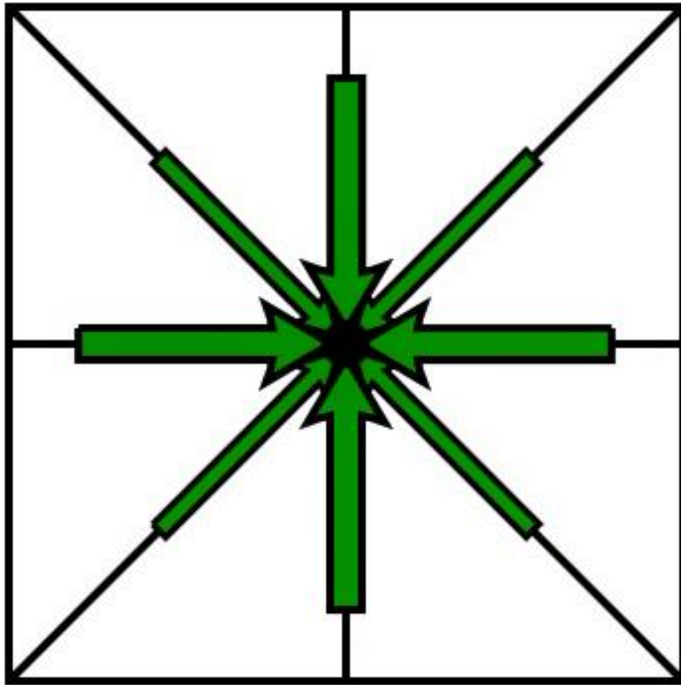
## Space Discretization:

- Every cell represents flow variables at a point in space.
- Equal distance  $\delta x$  between cells in x- and y-direction.
- To represent space with 8x8 points, we need a total of  $8 \times 8 \times 9 = 576$  floating point variables.

## Python code to allocate memory:

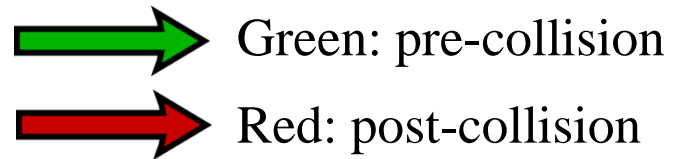
```
f = zeros(9, 8, 8)
```

# Collision step

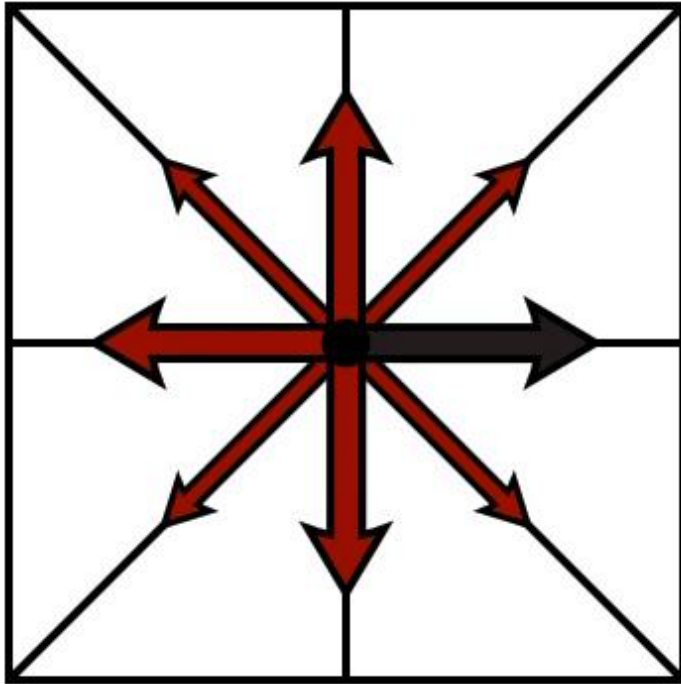


- Like in lattice gas: collision maps pre-collision populations to post-collision populations.
- All populations are modified during collision. Changes are not visible on video, because they are small perturbations.

Our color scheme:



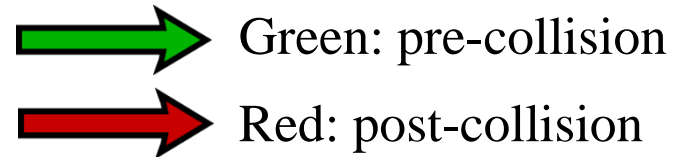
# Collision step



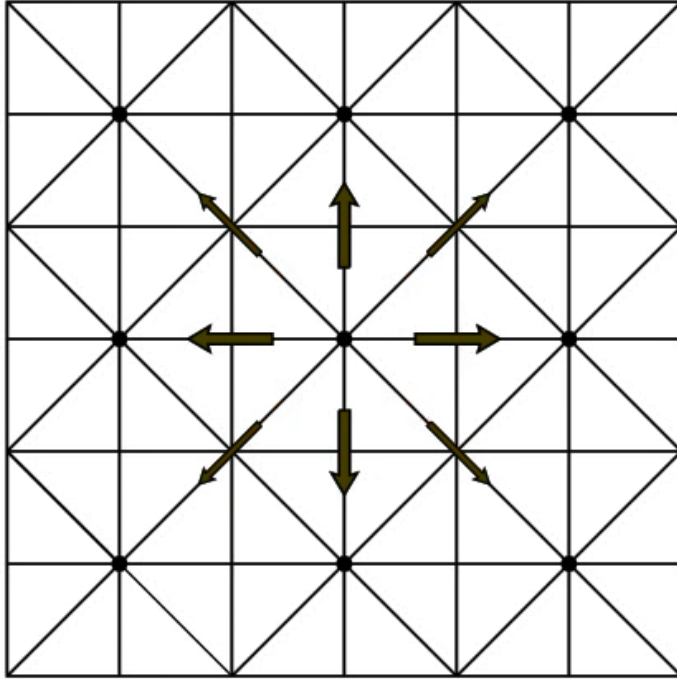
Model:

- Collision is instantaneous.
- Collision is localized at cell coordinates.

Our color scheme:



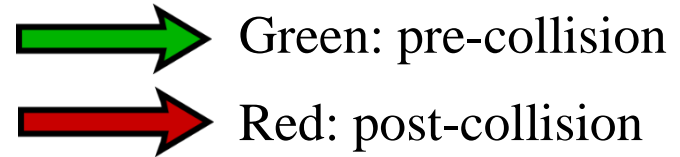
# Streaming step



Model:

- Streaming takes the system from time  $t$  to time  $t + \delta t$ .
- Streaming includes nearest-neighbor access.

