Simulation of spatially evolving flow past a sphere in a stratified fluid

Matthew B. de Stadler, Advisor: Prof. Sutanu Sarkar

UCSD Jacobs | Mechanical and Aerospace Engineering

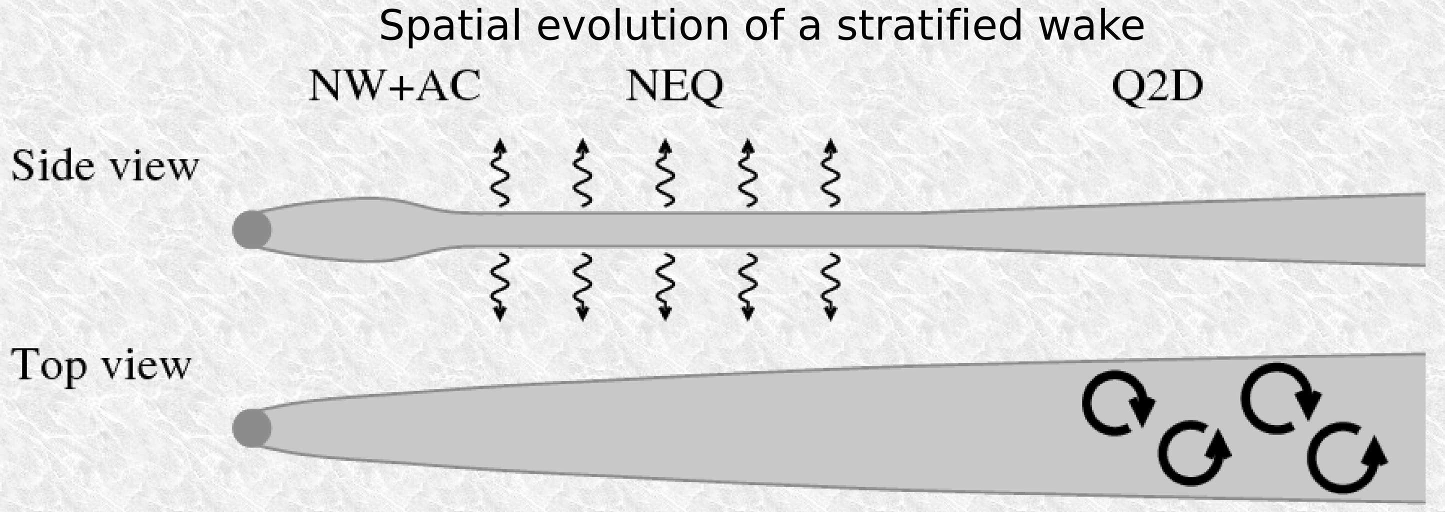
Motivation

Flow past a sphere is an established benchmark problem as it combines fully three-dimensional unsteady flow dynamics with a transition to turbulence. The presence of a density gradient significantly complicates matters as it destroys the symmetry of the problem and introduces a complex coupling between kinetic and potential energy. The standard approach for numerical simulation of flow past a sphere in a density stratified fluid is to use a temporal approximation to relate time evolution in an auxiliary domain not resolving the sphere to distance downstream of the sphere in a spatial frame. Very few simulations resolving the sphere in the computational domain have been performed and those that have are characterized by low Reynolds number or the use of turbulence models. The goal of the present study is to simulate spatially evolving flow past a sphere at high Reynolds number without the use of turbulence models.

Principal motivation: To characterize the near to intermediate wake region as buoyancy effects become significant

Background

Spatial evolution of a stratified wake



Wake evolution in the vertical, x_3, and horizontal, x_2, directions. Curvy arrows show the time when internal waves are significant and pancake eddies are shown in the late wake.

