Rayleigh-Bénard Convection: Drekar Simulation on Odyssey

Harvard IACS AC 290R

Problem Statement & Motivation

- Rayleigh-Bénard Convection (RBC) arises when a fluid has a temperature gradient in a gravitational field
- It is an important physical phenomenon arising in stellar evolution, plate tectonics, and weather systems
- We ran a numerical simulation of RBC based on the Boussinesq approximation, in which we assume that fluids develop a buoyancy force that is linear in the temperature
- The simulation was run on the Harvard Odyssey computing cluster using the Drekar simulation code developed at Sandia National Lab

Description of Code

- Drekar is a large scale computational fluid dynamics (CFD) code developed at Sandia National Lab
- Drekar solves fluid PDEs using the Finite Element Method (FEM)
- Core functionality is a massively parallel FEM implementation that runs on MPI with back ends from threads, OpenMP and CUDA
- Drekar is part of the Trilinos package, which provides functionality including linear algebra, nonlinear solvers, automatic differentiation, and time integration

Overview of Numerical Methods Used

- The Finite Element Method (FEM) solves PDEs by discretizing them in space on a *mesh*.
- The functions to be solved for (here pressure, temperature, and velocity) are represented by choosing a finite dimensional basis
- This simulation used piecewise polynomials of order 1, i.e. rectilinear planar functions (analog to piecewise linear in 2D)
- The time stepper discretizes in time; each time step leads to a set of linear equations to be solved and then a nonlinear solution

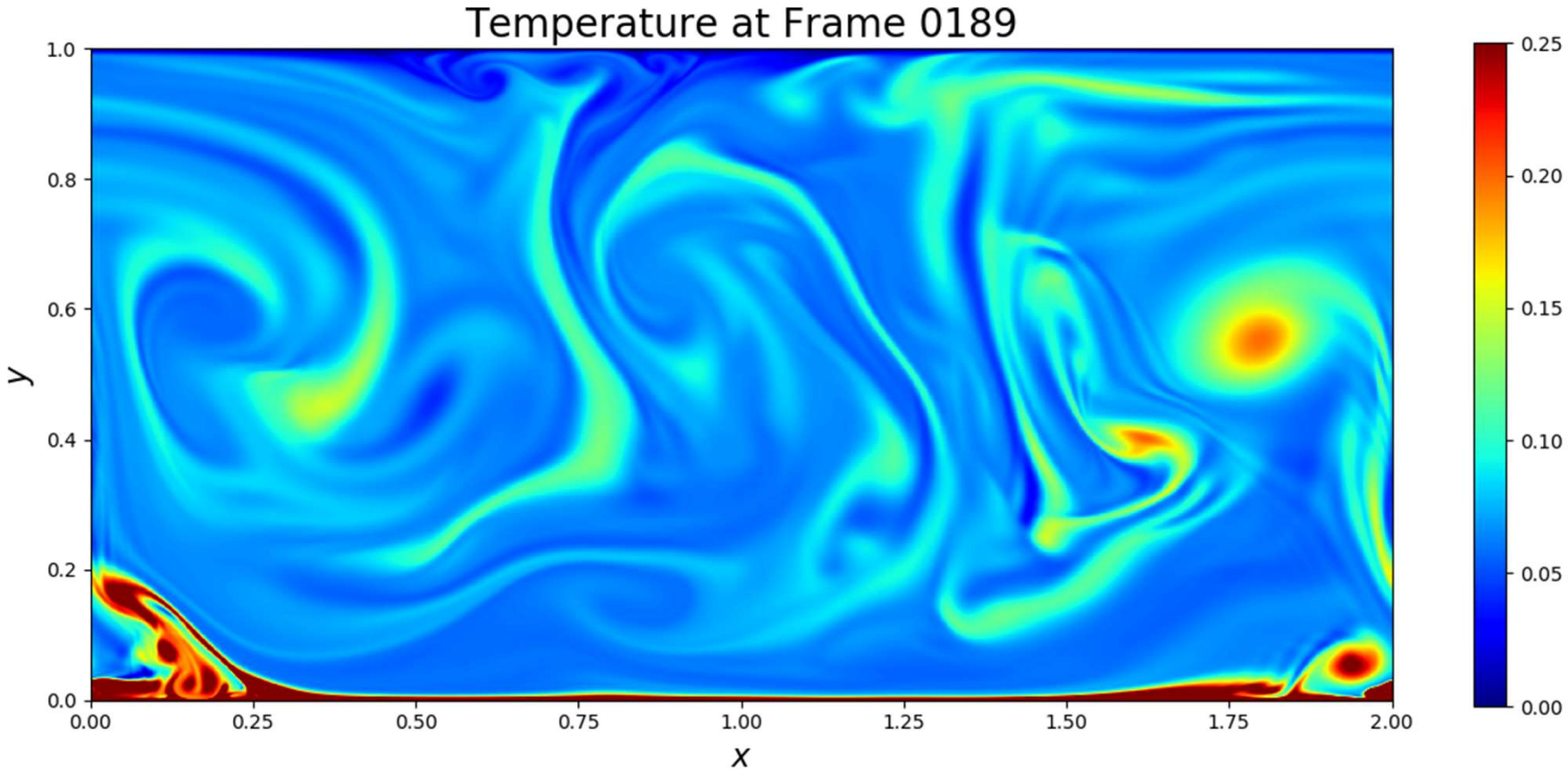
Parameters of the Simulation

- Dimensions: Length L=2.0, Height =1.0, $\Gamma = 2$; $N_x = 1024$, $N_y = 2048$
- Density $\rho_0 = 1.0$
- Viscosity $\nu = 0.01$
- Volume Expansion Coefficient $\alpha_V = 10^6$
- Gravity $g = 1.0$
- Heat Capacity $C_p = 1.0$
- Thermal Conductivity $k = 0.01$
- Thermal diffusivity $\kappa = k / \rho_0 C_p = 0.01$
- Temperature Change $\Delta T = 1.0$
- Time of Simulation: 5.0 target (0.22 achieved on case 1)

