GPU-BASED FLUID STRUCTURE INTERACTION

Christopher S. Minar & Dr. Kyle E. Niemeyer

minarc@oregonstate.edu

INTRODUCTION

In the world of wave energy, computational fluid dynamics is a powerful tool that designers can use to develop better wave-energy converters (WEC). For a WEC to be competitive in the energy market, it needs to be able to produce as much energy as possible while also being able to withstand harsh ocean conditions. Designing better WECs requires fast and efficient fluid dynamics solvers.

We are developing a tool to solve the Navier-Stokes equations for flow around complex bodies using the computational power of graphics processing units (GPUs). The solver uses the immersed boundary method to handle complex bodies. This method adds a forcing term, $\boldsymbol{f}$, to the Navier-Stokes equations that imposes the force of a body onto the fluid:

$$\frac{\partial \boldsymbol{u}}{\partial t} + \nabla(\boldsymbol{u}\boldsymbol{u}) = -\nabla p + \nu \nabla^2 \boldsymbol{u} + \boldsymbol{f}$$

This can be discretized using the fractional step method (Fadlun et al., 2000). First, the Navier-Stokes equations are solved for an intermediate velocity $\hat{\boldsymbol{u}}$. Then, $\hat{\boldsymbol{u}}$ is used to solve for a pressure correction term ϕ. Finally, $\hat{\boldsymbol{u}}$ and ϕ are used to update the velocity and pressure:

$$A\hat{\boldsymbol{u}} = \Delta t\left(\boldsymbol{RHS}^l\right) + \boldsymbol{u}^l$$

