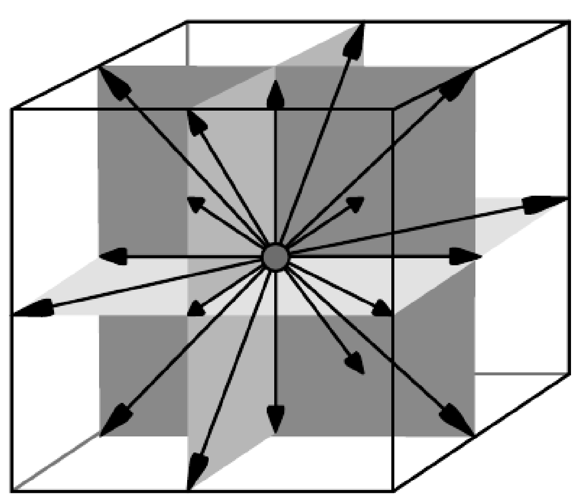
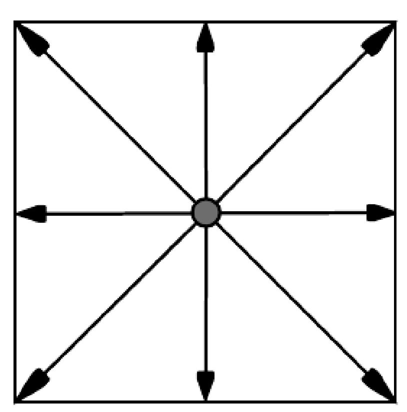
Realtime Computational Fluid Dynamics Simulations Using the Lattice Boltzmann Method

Abstract

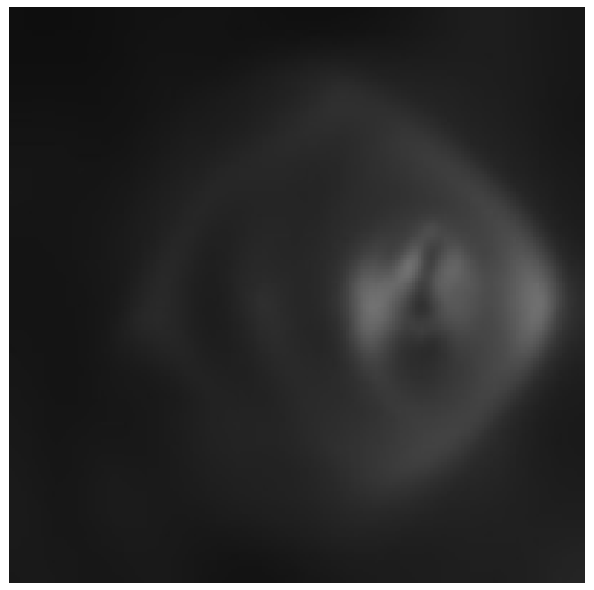
Fluid simulations are useful in many different areas ranging from weather modeling to microscopic physics. Using the conventional method of solving the districtized Navier-Stokes equations is very computationally intensive and relatively hard to parallelize. The lattice boltzmann method instead uses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