Formulation

Simulation details

Direct numerical simulation

Collocated grid arrangement using pressure-correction algorithm

Semi-implicit mixed RK3-ADI method for time advancement

RK3 for convective terms, ADI for diffusive terms

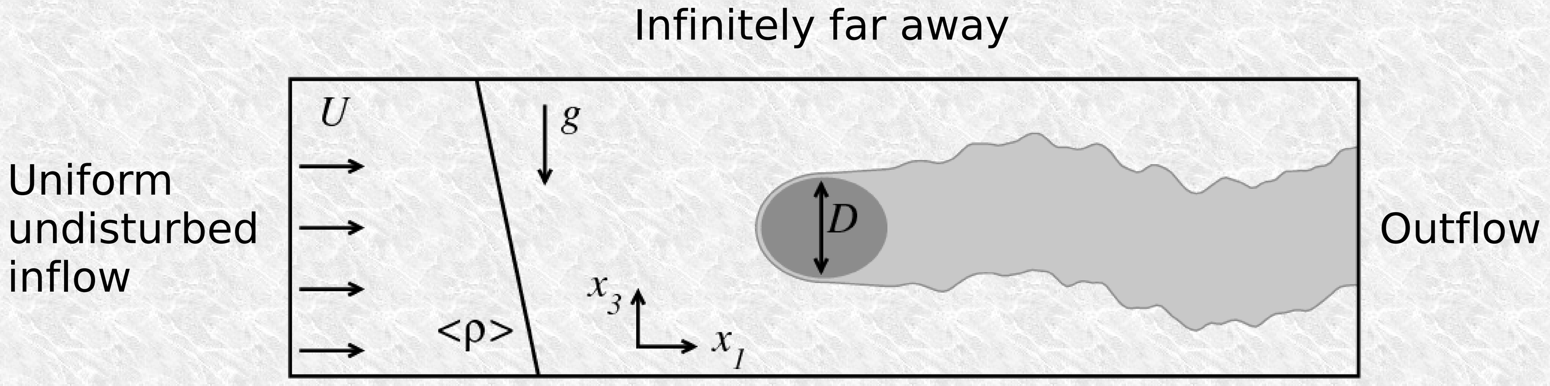
2nd order centered differences for spatial derivatives

Multigrid pressure solver

3D domain decomposition, parallelization with MPICH-II

Computational test section

Infinitely far away



Governing equations

3D incompressible, unsteady Navier-Stokes equations, Boussinesq approx.

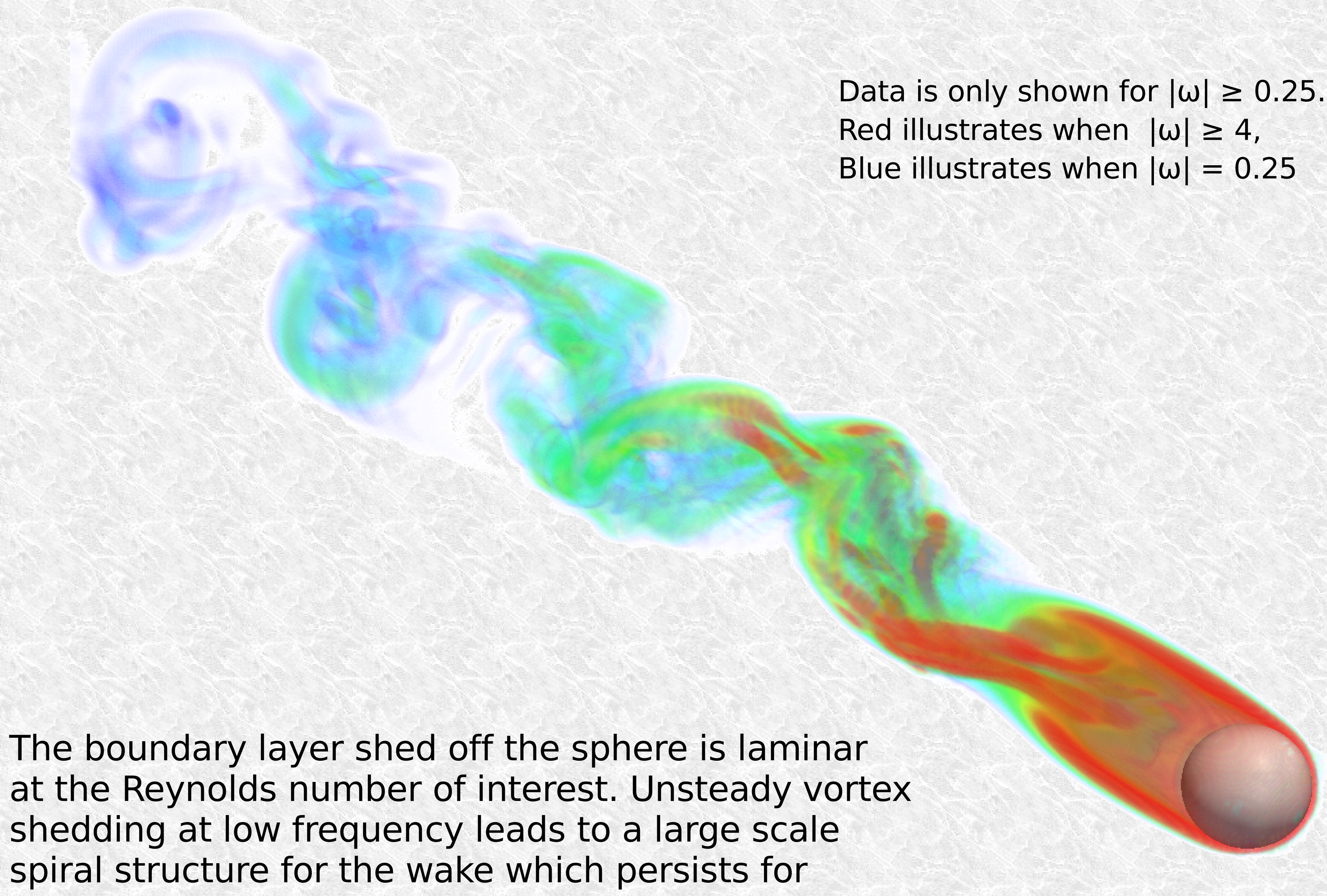
Momentum $$\frac{\partial u_i}{\partial t} + \frac{\partial (u_k u_i)}{\partial x_k} = -\frac{\partial p}{\partial x_i} + \frac{1}{Re}\frac{\partial^2 u_i}{\partial x_k \partial x_k} - \frac{1}{Fr^2}\rho' \delta_{i3}$$