$$\nabla^2 \phi = -\frac{\nabla \hat{\boldsymbol{u}}}{\Delta t}$$

$$\boldsymbol{u}^{l+1} = \hat{\boldsymbol{u}} - \Delta t \nabla \phi$$

$$p^{l+1} = p^l + \phi - \frac{\Delta t}{2Re}\nabla^2 \phi$$

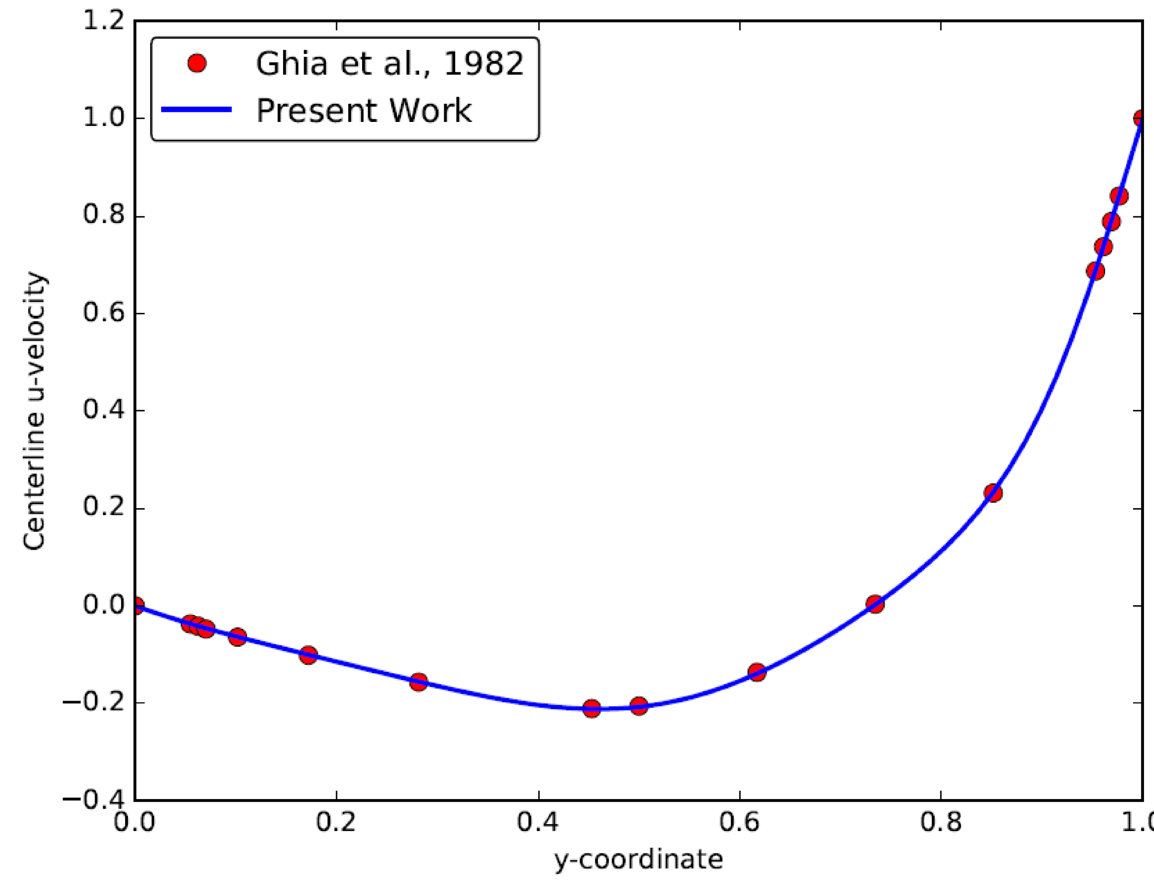
HARDWARE

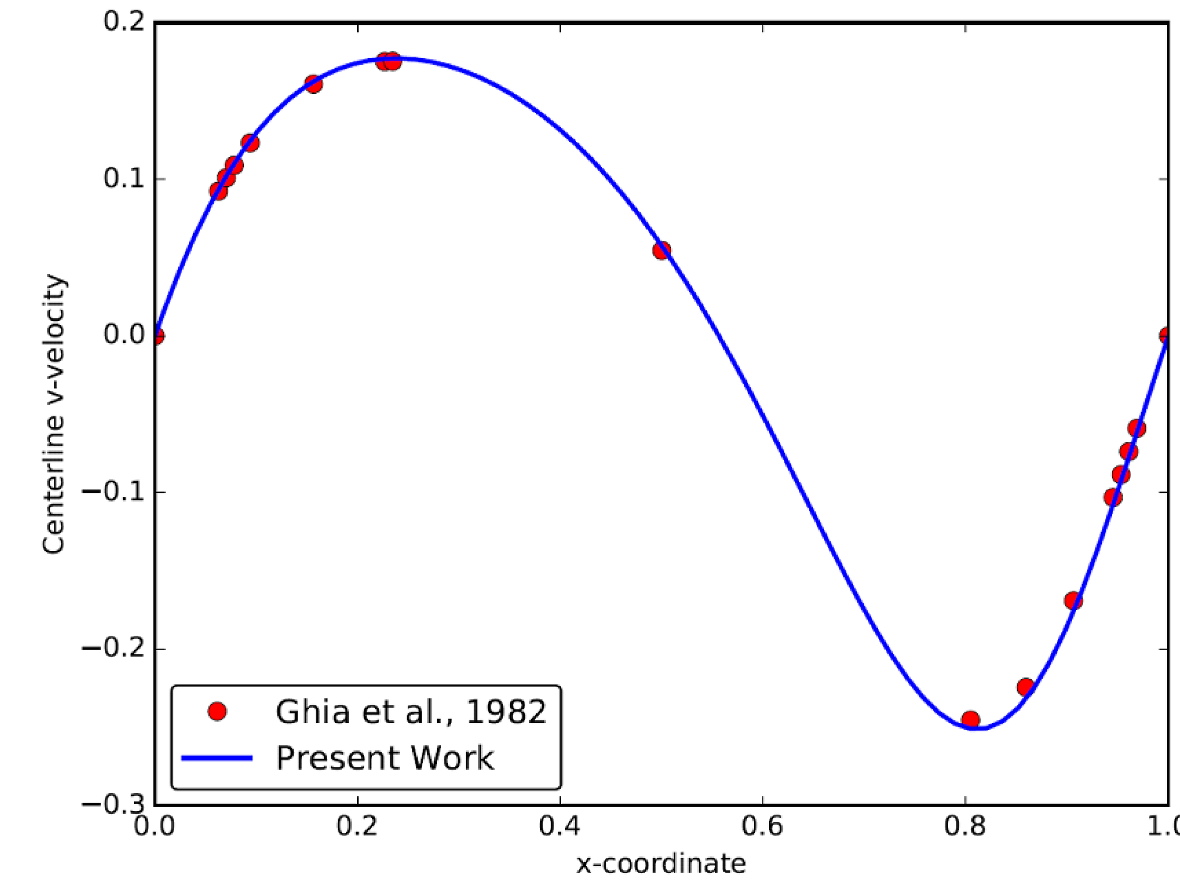
The solver is being tested on an NVIDIA Tesla K40 GPU, pictured below. The performance of this card peaks at around 4.3 TFLOPS for single-point floating precision, as compared with 50-100 GFLOPS for a modern Intel i7 CPU.