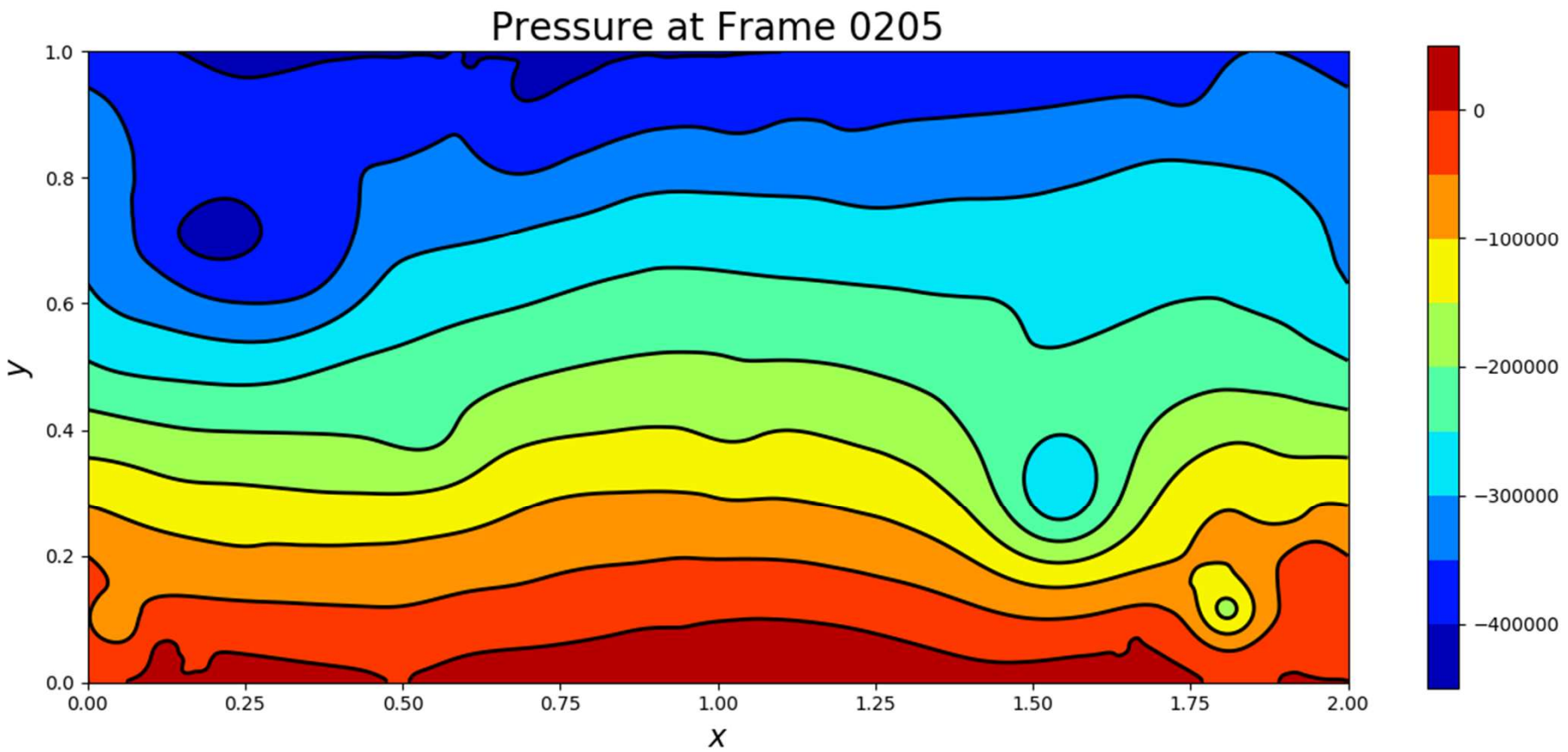
Results

- Case 1 was initiated on 1024 cores but only ran on 512 cores due to a hardware fault.
- It ran for about 1 day and got to time 0.22 before we restarted it.
- Case 2 ran on all 1024 cores for about 5 days. It bogged down, getting progressively slower, and eventually crashed (out of memory)
- It only got to a simulation time of 0.77; it should have gotten much further. After it crashed, out .exo output files became corrupted.
- We suspect a memory leak
- We present visualizations from the successful case 1 simulation