Density $$\frac{\partial \rho}{\partial t} + \frac{\partial (u_k \rho)}{\partial x_k} = \frac{1}{RePr}\frac{\partial^2 \rho}{\partial x_k \partial x_k}$$ Mass $$\frac{\partial u_k}{\partial x_k} = 0$$

$$Re = \frac{UD}{\nu}, \qquad Fr = \frac{U}{ND}, \qquad Pr = \frac{\nu}{\kappa}$$

Visualization of flow past a sphere

Here we are looking at instantaneous transparent contours of the vorticity magnitude at Re=1,000 to visualize vortical structures in the wake.



Data is only shown for |ω| ≥ 0.25.
Red illustrates when |ω| ≥ 4,
Blue illustrates when |ω| = 0.25

The boundary layer shed off the sphere is laminar at the Reynolds number of interest. Unsteady vortex shedding at low frequency leads to a large scale spiral structure for the wake which persists for significant distance downstream. The separated boundary layer forms a thin shear layer which becomes unstable due to a Kelvin-Helmholtz type instability leading to small scale breakup of the spiral structure. Further downstream, the growth of instabilities leads to a transition from unsteady laminar flow near the sphere to turbulent flow in the intermediate wake.

Immersed boundary method

The immersed boundary method is a technique for simulating flow with complex geometry on a simple grid.

Basic idea:

1. Classify grid points as solid or fluid
2. Modify Navier-Stokes equations with forcing term to represent complex geometry boundary at locations not coincident with with gridpoints
3. Modify computational stencil around immersed body
4. Solve Navier-Stokes equations and equations for interpolation points simultaneously

Implementation:

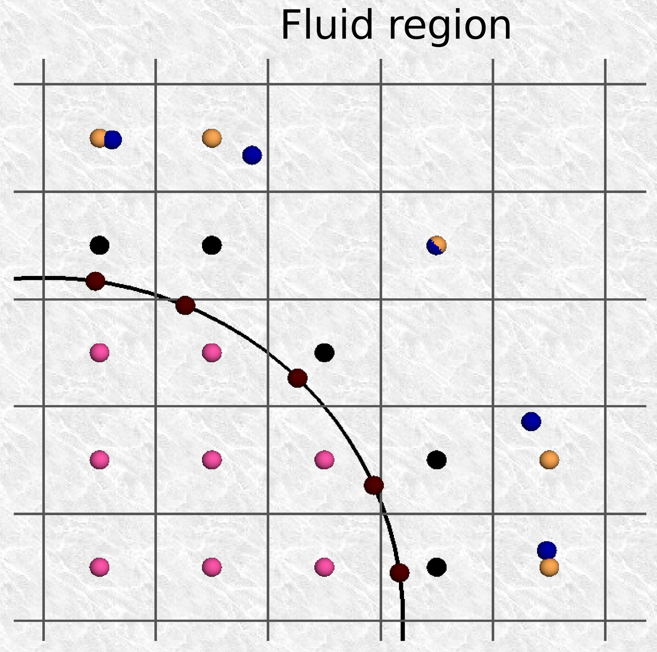
Algorithm of Roman et. al., *Computers & Fluids*, 2009

