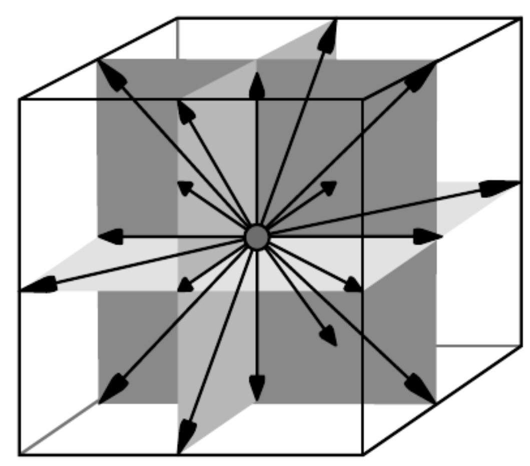
Computational Fluid Dynamics Simulations Using the Lattice Boltzmann Method

Computer Systems Lab 2009-2010

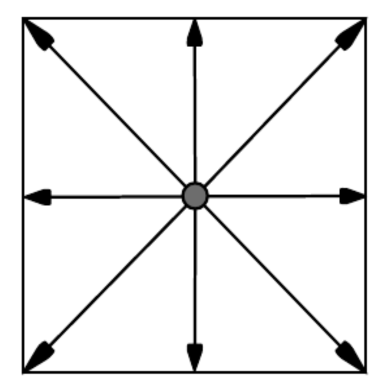
Thomas Georgiou

Abstract

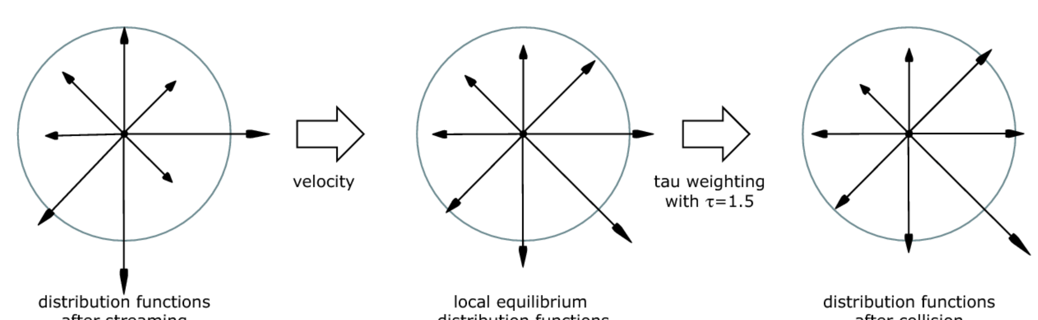
Fluid simulations are useful in many different areas ranging from weather modeling to microscopic physics. Using the conventional method of solving the districtized Navier-Stokes equations is very computationally intensive and relatively hard to parallelize. The lattice boltzmann method instead uses the discrete Boltzmann equation to simulate Newtonian fluids using various collision models.