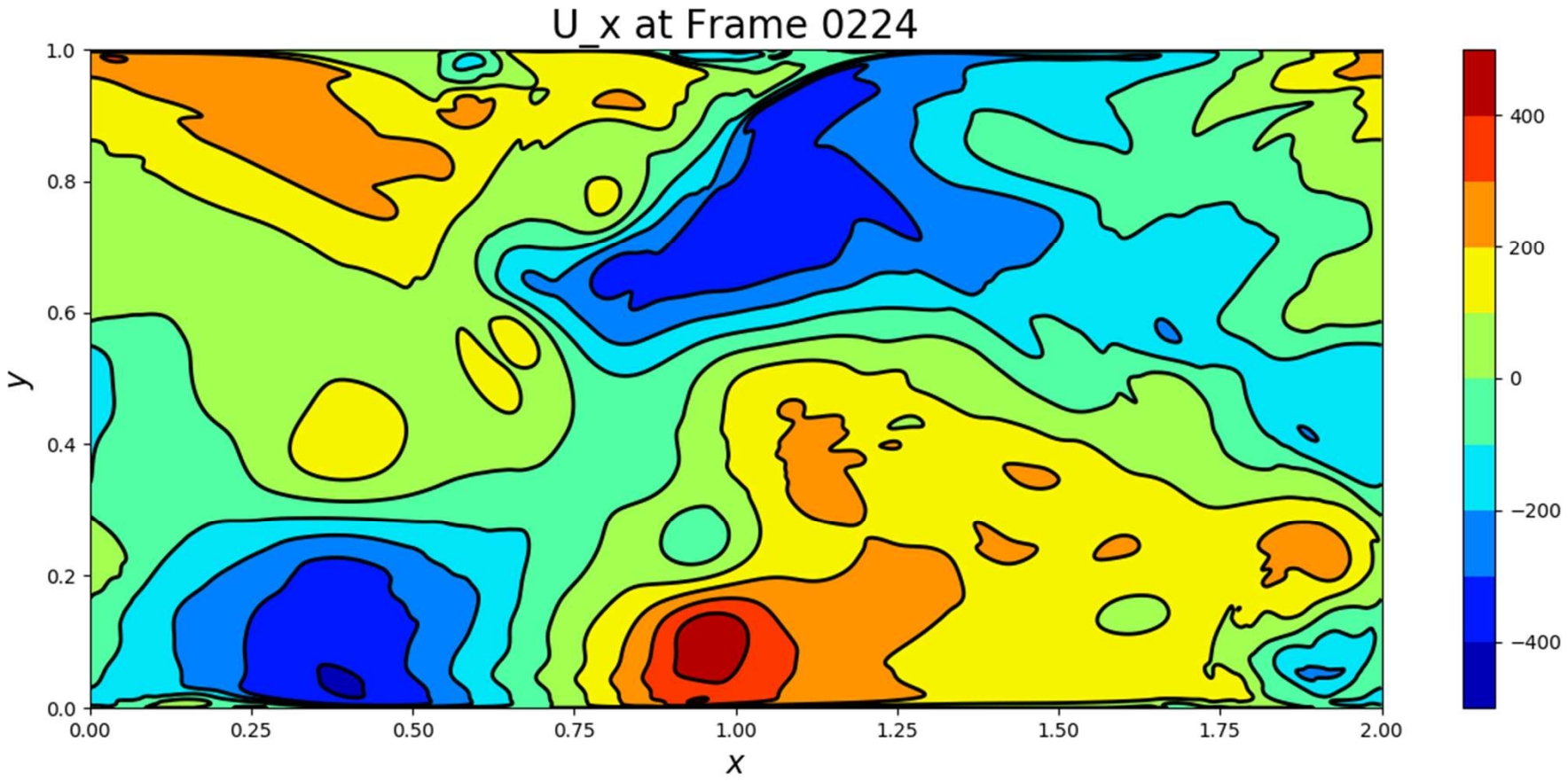
Sample Output: Temperature



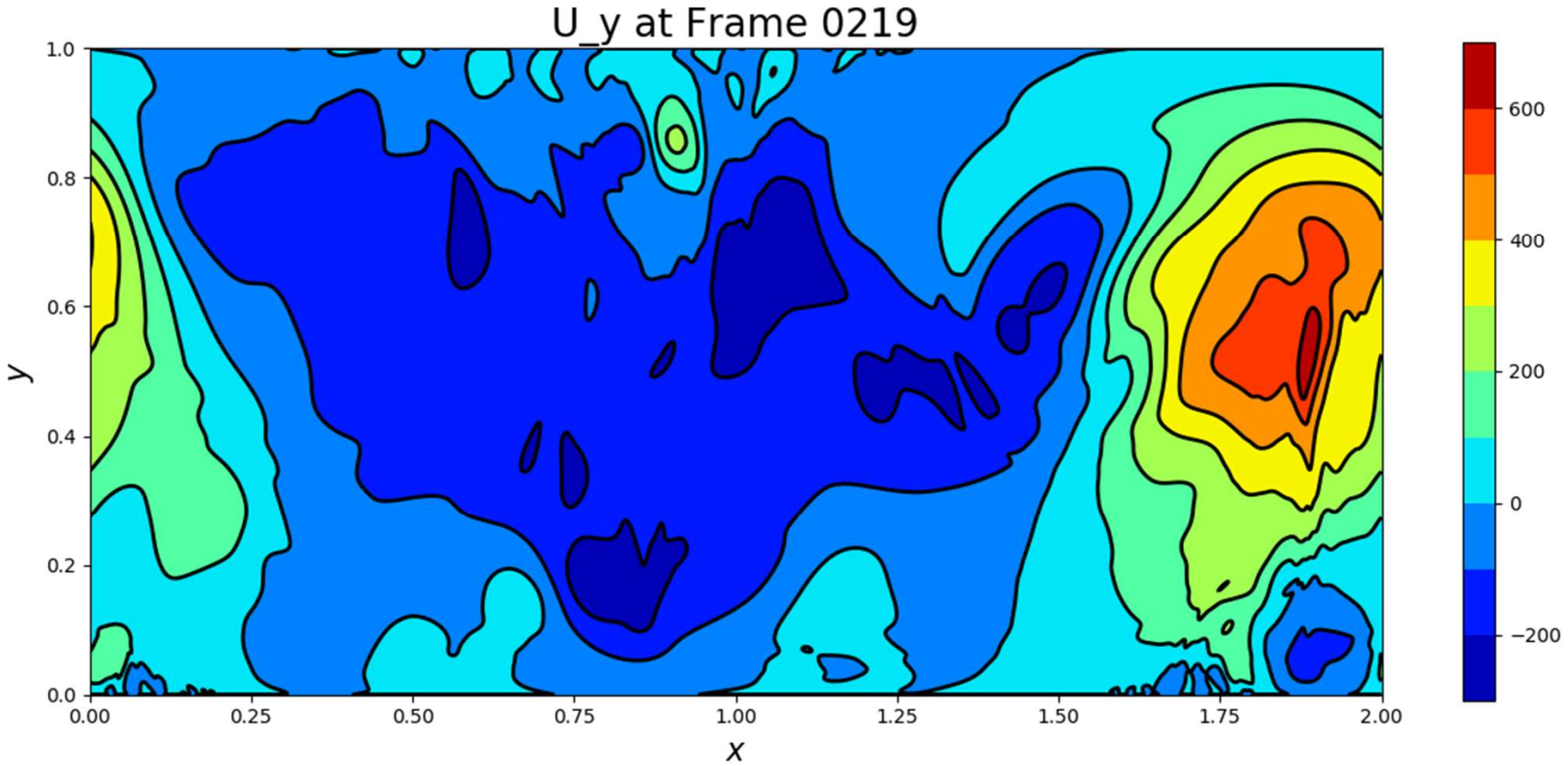
Sample Output: Pressure



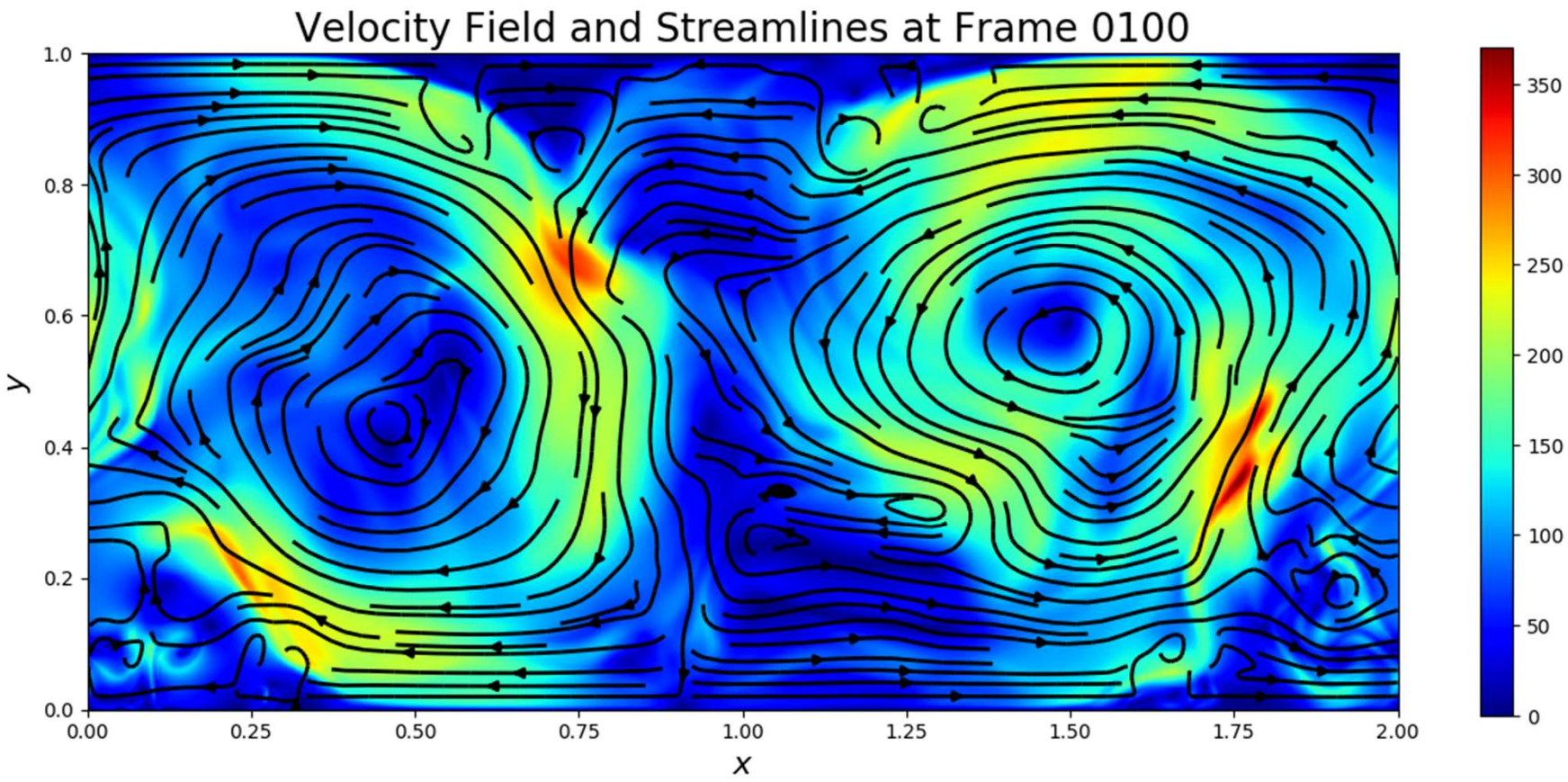
Sample Output: Velocity in x Direction



Sample Output: Velocity in y Direction



Sample Output: Velocity Field & Streamlines



Rayleigh-Bénard Convection: Drekar Simulation on Odyssey

Harvard IACS AC 290R

Problem Statement & Motivation

- Rayleigh-Bénard Convection (RBC) arises when a fluid has a temperature gradient in a gravitational field
- It is an important physical phenomenon arising in stellar evolution, plate tectonics, and weather systems
- We ran a numerical simulation of RBC based on the Boussinesq approximation, in which we assume that fluids develop a buoyancy force that is linear in the temperature
- The simulation was run on the Harvard Odyssey computing cluster using the Drekar simulation code developed at Sandia National Lab

Description of Code

- Drekar is a large scale computational fluid dynamics (CFD) code developed at Sandia National Lab
- Drekar solves fluid PDEs using the Finite Element Method (FEM)
- Core functionality is a massively parallel FEM implementation that runs on MPI with back ends from threads, OpenMP and CUDA
- Drekar is part of the Trilinos package, which provides functionality including linear algebra, nonlinear solvers, automatic differentiation, and time integration

Overview of Numerical Methods Used

- The Finite Element Method (FEM) solves PDEs by discretizing them in space on a *mesh*.
- The functions to be solved for (here pressure, temperature, and velocity) are represented by choosing a finite dimensional basis
- This simulation used piecewise polynomials of order 1, i.e. rectilinear planar functions (analog to piecewise linear in 2D)
- The time stepper discretizes in time; each time step leads to a set of linear equations to be solved and then a nonlinear solution