ACKNOWLEDGEMENTS

We gratefully acknowledge the support of NVIDIA Corporation with the donation of the Tesla K40 GPU used for this research.

RESULTS & VALIDATION



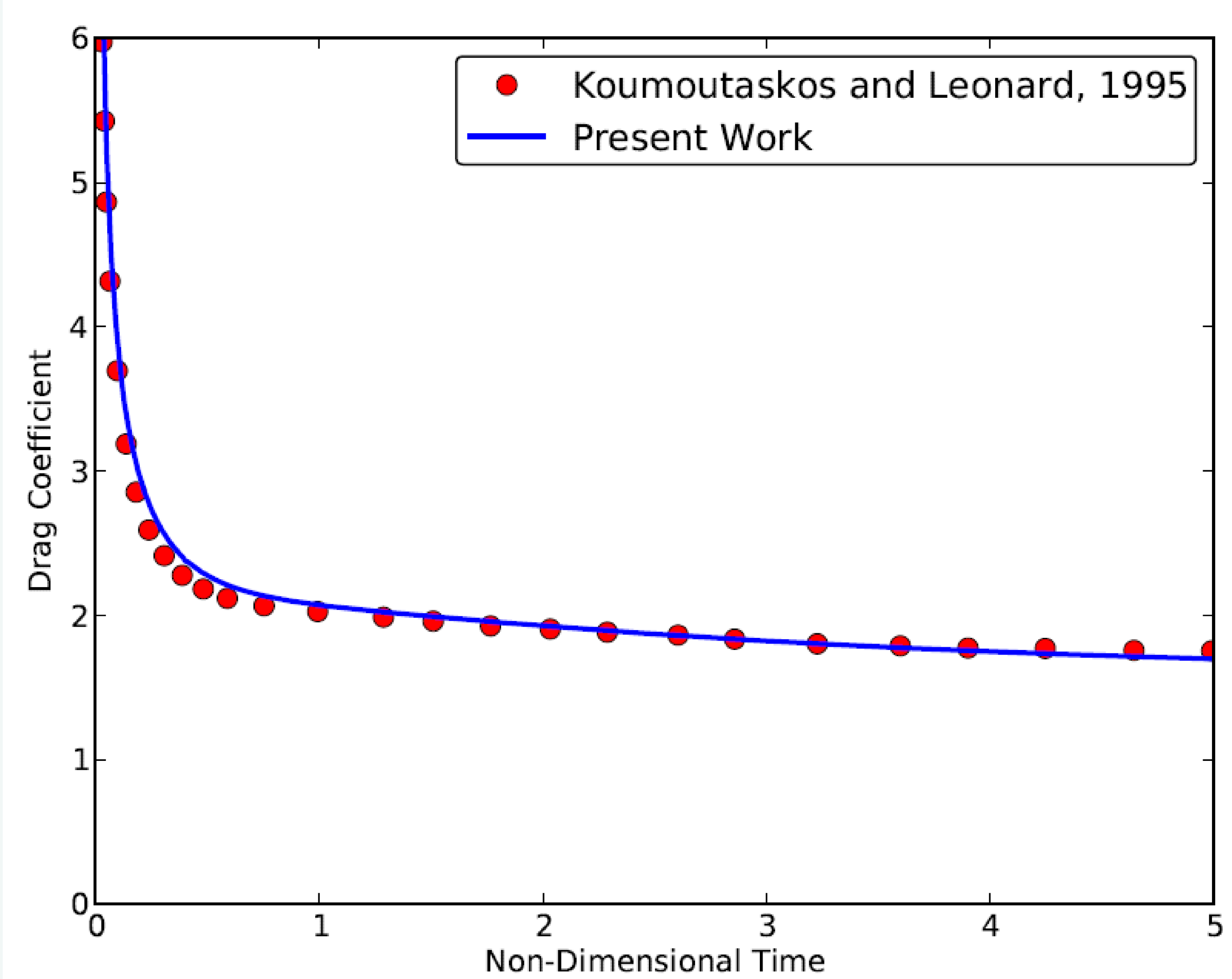


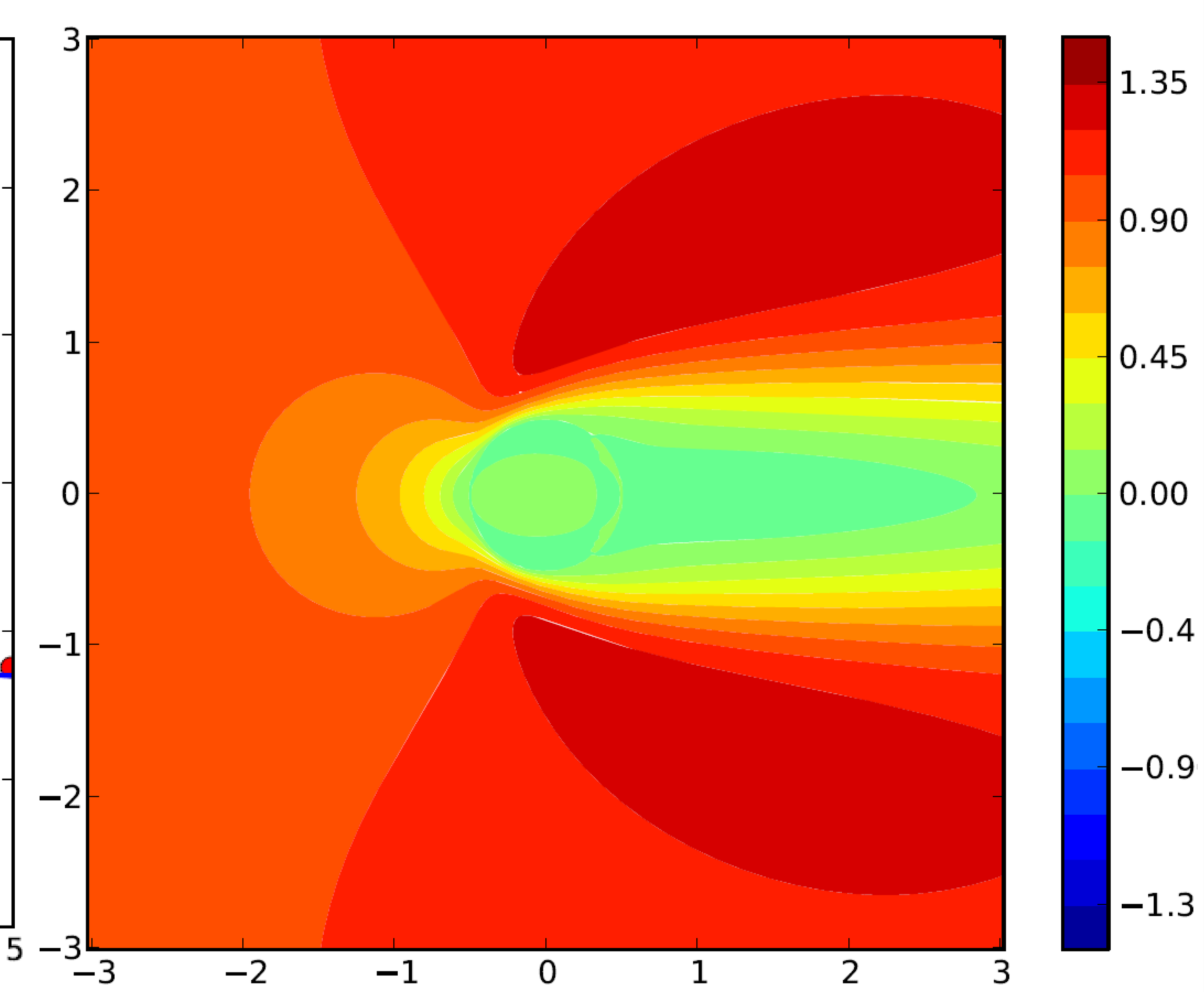
LID DRIVEN CAVITY VALIDATION

The above figures show validation of the solver using the classic Navier-Stokes case of flow in a lid-driven cavity. In this simulation, flow inside of a two-dimensional box with a moving lid is modeled. The lid is kept at a constant velocity and the viscosity (via Reynolds number) is changed. The velocities at the centerlines are then compared to results obtained in literature. The left figure shows the *x*-velocity at the horizontal centerline and the right figure shows the *y*-velocity at the vertical centerline, for a Reynolds number of 100.

IMPULSIVELY STARTED CYLINDER VALIDATION