Our color scheme:



*End of module*

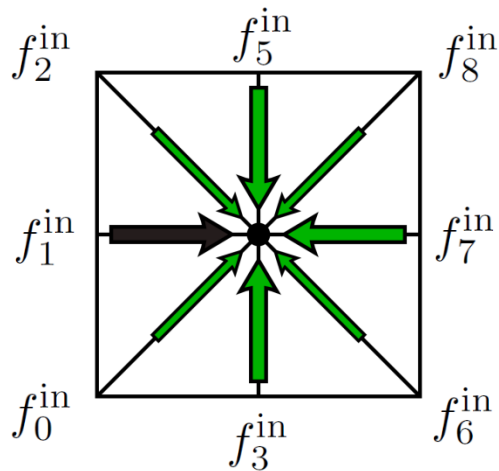
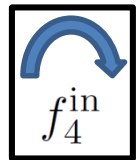
From Lattice Gas to Lattice Boltzmann

*Coming next*

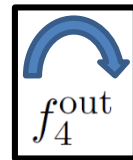
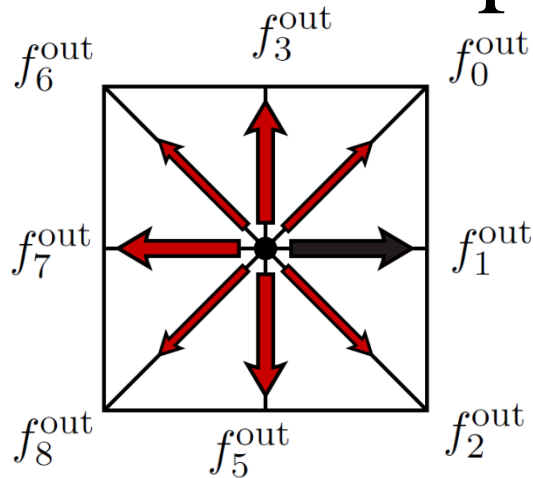
Macroscopic Variables

# Macroscopic Variables

# Back to the collision step



Pre-collision:  $f_i^{\text{in}}(\mathbf{x}, t)$   
«Incoming populations»



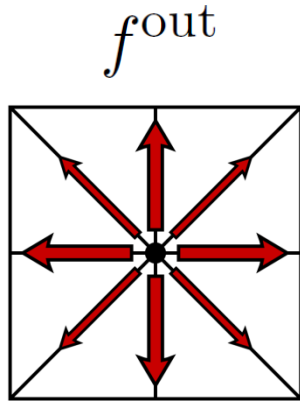
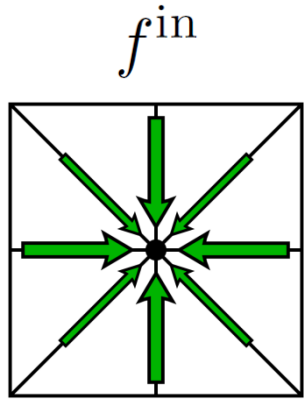
Post-collision:  $f_i^{\text{out}}(\mathbf{x}, t)$   
«Outgoing populations»

The index  $i$  runs from 0 to 8.



# Pre- and post-collision: variables

In our Python code, we save incoming and outgoing populations in separate matrices.



## Python code:

```
# assign some size  
 $nx, ny = \dots$   
 $f_{in} = \text{zeros}(9, nx, ny)$   
 $f_{out} = \text{zeros}(9, nx, ny)$ 
```

# Density

Particle density is defined the same as in a lattice gas automaton:

$$\rho(\boldsymbol{x}, t) = \sum_{i=0}^8 f_i^{\text{in}}(\boldsymbol{x}, t)$$

Each cell has its own density, which is the sum of the nine populations.

# Density

$$\rho(\boldsymbol{x}, t) = \sum_{i=0}^8 f_i^{\text{in}}(\boldsymbol{x}, t)$$

**Python code:**

```
rho = zeros((nx, ny))  
for ix in range(nx):  
    for iy in range(ny):  
        rho[ix, iy] = 0  
        for i in range(9):  
            rho[ix, iy] += fin[i, ix, iy]
```

**Simpler and faster, with NumPy  
array-based syntax:**

```
rho = sum(fin, axis=0)
```

# Pressure

Pressure is proportional to density, according to the ideal gas law, at constant temperature:

$$p = c_s^2 \rho$$

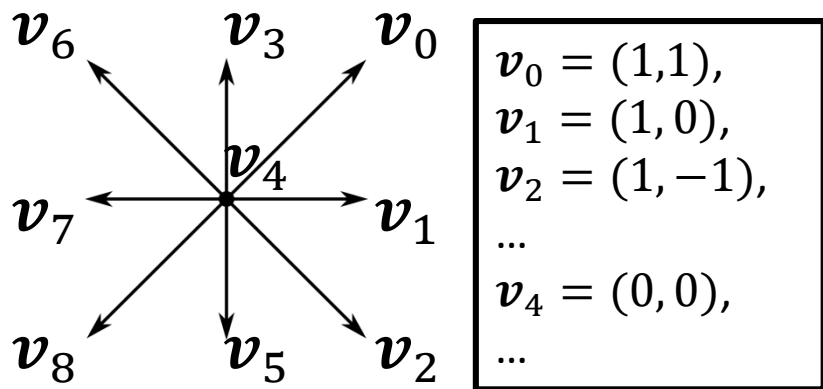
The speed of sound  $c_s$  is, in our D2Q9 model, a lattice constant:

$$c_s^2 = \frac{1}{3} \frac{\delta x^2}{\delta t^2}$$

Units: Say that  $\delta x$  has units of meters  $[m]$  and  $\delta t$  has units of seconds  $[s]$ . Then pressure has units of  $\frac{m^2}{s^2}$ . Multiply this by the physical density of our fluid (e. g.  $1 \frac{kg}{m^3}$ ) to obtain units of Pascal  $\left(\frac{kg}{ms^2}\right)$ .

# Velocity

Let's introduce a set of 9 vectors:



Again, velocity is defined just the same as in a lattice gas automaton:

$$u(\mathbf{x}, t) = \frac{1}{\rho(\mathbf{x}, t)} \frac{\delta x}{\delta t} \sum_{i=0}^8 \mathbf{v}_i f_i^{\text{in}}(\mathbf{x}, t)$$

They are called lattice velocities: If there's a cell at position  $\mathbf{x}$ , there's also one at  $\mathbf{x} + \mathbf{v}_i \delta t$ .

# Velocity

```
v = array([ [ 1, 1], [ 1, 0], [ 1, -1], [ 0, 1], [ 0, 0], [ 0, -1], [-1, 1], [-1, 0], [-1, -1] ])
```

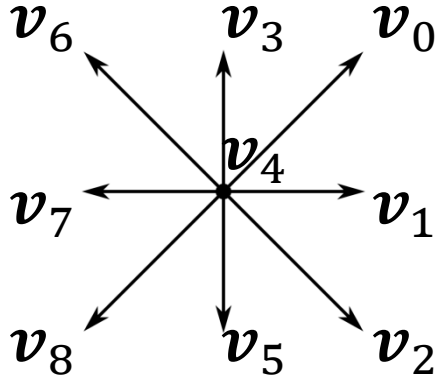
## Traditional Python code:

```
u = zeros((2, nx, ny))
for ix in range(nx):
    for iy in range(ny):
        u[0, ix, iy] = 0
        u[1, ix, iy] = 0
        for i in range(9):
            u[0, ix, iy] += v[i,0] * fin[i, ix, iy]
            u[1, ix, iy] += v[i,1] * fin[i, ix, iy]
        u[0, ix, iy] /= rho[ix, iy]
        u[1, ix, iy] /= rho[ix, iy]
```

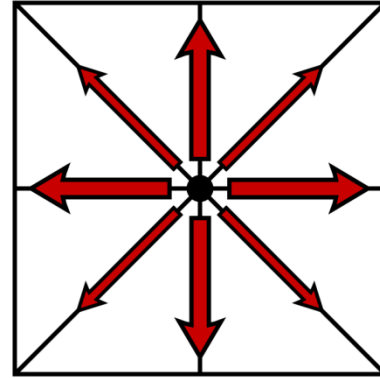
## Simpler and faster, with NumPy array-based syntax:

```
u = zeros((2, nx, ny))
for i in range(9):
    u[0,:,:] += v[i,0] * fin[i,:,:]
    u[1,:,:] += v[i,1] * fin[i,:,:]
u /= rho
```

# Don't be confused



- These are 9 2D vectors.
- They have constant values.