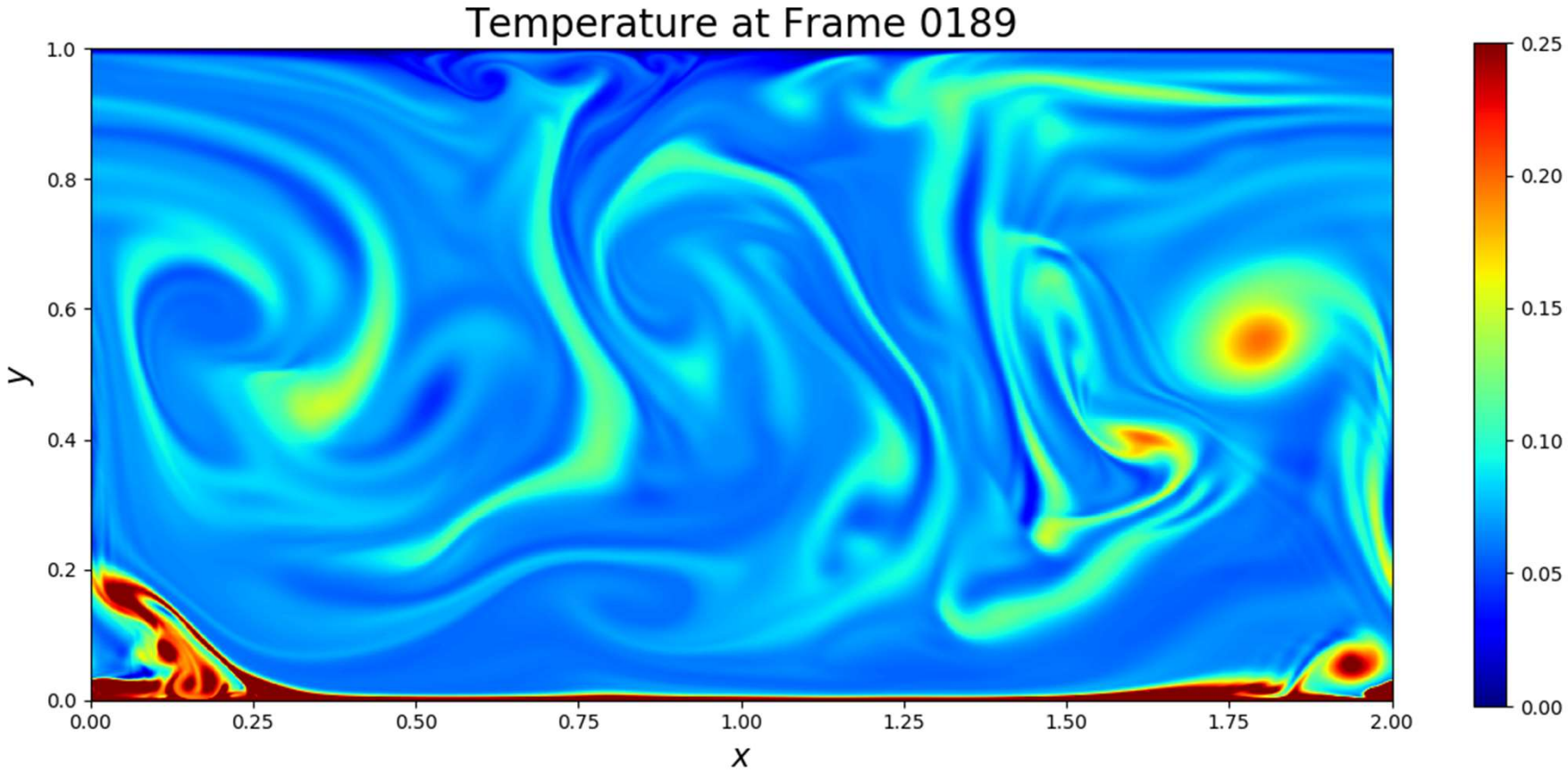
Parameters of the Simulation

- Dimensions: Length L=2.0, Height =1.0, $\Gamma = 2$; $N_x = 1024$, $N_y = 2048$
- Density $\rho_0 = 1.0$
- Viscosity $\nu = 0.01$
- Volume Expansion Coefficient $\alpha_V = 10^6$
- Gravity $g = 1.0$
- Heat Capacity $C_p = 1.0$
- Thermal Conductivity $k = 0.01$
- Thermal diffusivity $\kappa = k / \rho_0 C_p = 0.01$
- Temperature Change $\Delta T = 1.0$
- Time of Simulation: 5.0 target (0.22 achieved on case 1)

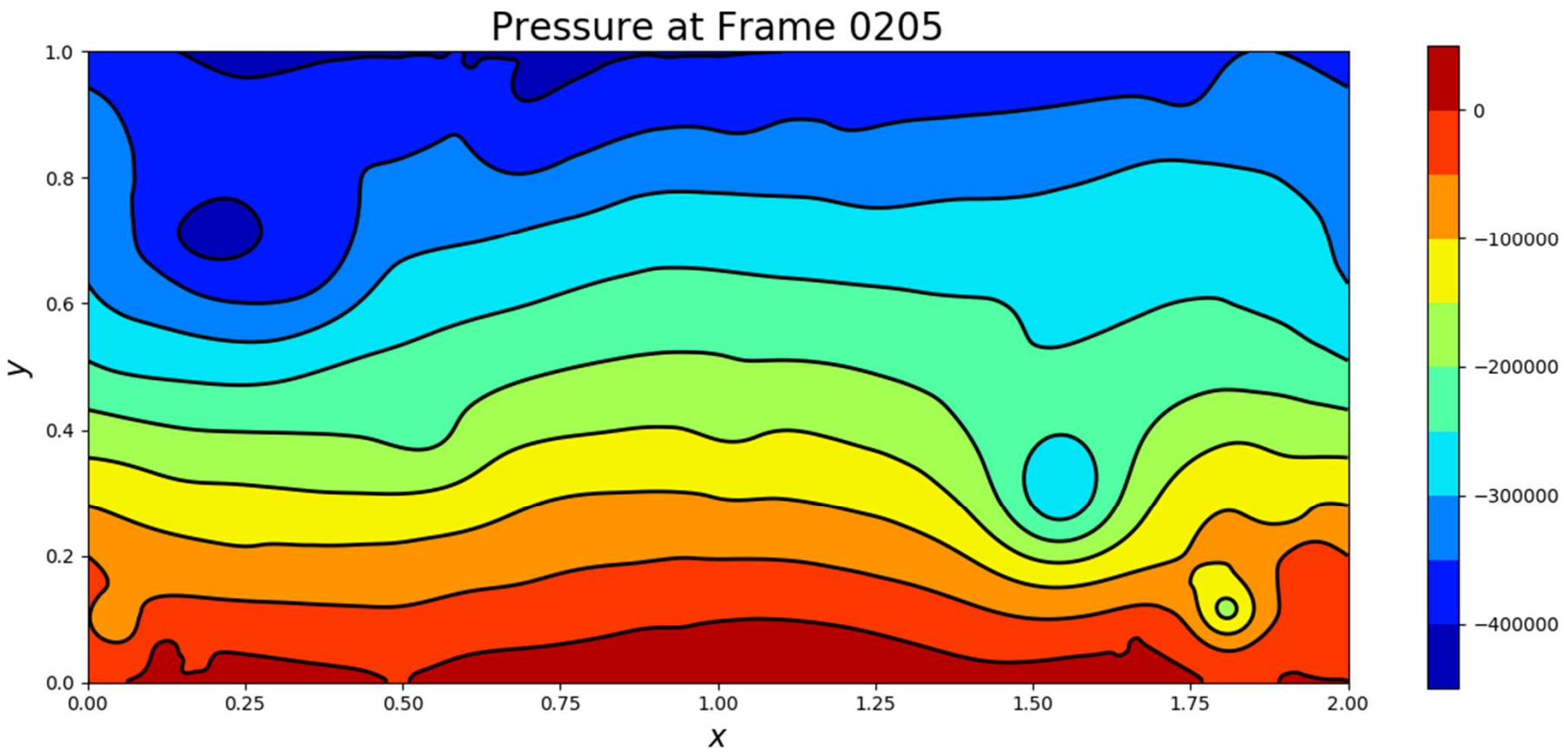
Results

- Case 1 was initiated on 1024 cores but only ran on 512 cores due to a hardware fault.
- It ran for about 1 day and got to time 0.22 before we restarted it.
- Case 2 ran on all 1024 cores for about 5 days. It bogged down, getting progressively slower, and eventually crashed (out of memory)
- It only got to a simulation time of 0.77; it should have gotten much further. After it crashed, out .exo output files became corrupted.
- We suspect a memory leak
- We present visualizations from the successful case 1 simulation