The below figures show validation for impulsively started flow over a cylinder. In this validation case, two-dimensional flow with unity velocity starts around a stationary cylinder. The figure on the left shows the drag coefficient as it changes with the developing flowfield. The right figure shows a fully developed *x*-velocity profile. The Reynolds number for this simulation is 40.

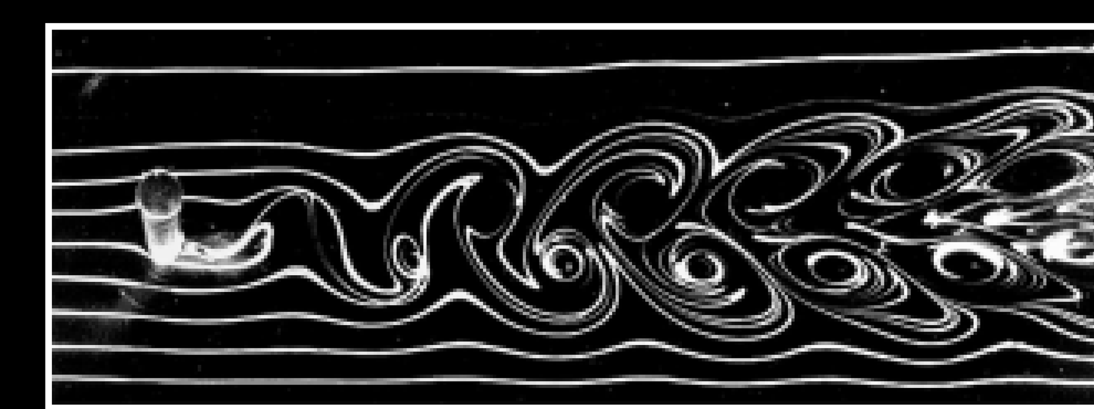




LEAVE BLANK

FUTURE WORK

The next step for the solver is to let the body move freely due the forces imposed upon it from the fluid. This is called fluid-structure interaction. The validation case for fluid-structure interaction is vortex-induced vibration flow. The simulation is similar to the impulsively started cylinder validation except the body is allowed to move in the *y*-direction. A restoring force will be applied to the body (modeled as a spring) and the movement of the body will be validated for various spring constants.



Blevins, 2009

CITATIONS

Blevins, R. D. (2009). Models for vortex-induced vibration of cylinders based on measured forces. *Journal of Fluids Engineering*, *131*(10), 101203.

Fadlun, E. A., Verzicco, R., Orlandi, P., & Mohd-Yusof, J. (2000). Combined immersed-boundary finite-difference methods for three-dimensional complex flow simulations. *Journal of Computational Physics*, *161*(1), 35-60.

Ghia, U., Ghia, K. N., & Shin, C. T. (1982). High-Re solutions for incompressible flow using the Navier-Stokes equations and a multigrid method. *Journal of Computational Physics*, 48(3), 387-411.

Koumoutsakos, P., & Leonard, A. (1995). High-resolution simulations of the flow around an impulsively started cylinder using vortex methods. *Journal of Fluid Mechanics*, 296, 1-38.

Oregon State UNIVERSITY