- These are 9 scalar values.
- Their value is time-dependent.

*End of module*

Macroscopic variables

*Coming next*

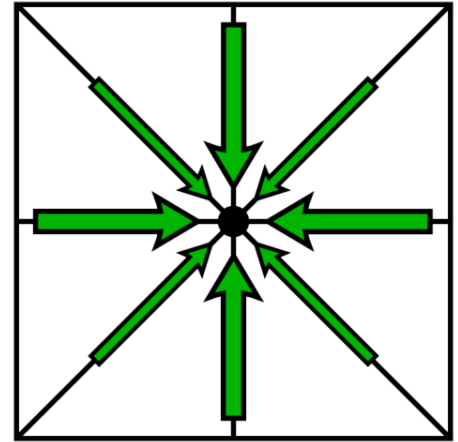
Collision step: the BGK model



Collision step: the BGK model

# What's the value of the populations?

- So far, we have interpreted populations as particle averages of a lattice gas automaton.
- Alternative interpretation: populations are probability densities of particles in a real gas.
- In *kinetic theory*, gas densities are described by a single *probability density function*  $f(\mathbf{x}, \mathbf{v}, t)$ .
- The motion of  $f$  is described by the *Boltzmann equation*.

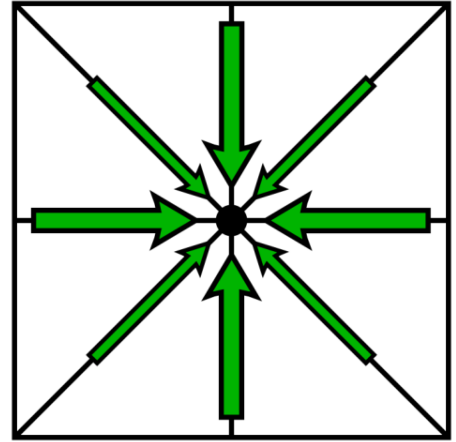


# What's the value of the populations?

- When a gas is macroscopically at rest, or has a constant velocity  $\mathbf{u}$ , its molecules are still moving.
- The distribution of their velocities  $\mathbf{v}$  at every point  $\mathbf{x}$  in space is given by a *Maxwell-Boltzmann distribution*:

$$f(\mathbf{x}, \mathbf{v}, t) \sim e^{-|\mathbf{v}-\mathbf{u}|^2}.$$

- Even when it moves, a gas remains close to its equilibrium state. We consider its motion as a *perturbation around the equilibrium*.



# Collision: BGK model

Inspired by kinetic theory, we formulate the collision term in lattice Boltzmann as a relaxation to local equilibrium:

$$f_i^{\text{out}} - f_i^{\text{in}} = -\omega \left( f_i^{\text{in}} - E(i, \rho, \vec{u}) \right)$$

- $E$  is the local equilibrium. It depends on the macroscopic variables and has a different value for every direction  $i$ .
- The parameter  $\omega$  is the frequency of relaxation.
- This collision model is called the *BGK model*.

# Collision frequency and viscosity

- If  $\omega$  is small, the fluid converges only slowly to its equilibrium: it is highly viscous.
- More generally, one can show that the fluid viscosity depends inversely on the relaxation parameter  $\omega$ :

$$\nu = \delta t c_s^2 \left( \frac{1}{\omega} - \frac{1}{2} \right)$$

# Collision: BGK model

The equilibrium is obtained as a truncated series of the Maxwell-Boltzmann distribution:

$$E(i, \rho, \mathbf{u}) = \rho t_i \left( 1 - \frac{\mathbf{v}_i \cdot \mathbf{u}}{c_s^2} + \frac{1}{2 c_s^4} (\mathbf{v}_i \cdot \mathbf{u})^2 - \frac{1}{2 c_s^2} |\mathbf{u}|^2 \right)$$

- The constants  $t_i$  compensate for the different lengths of velocities  $\mathbf{v}_i$ .
- $t_i$  is 1/9 for orthogonal directions, 1/36 for diagonal directions, and 4/9 for the rest velocity:

```
t = array([ 1/36, 1/9, 1/36, 1/9, 4/9, 1/9, 1/36, 1/9, 1/36])
```

# Collision: Python codes

Equilibrium:

$$E(i, \rho, \mathbf{u}) = \rho t_i \left( 1 + \frac{\mathbf{v}_i \cdot \mathbf{u}}{c_s^2} + \frac{1}{2 c_s^4} (\mathbf{v}_i \cdot \mathbf{u})^2 - \frac{1}{2 c_s^2} |\mathbf{u}|^2 \right)$$

We write the code in  
«lattice units», i.e.  
with  $\delta x = \delta t = 1$ .

```
def equilibrium(rho, u):  
    usqr = 3/2 * (u[0]**2 + u[1]**2)  
    eq = zeros((9, nx, ny))  
    for i in range(9):  
        vu = 3 * (v[i,0]*u[0,::] + v[i,1]*u[1,::])  
        eq[i,::] = rho*t[i] * (1 + vu + 0.5*vu**2 - usqr)  
    return eq
```

# Collision: Python codes

Collision:

$$f_i^{\text{out}} - f_i^{\text{in}} = -\omega (f_i^{\text{in}} - E(i, \rho, \vec{u}))$$

$$f_{out} = f_{in} - \omega * (f_{in} - eq)$$



*End of module*

Collision step: the BGK model

*Coming next*

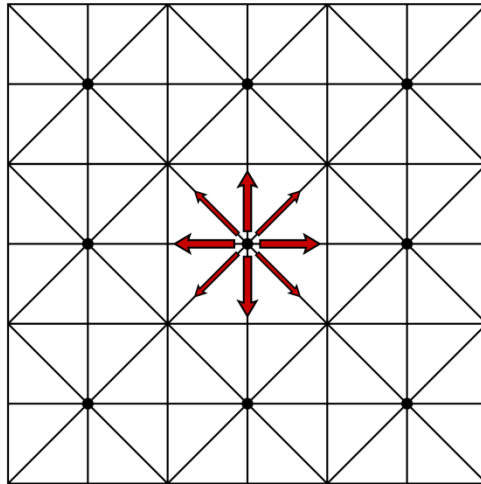
Streaming Step

The streaming step

# Streaming step

Streaming takes  
the system to  
the next time  
iteration,  $t + \delta t$

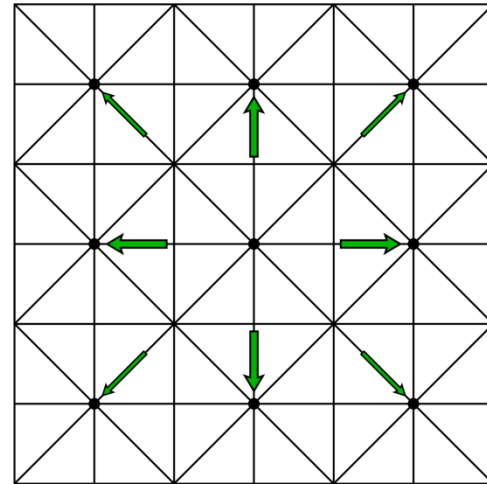
Time  $t$



Post-collision  $f^{\text{out}}$



Time  $t + \delta t$



Pre-collision  $f^{\text{in}}$



# Streaming step: code

Conventional Python code:

- On boundaries, we apply periodicity.
- If a population leaves the domain, it enters the domain again on the other side.

```
for ix in range(nx):  
    for iy in range(ny):  
        for i in range(9):  
            next_x = ix + v[i,0]  
            if next_x < 0: next_x = nx-1  
            if next_x >= nx: next_x = 0  
  
            next_y = iy + v[i,1]  
            if next_y < 0: next_y = ny-1  
            if next_y >= ny: next_y = 0  
            fin[i, next_x, next_y] = fout[i, ix, iy]
```

# Streaming step: code

Conventional Python code:

```
for ix in range(nx):  
    for iy in range(ny):  
        for i in range(9):  
            next_x = ix + v[i,0]  
            if next_x < 0: next_x = nx-1  
            if next_x >= nx: next_x = 0  
  
            next_y = iy + v[i,1]  
            if next_x < 0: next_x = nx-1  
            if next_x >= nx: next_x = 0  
            fin[i, next_x, next_y] = fout[i, ix, iy]
```

NumPy Array-based code:

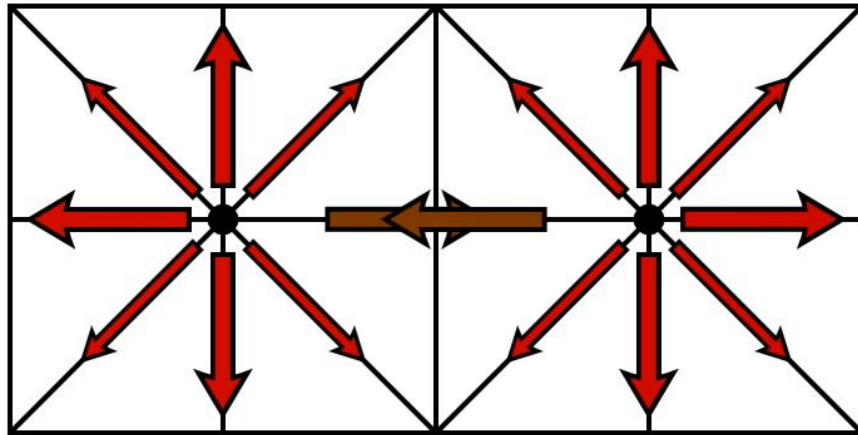
```
for i in range(9):  
    fin[i,:,:] = roll(  
        roll(fout[i,:,:], v[i,0], axis=0),  
        v[i,1], axis=1 )
```

# Do we really need double the memory?

- So far we allocated two sets of populations  $f^{in}$  and  $f^{out}$ .
- Could we spare one, and store both pre-collision and post-collision in the same variable  $f$ ?
- Answer: yes, but it needs some care.
- I will show how to do it. But for simplicity, we keep our code with two sets of populations.

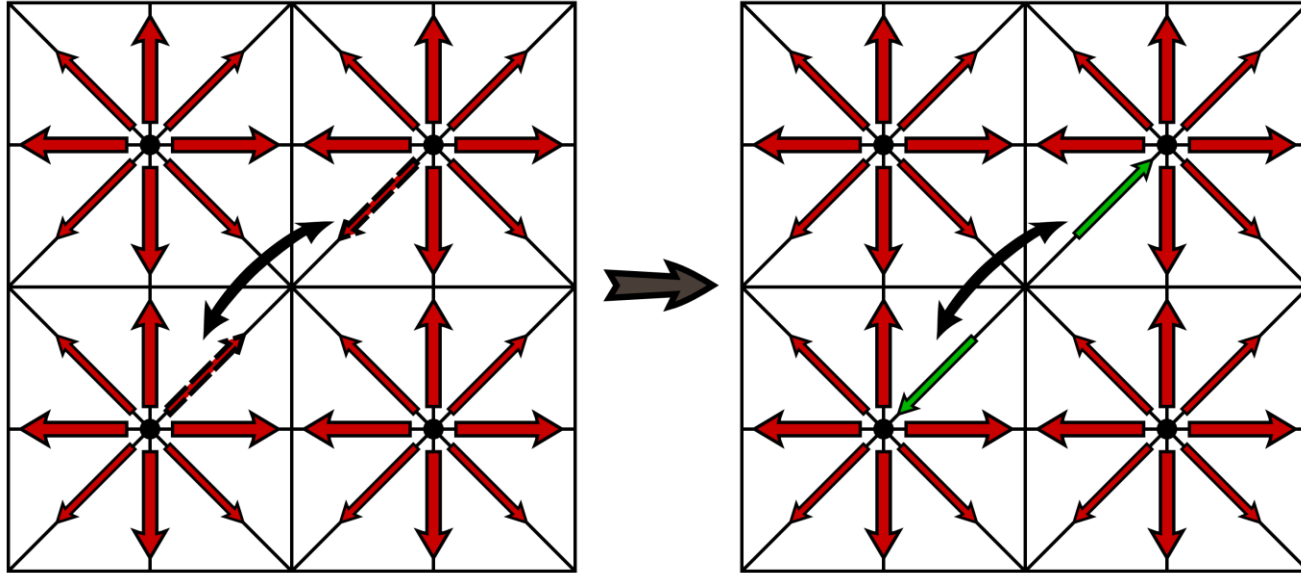
# Single-population implementation

- Collision step, no problem: it is local. Simply overwrite  $f^{in}$  with  $f^{out}$ .
- Streaming step, more tricky. As you stream populations to a neighbor, you risk overwriting one that is still needed.
- Solution: *stream to the neighbor* and *from the neighbor* at the same time, exchanging populations instead of overwriting them.



# Single-population implementation

This strategy works along all directions:



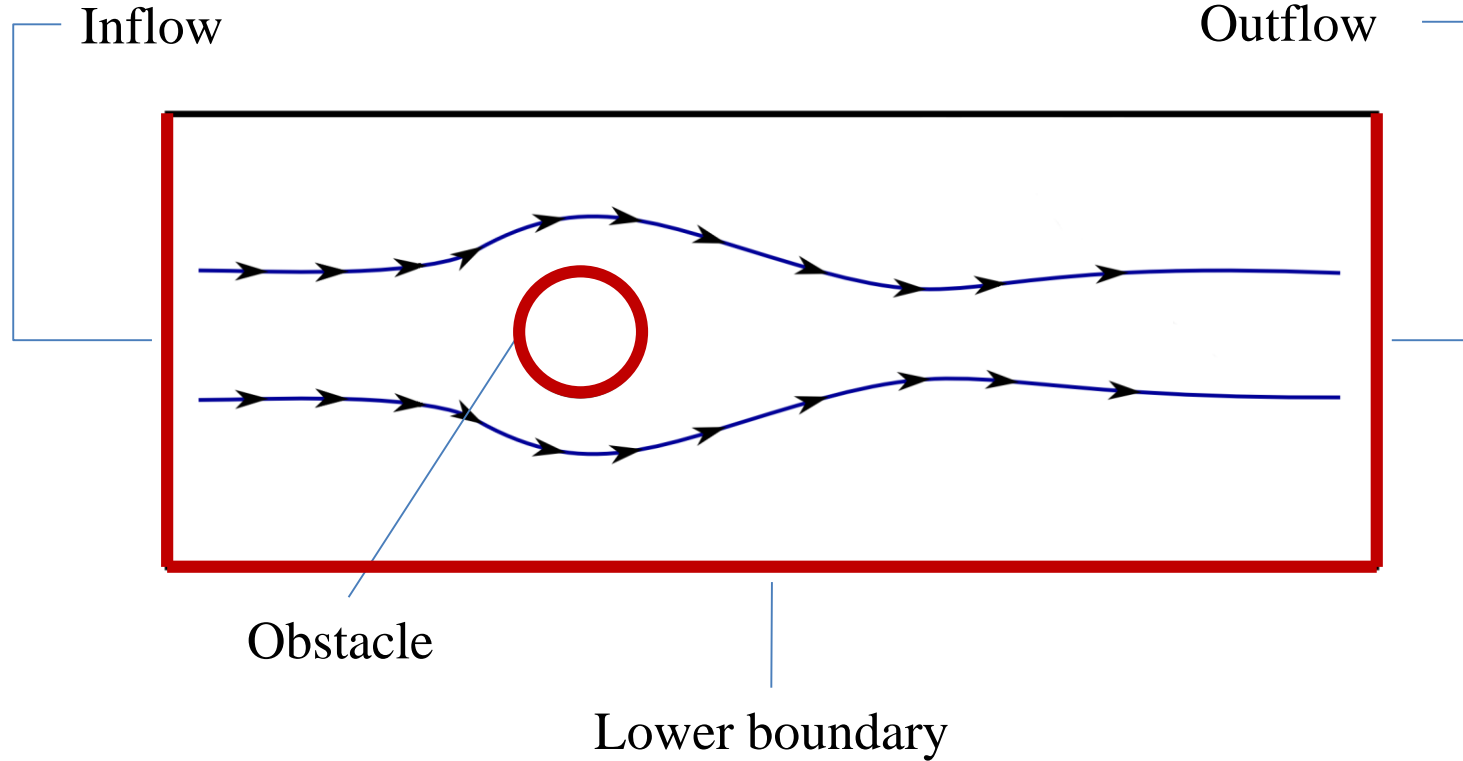


*End of module*  
Streaming Step

*Coming next*  
Boundary conditions

# Boundary conditions

# Boundary conditions



# Boundary conditions

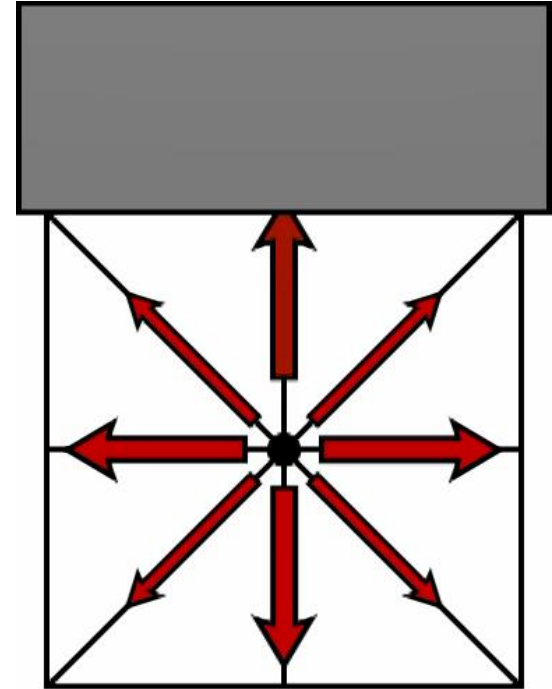
- We need different types of boundary conditions:
  - Inflow: imposes the velocity profile.
  - Outflow, and lower and upper boundary: pretend that the domain is larger.
  - Physical obstacle.
- There exists already a boundary condition in our code: periodicity.
  - Behaves as if our simulation was repeated an infinitely many times.
  - We will use periodicity for lower/upper boundary.

# Obstacle: no-slip wall

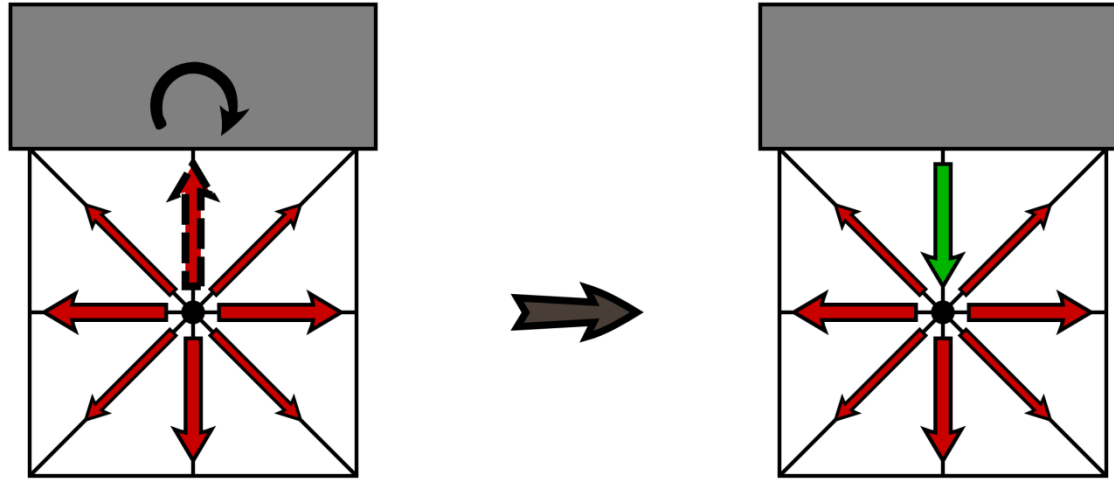
- Most physical obstacles act like no-slip walls.
- Macroscopically: the velocity is zero along the wall.
- At the molecular level: the obstacle has a surface roughness, and molecules «adhere» to the surface.
- Not all obstacles have a no-slip condition. Example: an air bubble in water is smooth, even at molecular level.

# No-slip walls: bounce-back condition

- The bounce-back condition is a way to implement a no-slip wall in lattice Boltzmann.
- During streaming, a population which leaves the fluid and hits an obstacle «bounces back».
- Its value is unchanged, and it is copied to the population with reverse direction.

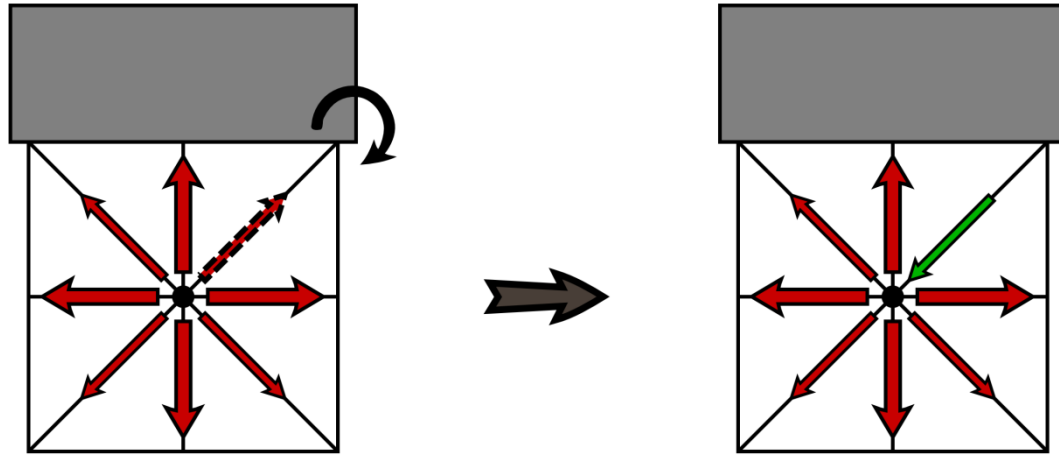


# No-slip walls: bounce-back condition



This condition mimicks the physical process that occurs when molecules hit a wall.