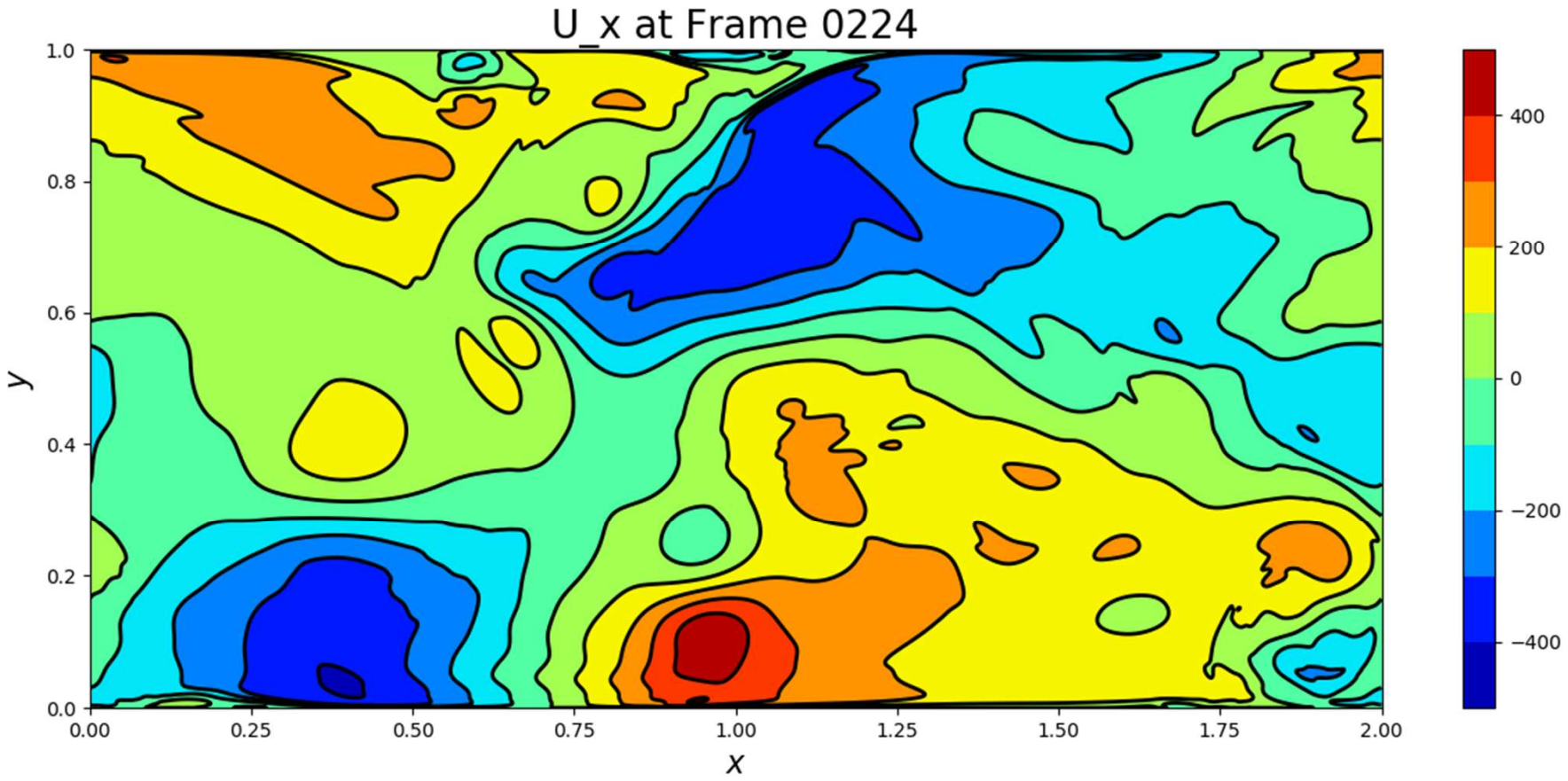
Sample Output: Temperature



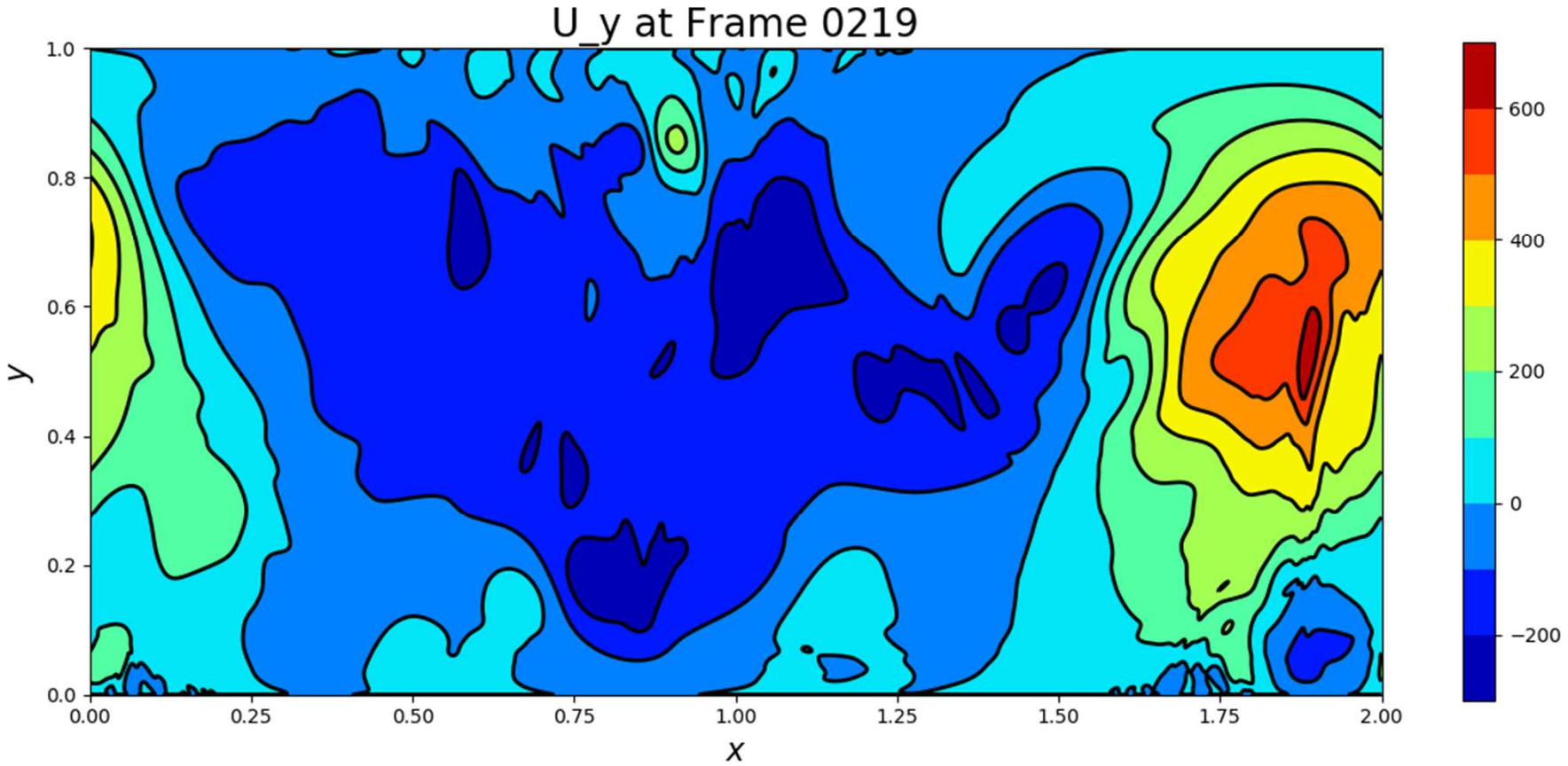
Sample Output: Pressure



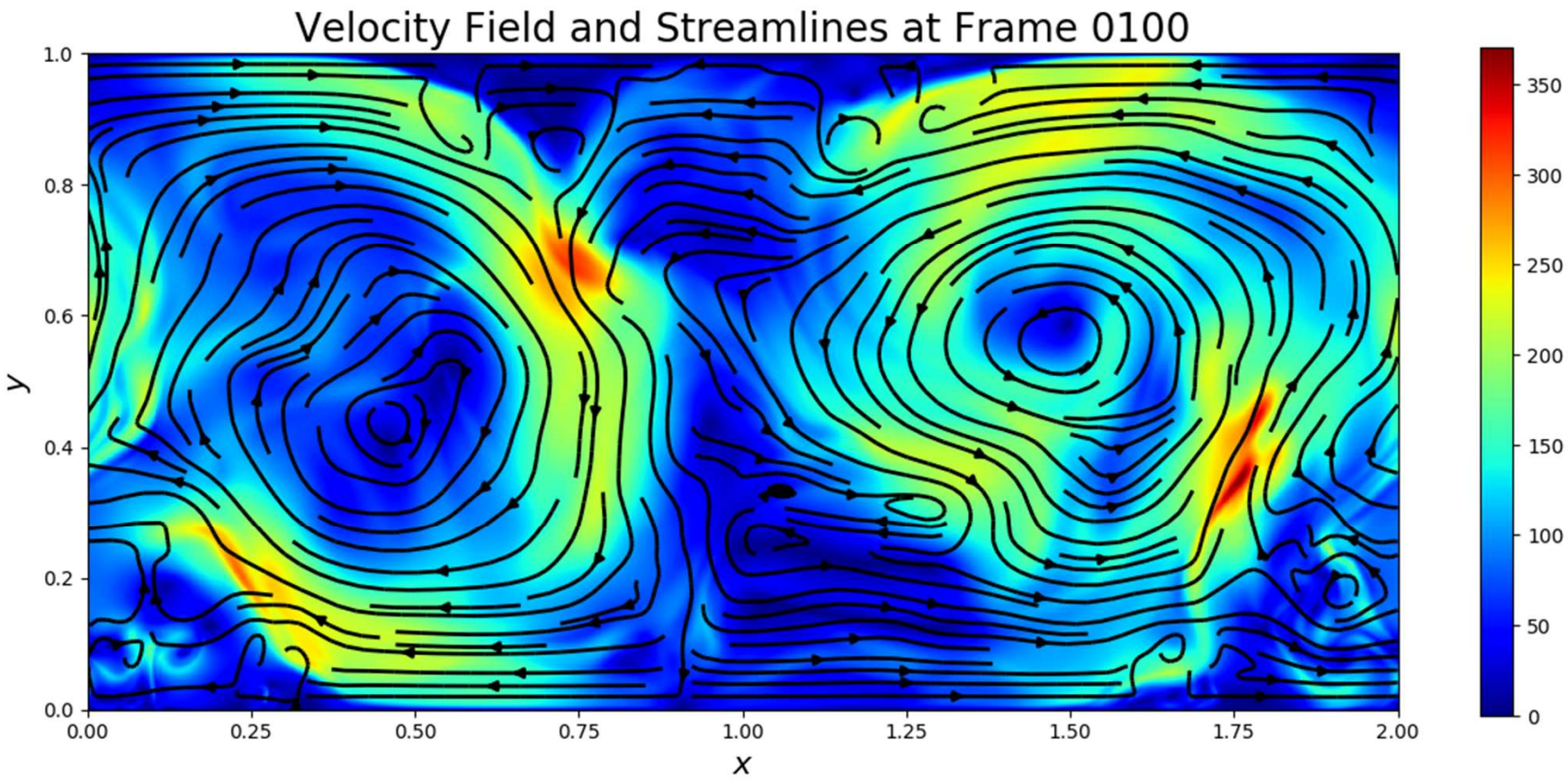
Sample Output: Velocity in x Direction



Sample Output: Velocity in y Direction



Sample Output: Velocity Field & Streamlines



Rayleigh-Bénard Convection: Drekar Simulation on Odyssey

Harvard IACS AC 290R

Problem Statement & Motivation

- Rayleigh-Bénard Convection (RBC) arises when a fluid has a temperature gradient in a gravitational field
- It is an important physical phenomenon arising in stellar evolution, plate tectonics, and weather systems
- We ran a numerical simulation of RBC based on the Boussinesq approximation, in which we assume that fluids develop a buoyancy force that is linear in the temperature
- The simulation was run on the Harvard Odyssey computing cluster using the Drekar simulation code developed at Sandia National Lab

Description of Code

- Drekar is a large scale computational fluid dynamics (CFD) code developed at Sandia National Lab
- Drekar solves fluid PDEs using the Finite Element Method (FEM)
- Core functionality is a massively parallel FEM implementation that runs on MPI with back ends from threads, OpenMP and CUDA
- Drekar is part of the Trilinos package, which provides functionality including linear algebra, nonlinear solvers, automatic differentiation, and time integration

Overview of Numerical Methods Used

- The Finite Element Method (FEM) solves PDEs by discretizing them in space on a *mesh*.
- The functions to be solved for (here pressure, temperature, and velocity) are represented by choosing a finite dimensional basis
- This simulation used piecewise polynomials of order 1, i.e. rectilinear planar functions (analog to piecewise linear in 2D)
- The time stepper discretizes in time; each time step leads to a set of linear equations to be solved and then a nonlinear solution