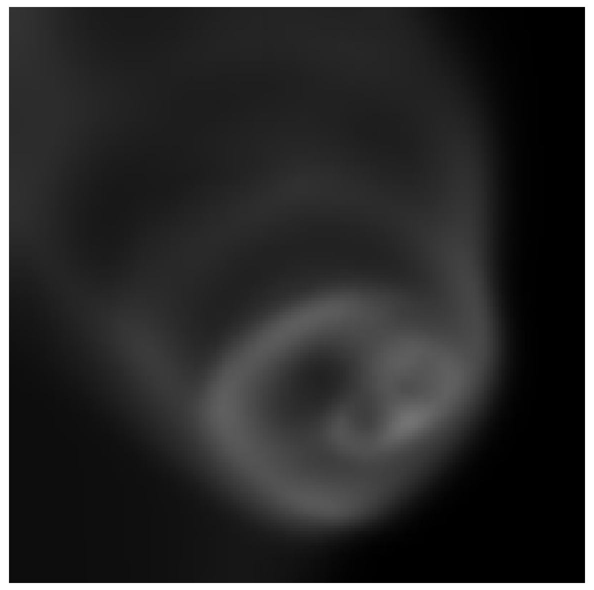


D3Q19



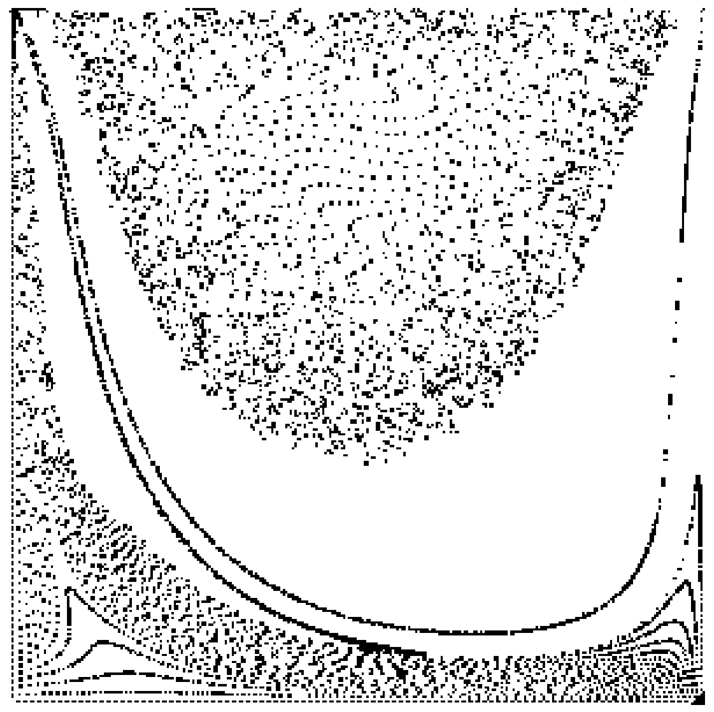
D2Q9

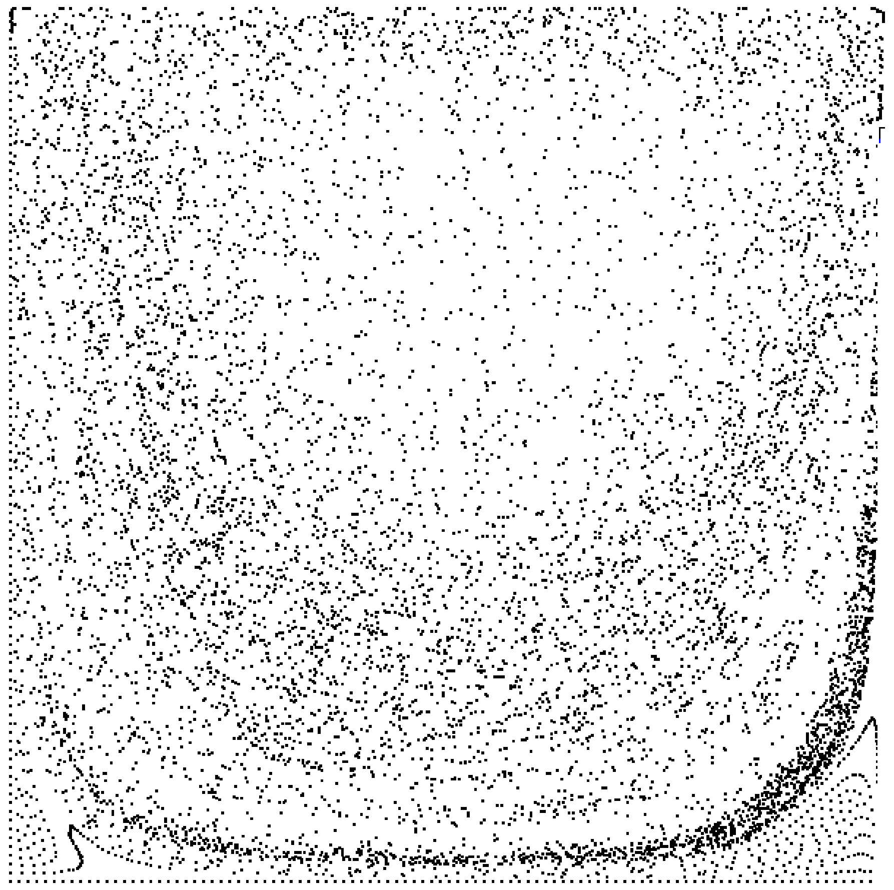




Introduction

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.



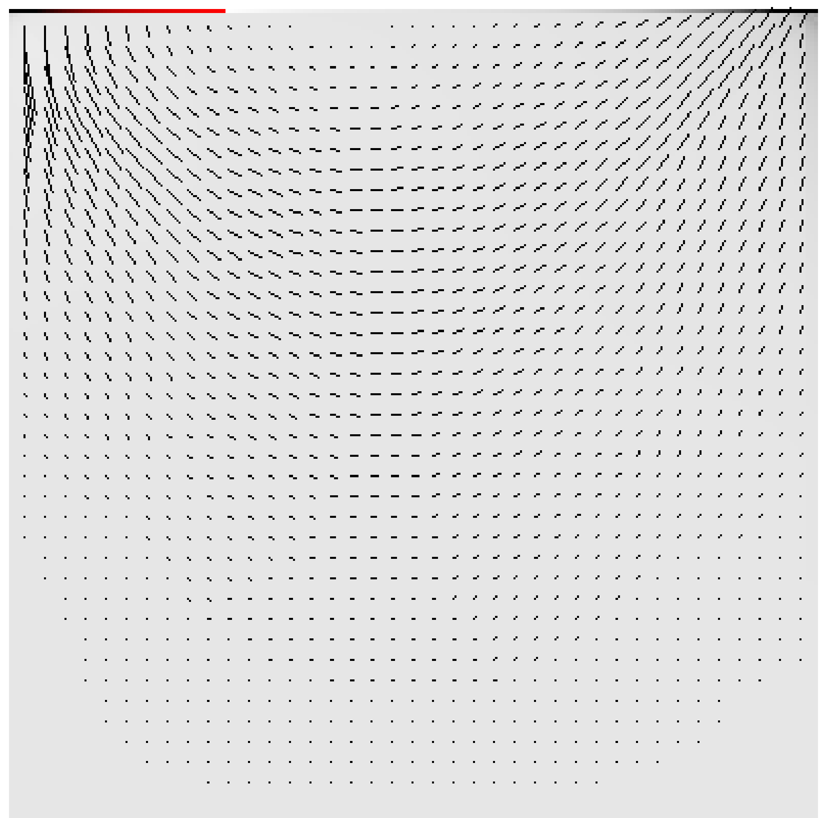


Implementation Details

Currently, my simulation is a D2Q9 simulation using the BGK collision operator. It is programmed in the C programming language as this is a very performance intensive project and I am more comfortable in C than in Fortran. OpenGL is used to provide visualization display and input. For visualization, a grayscale image is presented with each pixel taking on the value proportional to the amount of fluid present at a lattice point. Mouse presses currently add stationary fluid at the pointer location. OpenMP is used for intra-node parallelism and MPI is used for inter-node parallelism.

Visualization

Tracer particles are placed in the fluid and advected using Euler's method.
Vorticity is also displayed.



Results

Current results consist of a simulation of a two dimensional fluid that can be conducted in realtime on a single processor on a 300x300 grid. The simulation is currently unoptimized since I copy the memory at each timestep, which while good for simplicity and getting a simulation up and running, is terrible for performance.
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.