# No-slip walls: bounce-back condition



- Attention: populations on diagonal directions are not reflected like on a mirror. They also bounce-back to where they came from.
- This reflects the fact that at a molecular level, the obstacle is rough, and not smooth like a mirror.

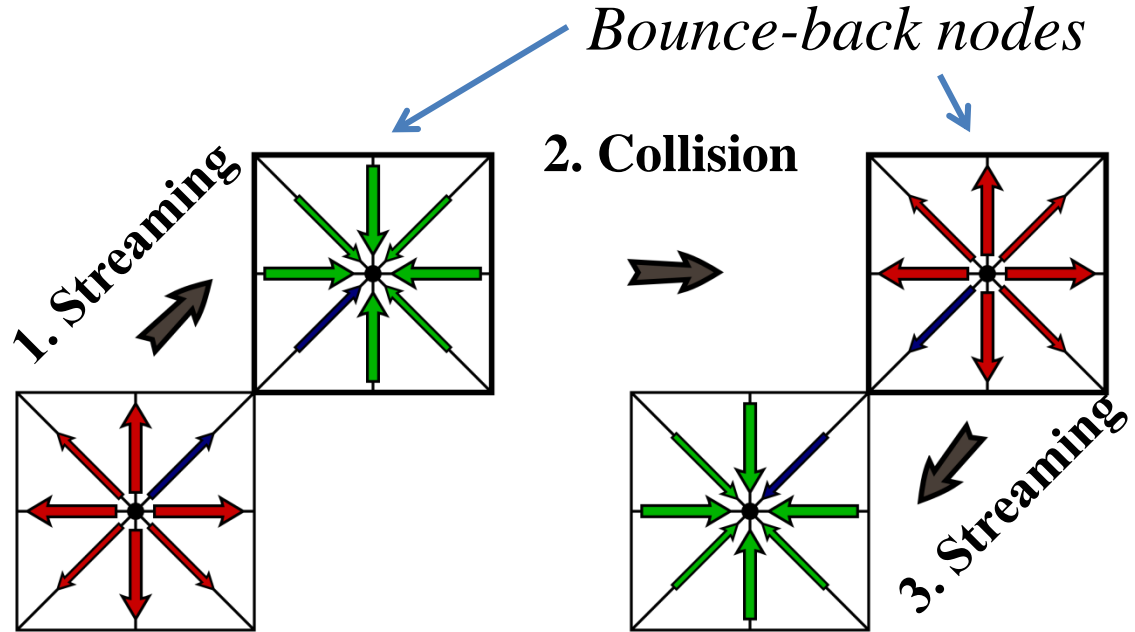


# No-slip walls: bounce-back condition

- Bounce-back is simple: track all populations that hit an obstacle, and send them back.
- Yet, this involves quite some book-keeping. Which nodes are affected? Which directions?
- Simpler solution: let the populations travel into the obstacle. Then, we revert them inside the obstacle, and send them back.
- No bookkeeping of directions. All cells inside an obstacle are bounce-back cells: instead of colliding, they revert all the directions.

# The bounce-back cycle

During collision, a bounce-back node replaces every population by the one in the opposite direction.



# The bounce-back cycle

Instead of colliding, bounce-back nodes implement the following relation:

$$f_i^{\text{in}}(\boldsymbol{x}, t + 1) = f_j^{\text{out}}(\boldsymbol{x}, t)$$

where direction  $i$  is opposite to direction  $j$ :

$$v_i = -v_j$$

# Bounce-back: code

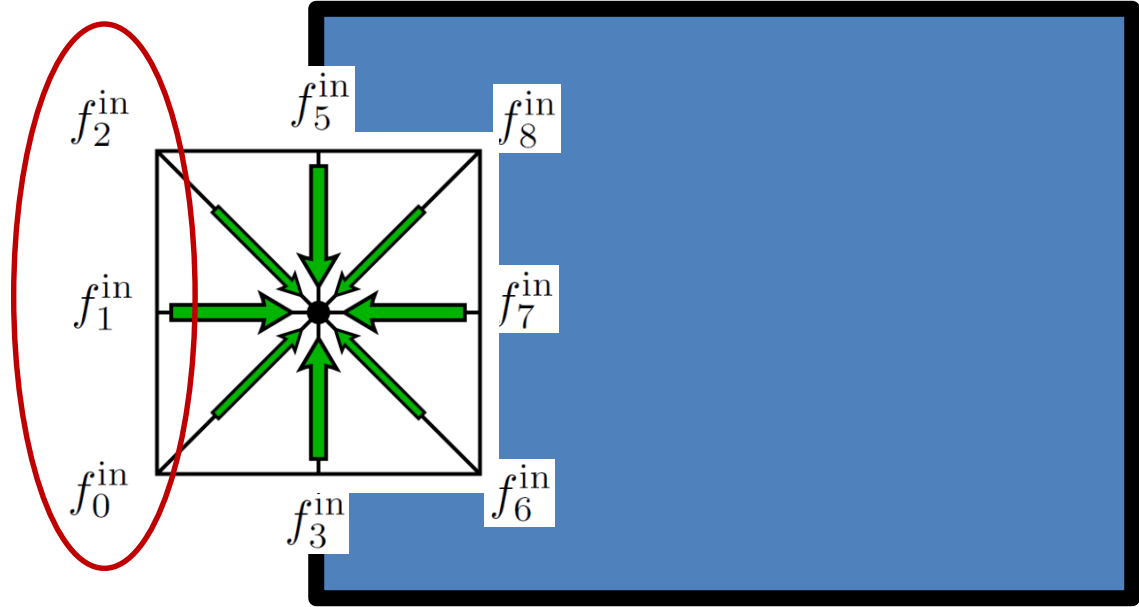
NumPy array-based code:

```
for i in range(9):  
    fout[i, obstacle] = fin[8-i, obstacle]
```

- We have on purpose listed the lattice velocities in such a way that the direction opposite to  $i$  is given by  $8-i$ .

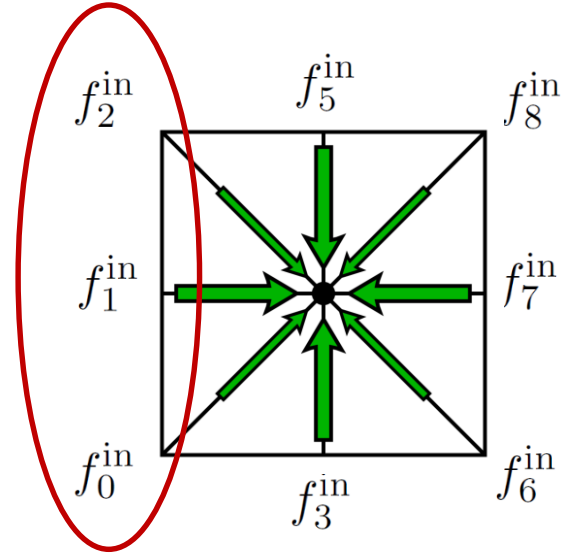
# Inflow condition

After streaming,  
the populations  
 $f_0, f_1$ , and  $f_2$  are  
unknown on all  
cells along the left  
boundary.



# Inflow condition

- We want to impose a velocity  $\mathbf{u}$  on inflow boundary cells. So,  $\mathbf{u}$  is known. But what about  $\rho$ ? And what about  $f_0, f_1$ , and  $f_2$ ?
- We must extract this missing information using the known populations, and using  $\mathbf{u}$ .