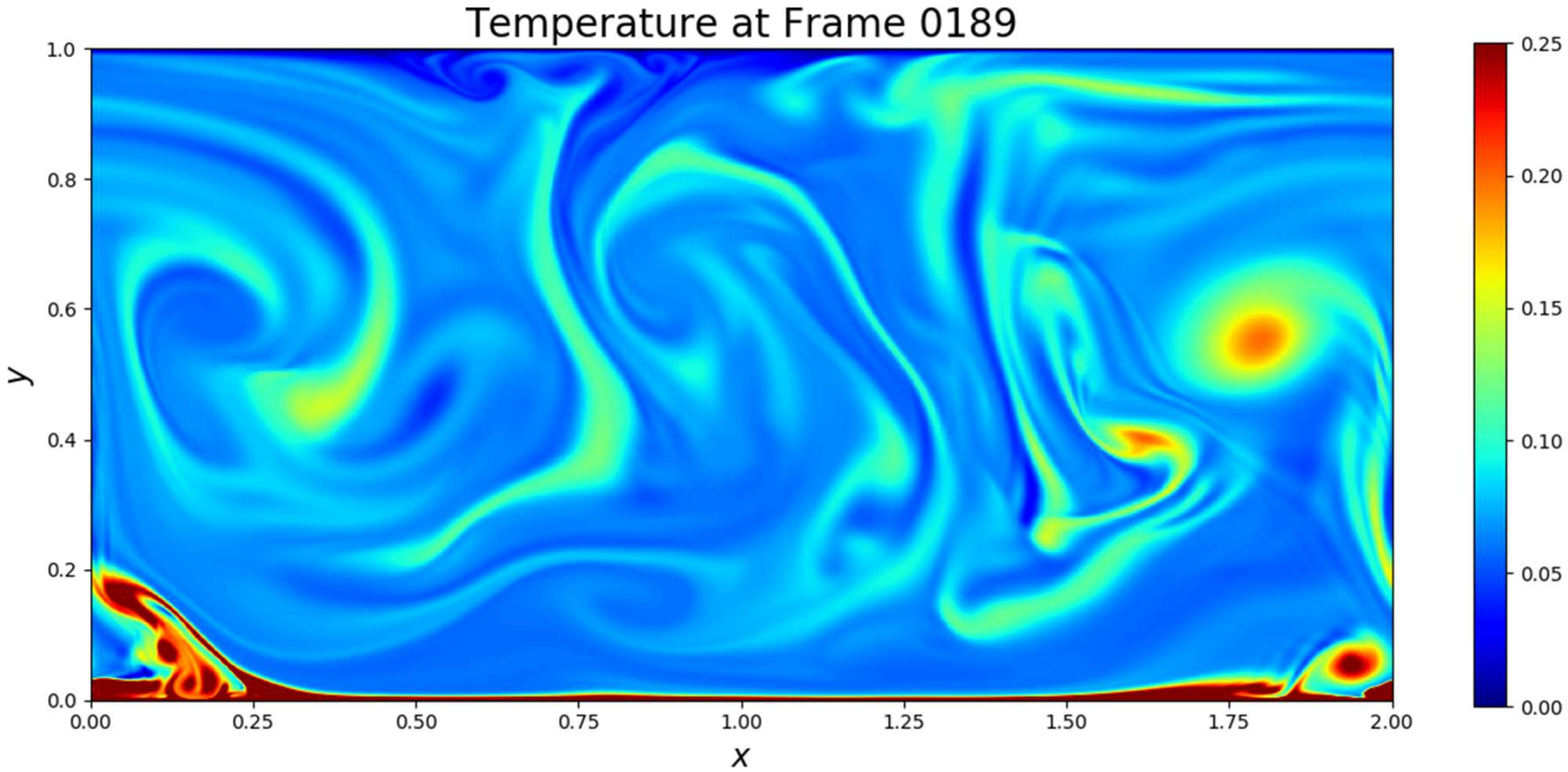
Parameters of the Simulation

- Dimensions: Length L=2.0, Height =1.0, $\Gamma = 2$; $N_x = 1024$, $N_y = 2048$
- Density $\rho_0 = 1.0$
- Viscosity $\nu = 0.01$
- Volume Expansion Coefficient $\alpha_V = 10^6$
- Gravity $g = 1.0$
- Heat Capacity $C_p = 1.0$
- Thermal Conductivity $k = 0.01$
- Thermal diffusivity $\kappa = k / \rho_0 C_p = 0.01$
- Temperature Change $\Delta T = 1.0$
- Time of Simulation: 5.0 target (0.22 achieved on case 1)

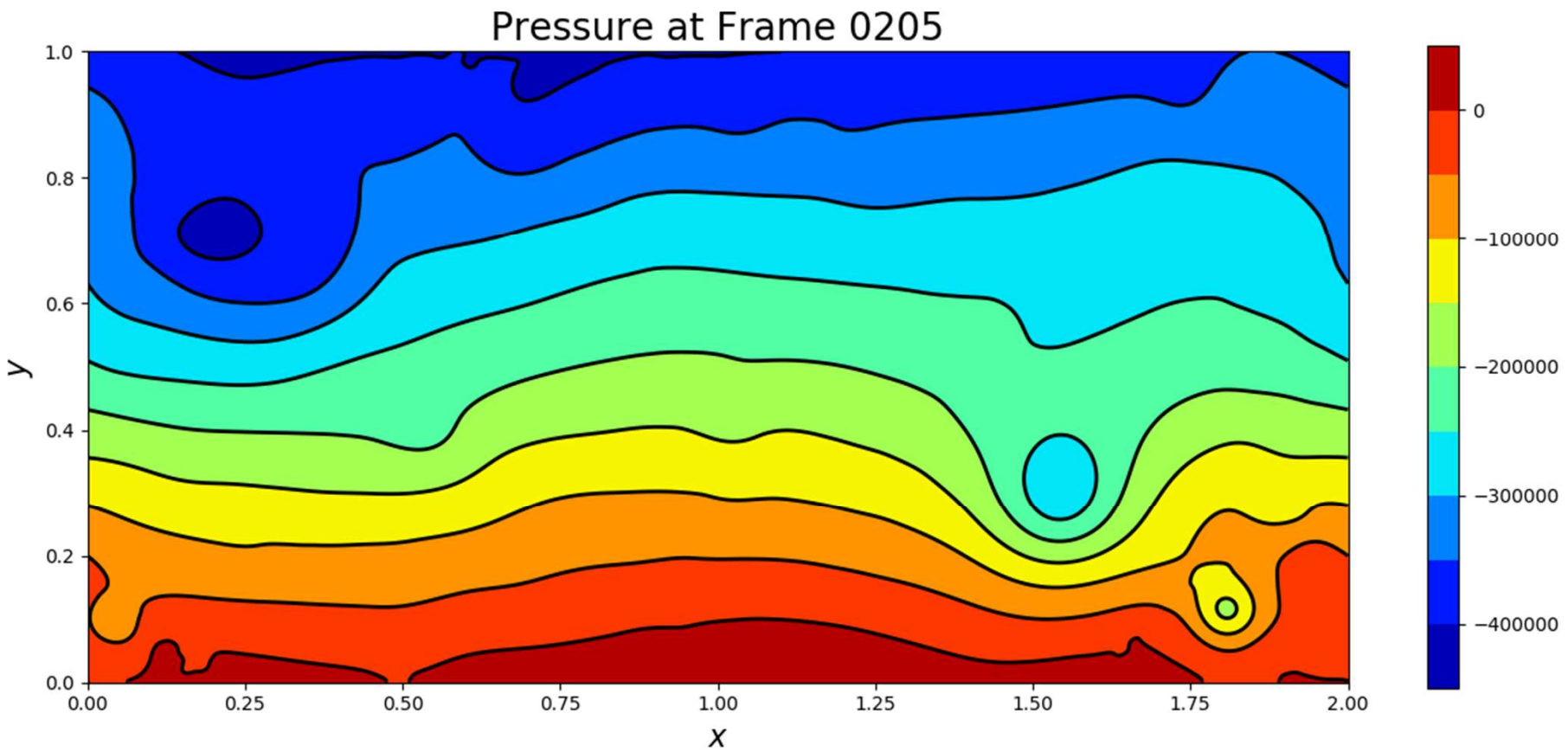
Results

- Case 1 was initiated on 1024 cores but only ran on 512 cores due to a hardware fault.
- It ran for about 1 day and got to time 0.22 before we restarted it.
- Case 2 ran on all 1024 cores for about 5 days. It bogged down, getting progressively slower, and eventually crashed (out of memory)
- It only got to a simulation time of 0.77; it should have gotten much further. After it crashed, out .exo output files became corrupted.
- We suspect a memory leak
- We present visualizations from the successful case 1 simulation