D3Q19

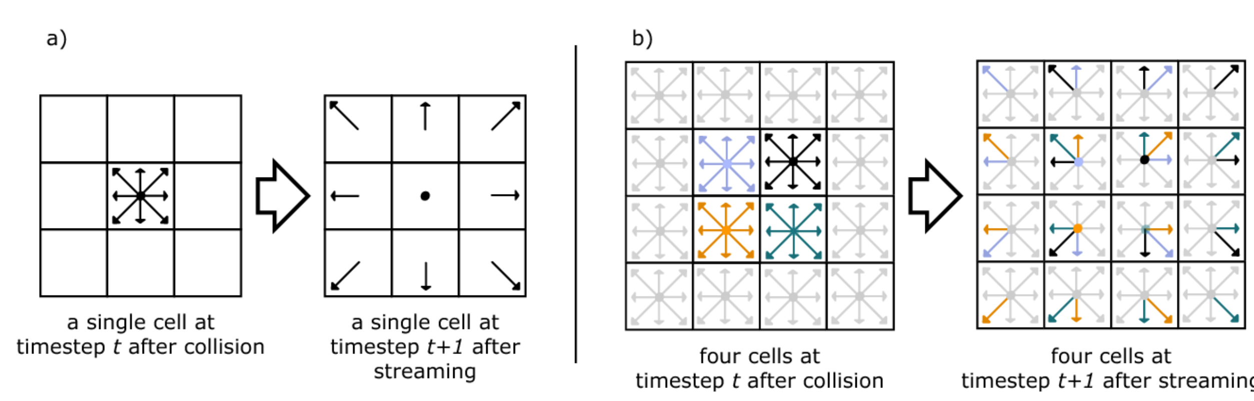


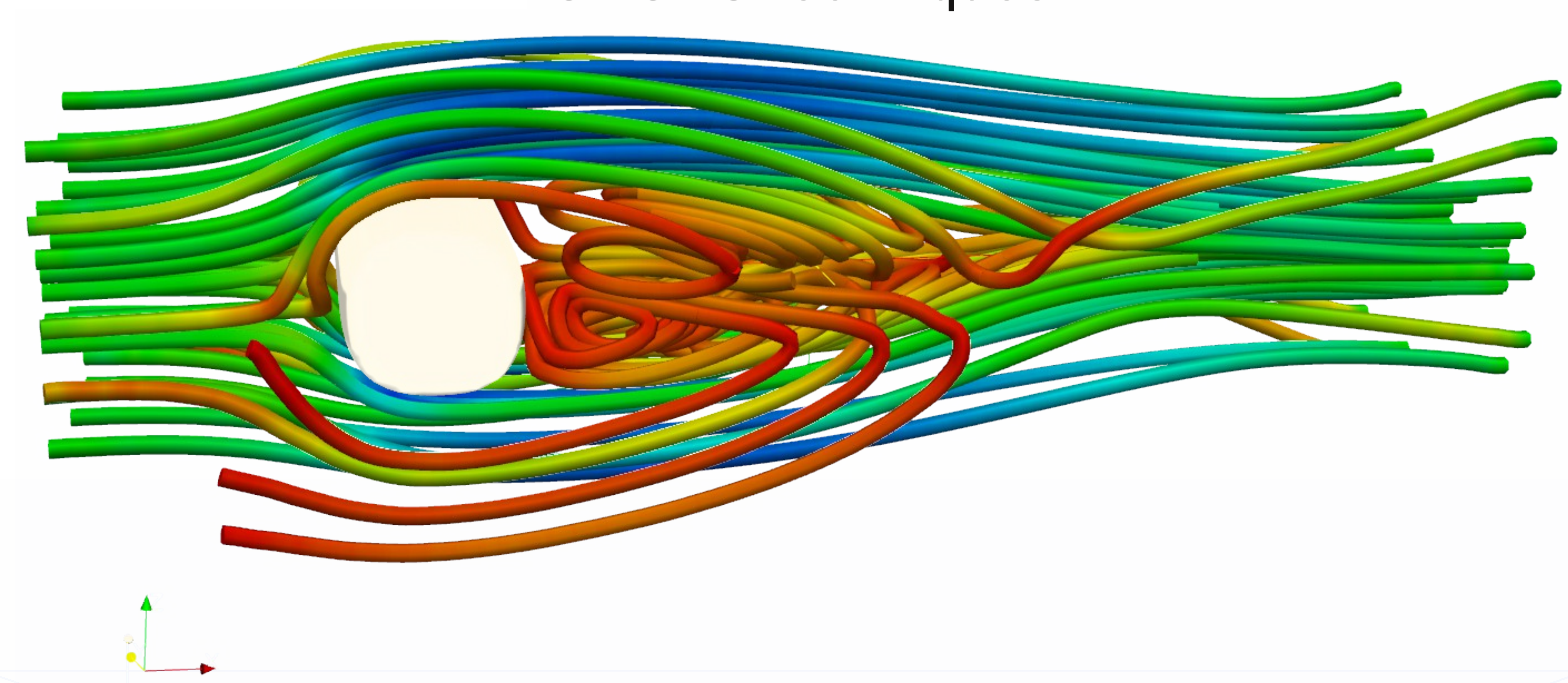
D2Q9

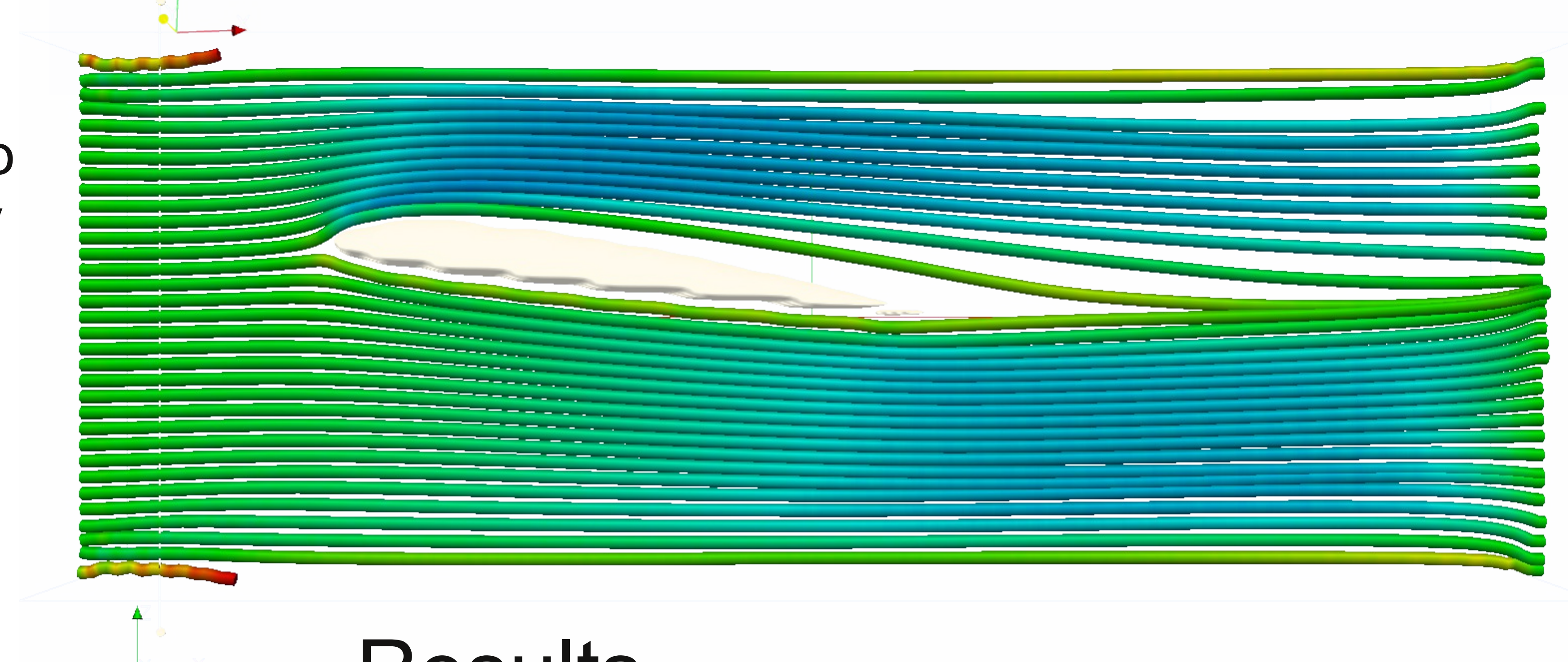


Introduction

Fluid dynamics are useful in a broad range of fields including meteorology, computer graphics, aerodynamics, and microscopic physics. The purpose of this project is to accelerate relatively new methods in the field of computational fluid dynamics in order to be able to run realtime simulations. This includes using new methods that can be parallelized more effectively and vectorizing these methods and running them on new hardware using GPGPU techniques.