# Inflow condition: density

Remember:

$$\rho(\mathbf{x}, t) = \sum_{i=0}^8 f_i^{\text{in}}(\mathbf{x}, t)$$

$$\mathbf{u}(\mathbf{x}, t) = \frac{1}{\rho(\mathbf{x}, t)} \frac{\delta \mathbf{x}}{\delta t} \sum_{i=0}^8 \mathbf{v}_i f_i^{\text{in}}(\mathbf{x}, t)$$

Let's define:

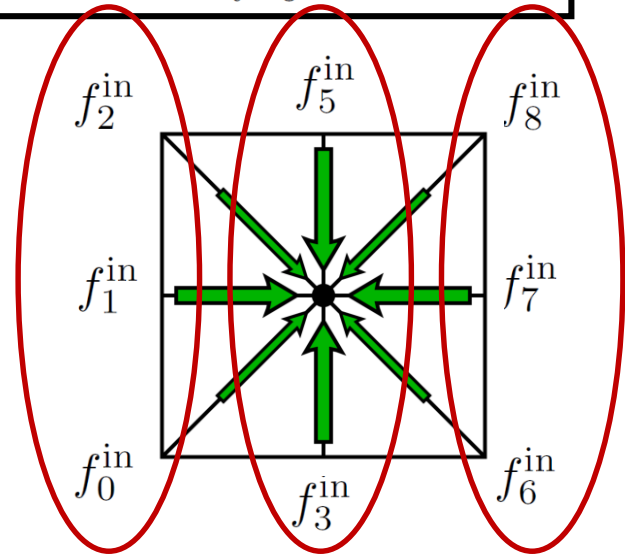
- $\rho_1 = f_0 + f_1 + f_2$  (unknown)
- $\rho_2 = f_3 + f_4 + f_5$  (known)
- $\rho_3 = f_6 + f_7 + f_8$  (known)

Then:

- $\rho = \rho_1 + \rho_2 + \rho_3$
- $\rho u_x = \rho_1 - \rho_3$

And hence:

$$\rho = \frac{\rho_2 + 2\rho_3}{1 - u_x}$$



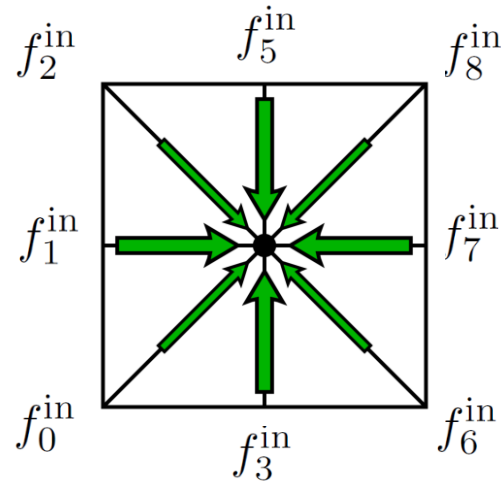
# Inflow condition: unknown populations

- Remember: populations are always close to their equilibrium.
- We first initialize the unknown populations to their equilibrium value.
- Then, we check how much the opposite population deviates from equilibrium, at add this value as a correction.

$$f_0^{\text{in}} = E(0, \rho, \mathbf{u}) + (f_8^{\text{in}} - E(8, \rho, \mathbf{u}))$$

$$f_1^{\text{in}} = E(1, \rho, \mathbf{u}) + (f_7^{\text{in}} - E(7, \rho, \mathbf{u}))$$

$$f_2^{\text{in}} = E(2, \rho, \mathbf{u}) + (f_6^{\text{in}} - E(6, \rho, \mathbf{u}))$$





# Inflow condition: code

- Define the indices of the three columns:

```
col1 = array([0, 1, 2])  
col2 = array([3, 4, 5])  
col3 = array([6, 7, 8])
```

- Calculate the density:

```
rho[0,:] = 1/(1-u[0,0,:]) * ( sum(fin[col2,0:], axis=0) +  
                                2*sum(fin[col3,0:], axis=0) )
```

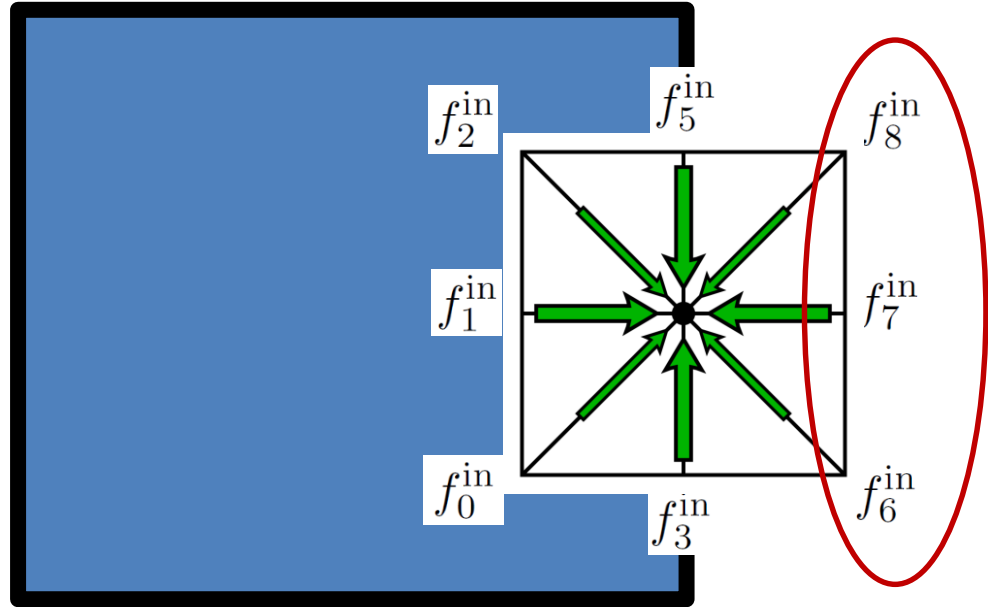
- Calculate the populations:

```
fin[[0,1,2],0,:] = feq[[0,1,2],0,:] + fin[[8,7,6],0,:] - feq[[8,7,6],0,:]
```

# Outflow condition

- The outflow boundary must behave as if the domain didn't end.
- For the unknown populations,  $f_6$ ,  $f_7$ , and  $f_8$ , we simply copy the value from the neighboring cell, right behind.

```
fin[col3,-1,:]=fin[col3,-2,:]
```