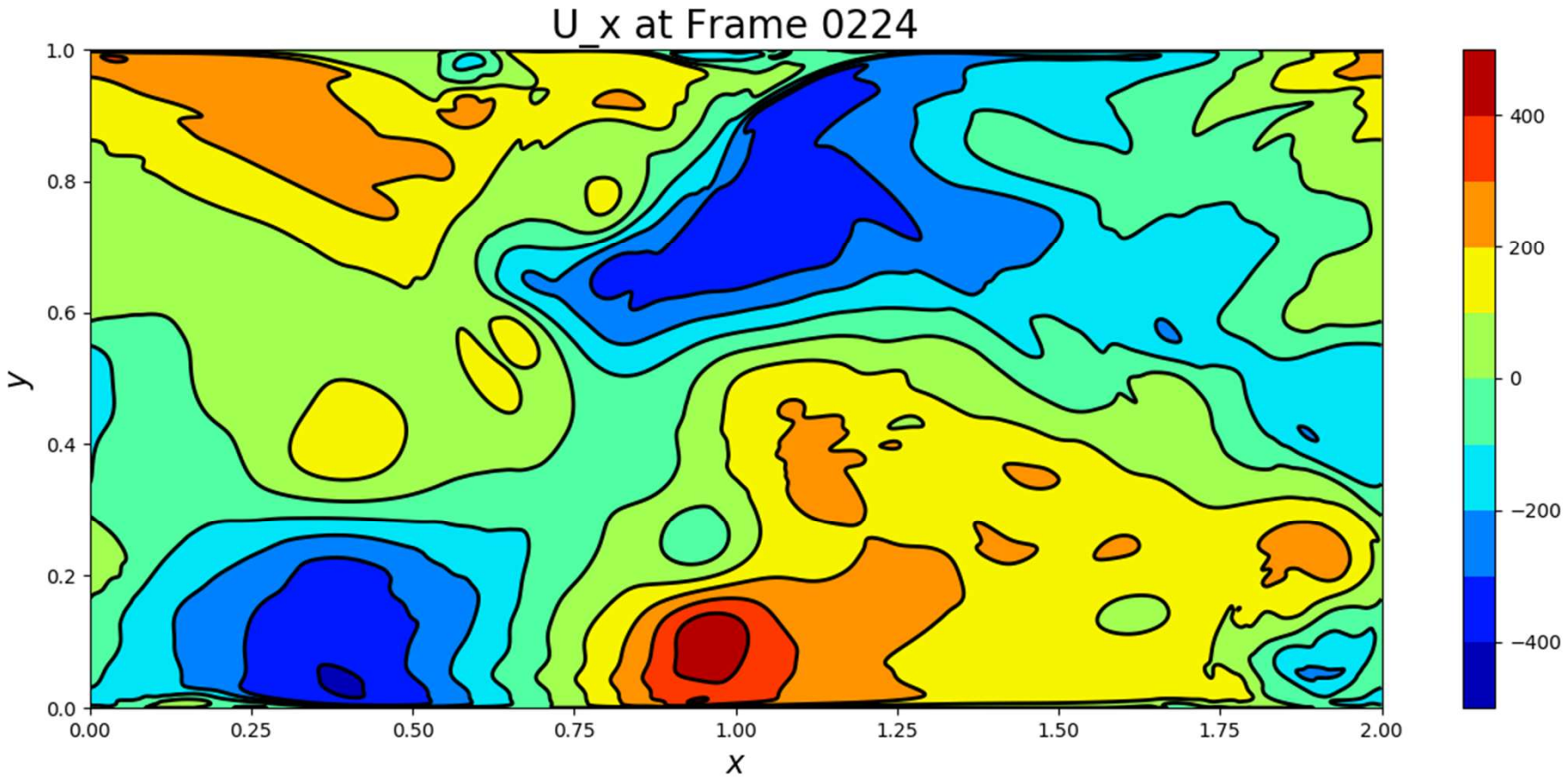
Sample Output: Temperature



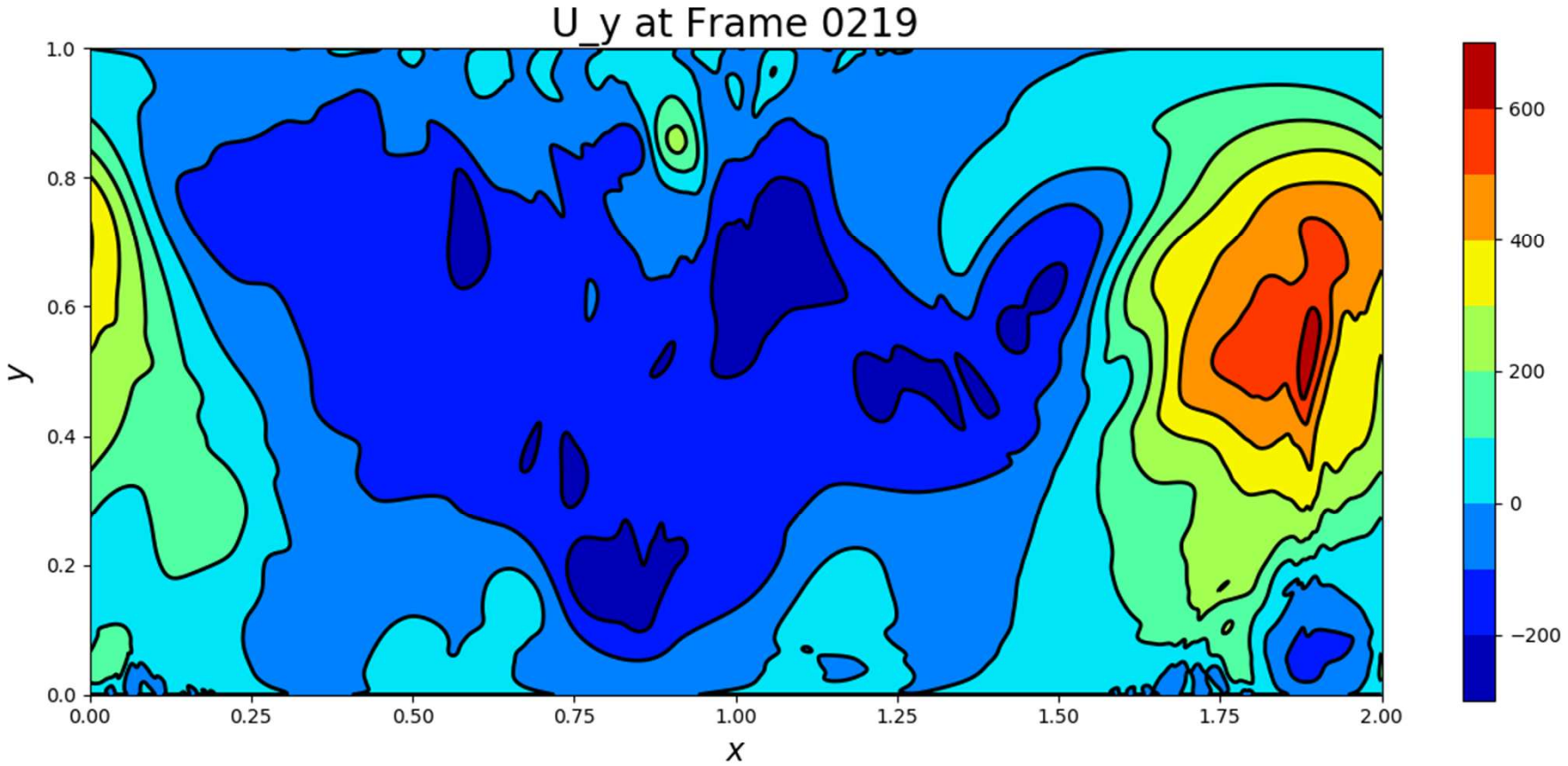
Sample Output: Pressure



Sample Output: Velocity in x Direction



Sample Output: Velocity in y Direction



Sample Output: Velocity Field & Streamlines