Decouples fluid and solid nodes, sharp interface

Designed for semi-implicit time advancement

Single solution of momentum equations at each step

Velocities in immersed boundary points are calculated by linear interpolation of intersection points on the body and projection points away from the body.

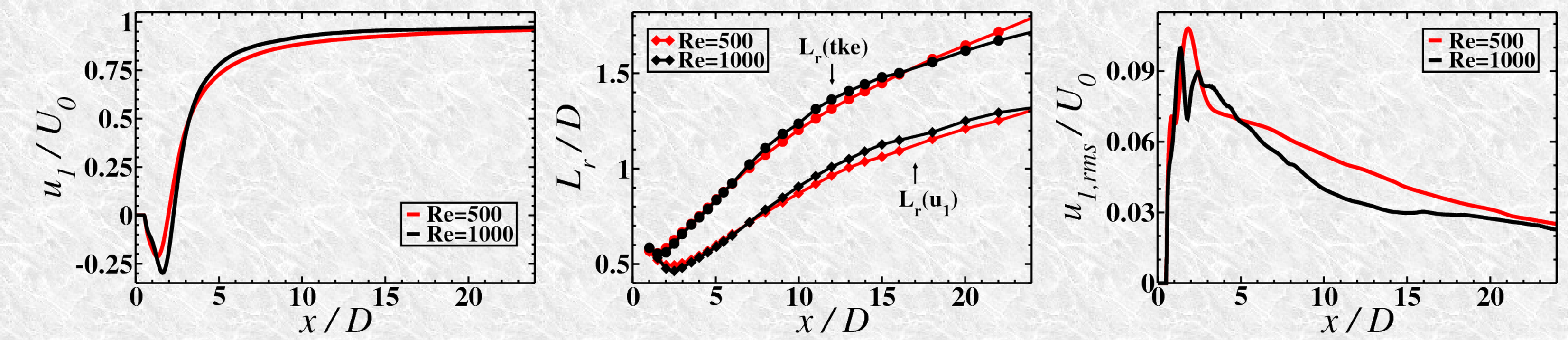


Velocities at the intersection points are known from the no-slip boundary condition.

Velocities at the projection points are calculated from a Taylor series expansion about the nearest fluid point.

- Immersed boundary point (IB)
- Intersection point (IP)
- Projection point (PP)
- Solid cell
- Nearest fluid neighbor to PP