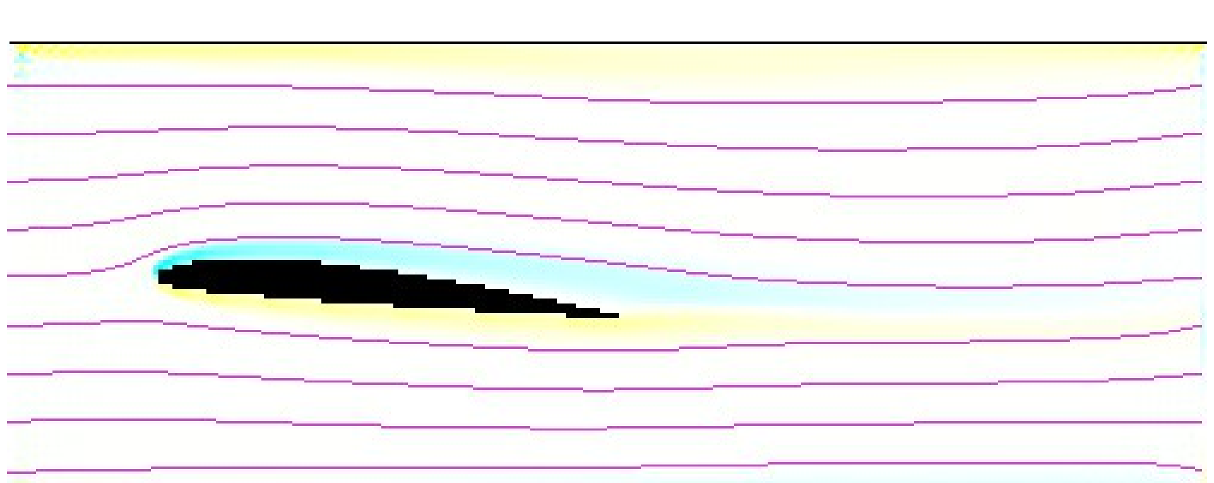


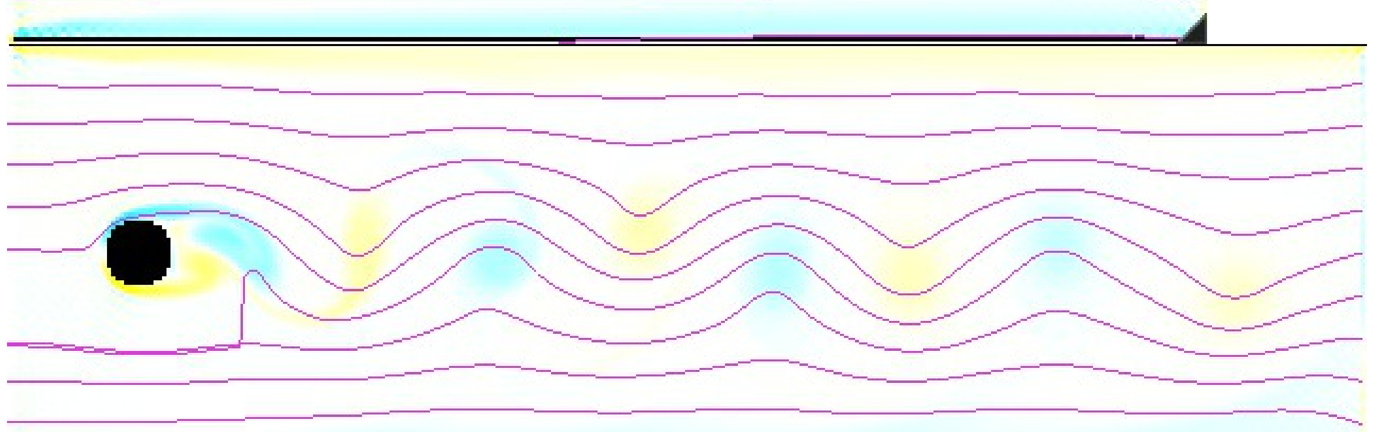




Implementation Details

Currently, my simulation is a D3Q19 simulation using the BGK collision operator. A grid is used with 19 velocities at each point. At every time step, fluid moving in a given velocity is advected to the neighboring node its velocity is pointing too and then inter-particle collisions are simulated by assuming collisions tend the system towards equilibrium. It is programmed in C as this is a very performance intensive project. OpenMP is used for intra-node parallelism and MPI is used for inter-node parallelism. Simulation results are exported in the 3D VTK file format and then imported into Paraview, an open source visualization utility used used in large supercomputing centers, for visualization.





Results

Current results consist of a simulation of a three dimensional fluid that can be conducted in realtime on a single processor on a 300x300 grid. An MPI parallel version runs much faster and a version in CUDA even faster still by an order of magnitude.

Physically, the simulation visually appears to model correct fluid dynamics behavior. Lid driven cavity simulations mimic real world results as evidenced by youtube videos of published experiments using water.

Wind tunnel tests can be performed to calculate lift and drag on arbitrary objects and airfoils.