*End of module*

Boundary conditions

*Coming next*

Application: flow around an obstacle

Application: flow around an obstacle

# Our code: all pieces put together

```
#!/usr/bin/python3
# Copyright (C) 2015 Universite de Geneve, Switzerland
# E-mail contact: jonas.latt@unige.ch
#
# 2D flow around a cylinder
#

from numpy import *
import matplotlib.pyplot as plt
from matplotlib import cm

##### Flow definition #####
maxIter = 200000 # Total number of time iterations.
Re = 10.0 # Reynolds number.
nx, ny = 420, 180 # Numer of lattice nodes.
ly = ny-1 # Height of the domain in lattice units.
cx, cy, r = nx//4, ny//2, ny//9 # Coordinates of the cylinder.
uLB = 0.04 # Velocity in lattice units.
nub = uLB*r/Re; # Viscosity in lattice units.
omega = 1 / (3*nub+0.5); # Relaxation parameter.

##### Lattice Constants #####
v = array([ [ 1, 1], [ 1, 0], [ 1, -1], [ 0, 1], [ 0, 0],
            [ 0, -1], [-1, 1], [-1, 0], [-1, -1] ])
t = array([ 1/36, 1/9, 1/36, 1/9, 4/9, 1/9, 1/36, 1/9, 1/36])

col1 = array([0, 1, 2])
col2 = array([3, 4, 5])
col3 = array([6, 7, 8])

##### Function Definitions #####
def macroscopic(fin):
    rho = sum(fin, axis=0)
    u = zeros((2, nx, ny))
    for i in range(9):
        u[0,:,i] += v[i,0] * fin[i,:,i]
        u[1,:,i] += v[i,1] * fin[i,:,i]
    u /= rho
    return rho, u

def equilibrium(rho, u):
    # Equilibrium distribution function.
    usqr = 3/2 * (u[0]**2 + u[1]**2)
    feq = zeros((9,nx,ny))
    for i in range(9):
        cu = 3 * (v[i,0]*u[0,:,i] + v[i,1]*u[1,:,i])
        feq[i,:,i] = rho*t[i] * (1 + cu + 0.5*cu**2 - usqr)
    return feq
```

```
##### Setup: cylindrical obstacle and velocity inlet with perturbation #####
# Creation of a mask with 1/0 values, defining the shape of the obstacle.
def obstacle_fun(x, y):
    return (x-cx)**2+(y-cy)**2<r**2

obstacle = fromfunction(obstacle_fun, (nx,ny))

# Initial velocity profile: almost zero, with a slight perturbation to trigger
# the instability.
def inivel(d, x, y):
    return (1-d) * uLB * (1 + 1e-4*sin(y/ly*2*pi))

vel = fromfunction(inivel, (2,nx,ny))

# Initialization of the populations at equilibrium with the given velocity.
fin = equilibrium(1, vel)

##### Main time loop #####
for time in range(maxIter):
    # Right wall: outflow condition.
    fin[col3,-1,:] = fin[col3,-2,:]

    # Compute macroscopic variables, density and velocity.
    rho, u = macroscopic(fin)

    # Left wall: inflow condition.
    u[:,0,:] = vel[:,0,:] * ( sum(fin[col2,0,:], axis=0) +
                               2*sum(fin[col3,0,:], axis=0) )

    # Compute equilibrium.
    feq = equilibrium(rho, u)
    fin[0,1,2,0,:] = feq[0,1,2,0,:] + fin[[8,7,6],0,:] - feq[[8,7,6],0,:]

    # Collision step.
    fout = fin - omega * (fin - feq)

    # Bounce-back condition for obstacle.
    for i in range(9):
        fout[i, obstacle] = fin[8-i, obstacle]

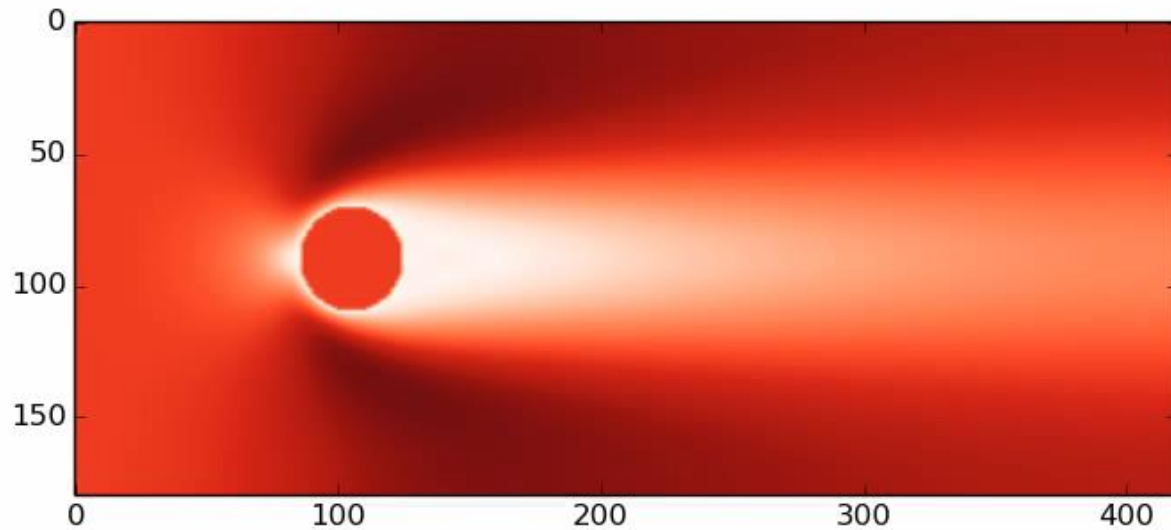
    # Streaming step.
    for i in range(9):
        fin[i,:,i] = roll(
            roll(fout[i,:,i], v[i,0], axis=0),
            v[i,1], axis=1)

    # Visualization of the velocity.
    if (time%100==0):
        plt.clf()
        plt.imshow(sqrt(u[0]**2+u[1]**2).transpose(), cmap=cm.Reds)
        plt.savefig("vel{:03d}.png".format(time//100))
```

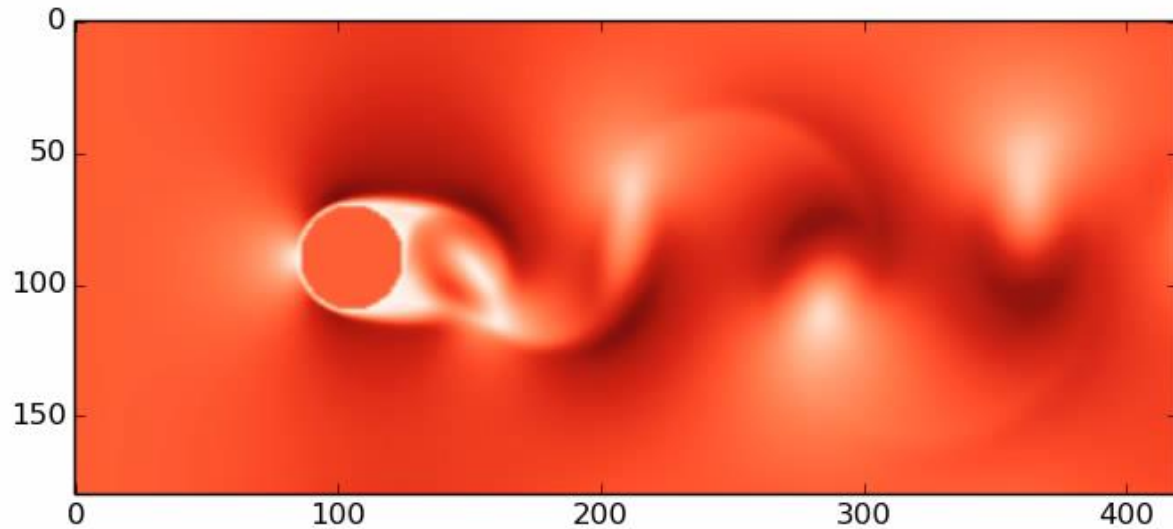
# In detail: simulation parameters

```
nx, ny = 420, 180 # Number of cells  
r = ny//9        # Radius of obstacle  
uLB = 0.04       # Velocity at inlet  
Re = 220.0       # Reynolds number  
nulb = uLB*r/Re # Viscosity  
omega = 1 / (3*nulb+0.5) # Relax. parameter
```

# Result: $Re=10$



# Result: $Re=220$





*End of module*

Application: flow around an obstacle

*End of week 5*