Wake evolution

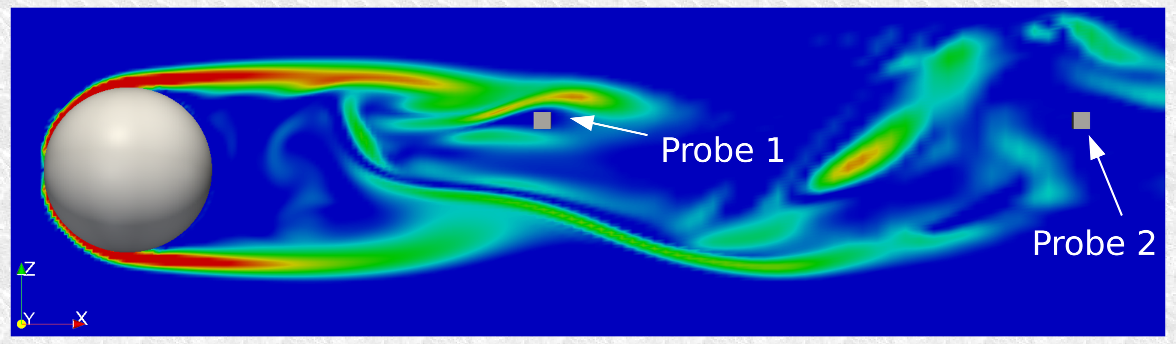


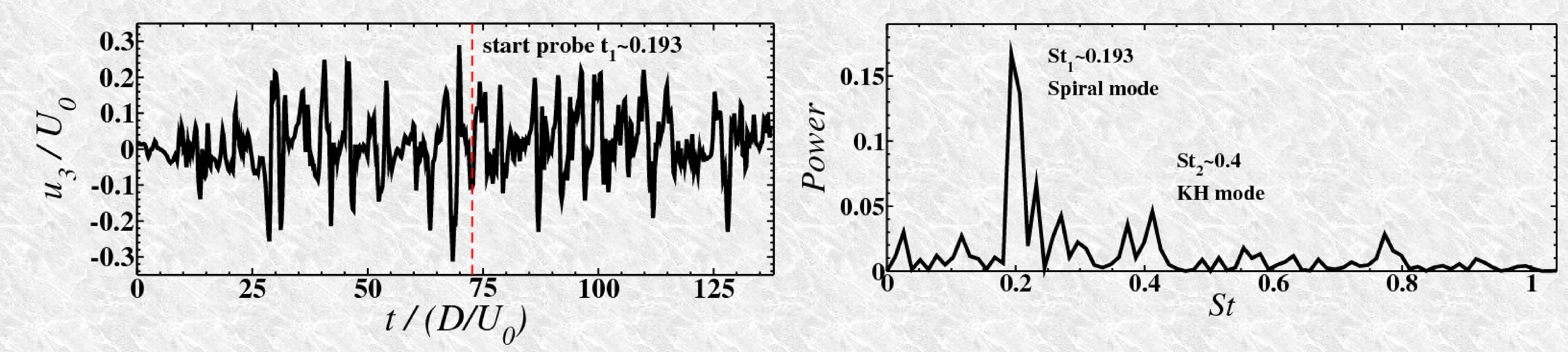
(Left) Mean streamwise velocity along the wake axis. (Middle) Wake width based on the mean velocity (diamond symbols) and the turbulent kinetic energy (circle symbols). (Right) Root mean square streamwise velocity along the wake axis. Data shown is averaged over 12 shedding periods for Re=1,000 and 15 shedding periods for Re=500.

Mean velocity: recirculating region near sphere (negative u_1) followed by steep decay.

Velocity fluctuations spread more than the mean flow

Velocity fluctuations increase sharply in the near wake





(Top) Instantaneous vorticity magnitude on an x_1-x_3 plane. Blue indicates |ω| ≤ 1, Red indicates |ω| ≥ 8. Grey boxes show the location of velocity probes. Probe 1 is located at x/D = 2.5, z/D= 0.3. Probe 2 is located at x/D = 5.75, z/D= 0.3. (Bottom left) Vertical velocity as a function of time at probe 2. (Bottom right) Power spectrum for u_3 at probe 2. The start of the averaging period is shown at left with the dashed red line. Data is averaged over 12 shedding periods. All 3 plots are for the Re=1,000 case.

Characteristic frequencies of spiral mode (St_1) and Kelvin-Helmholtz mode (St_2) are captured despite coarse resolution

St_1 very close to the 0.195 value reported in the literature

St_2 over-predicted (present St_2~0.4 vs. 0.35 in Tomboulides and Orszag)

Conclusions

Immersed boundary solver is giving qualitatively correct behavior for an unstratified turbulent wake behind a sphere at Re=500 and Re=1,000.

Results agrees qualitatively with the simulations of Tomboulides & Orszag, *JFM* 2000 and the experiments of Wu & Faeth, *AIAA* 1993. Quantitative discrepancies occur due to coarse temporal and spatial resolution in the present study.

Future work

Increasing the Reynolds number

Incorporating the effect of density stratification

Target values for thesis project: Re=10,000, Fr=4

Optimize numerical implementation

Target simulation requires O(1 billion grid points) and O(100,000) CPU hours

Acknowledgments

Matthew de Stadler received support from the Department of Defense (DOD) through the National Defense Science & Engineering Graduate Fellowship (NDSEG) Program and an Achievement Rewards for College Scientists Scholarship. Partial support was provided by ONR grants N00014-07-10133 and N00014-11-10469. All simulations were performed on the Utility Server at the ERDC DSRC.