

# Project 4 (30 points)

EOC 6189 - Computational Fluid Dynamics  
Florida Atlantic University

March 23, 2019

**Collaboration policy:** *You are strongly encouraged to discuss your work with fellow students, to brainstorm ideas, and to learn from each other. However, you are not allowed to copy solutions/code from each other, from the internet, or from any other source. All work shown must be your own. You are expected to follow the University Honor Code at all times.*

**Instructions:** Please create a single zip file containing all your PDFs, movies, etc., and upload it on Canvas by the deadline. Your code source files must be submitted via git repository. Try to keep your code as modular as possible, which involves writing separate functions, where one function does only one job (e.g., discretization, derivative computations, time integration, etc.). This will help you reuse your functions with minimal modification in future projects. Make sure to write short, but useful comments in the code and make regular git commits. Undocumented code will lose points. Please give the instructor access to your git repository when you are ready for submission.

## 1. 2D Navier-Stokes solver

- (a) For this problem, you will write your own incompressible Navier-Stokes solver in 2-dimensions. Use periodic boundary conditions in both  $x$  and  $y$  directions. The initial condition for the velocity components is given as:

$$u(x, y, t = 0) = \cos(x) \sin(y) \quad (1)$$

$$v(x, y, t = 0) = -\sin(x) \cos(y) \quad (2)$$

Use a domain of size  $2\pi$  units in both directions, and 31 grid points in each direction. Use a time step size of  $\Delta t = 1e - 2$ , viscosity  $\nu = 1e - 1$ , and run the simulation for  $t_{max} = 10s$ . You may use second order centered finite differences for both advection and diffusion, as well as for any other derivatives you need to compute (*Note: pure advection is unconditionally unstable with centered differences, but the combination is conditionally stable, so we do not need to use upwinding*).

- (b) Create a video showing the 2D velocity quiver plot as it evolves in time (please create an mp4 - MPEG4 format file).
- (c) The configuration you have solved is called the ‘Taylor-Green vortex’, and it is one of the few problems in fluid mechanics that have an analytical solution:

$$u(x, y, t) = \cos(x) \sin(y) \exp(-2\nu t) \quad (3)$$

$$v(x, y, t) = -\sin(x) \cos(y) \exp(-2\nu t) \quad (4)$$

Create a second quiver plot video, that overlays this exact solution on top of your numeric solution. Do the two solutions match?

- (d) Compute the error in your solution, and plot it as a function of time from  $t = 0$  to 10. Are you happy with your solution? What could you try to improve it?

## 2. Fluid-solid interaction

- (a) For this problem, you will expand your Navier-Stokes solver to include the capability of handling static walls. Use the same domain as in the previous problem (domain length =  $2\pi$  units in each direction), but this time impose a constant  $u = 1$ ,  $v = 0$  throughout the domain as initial conditions. Use  $Nx = Ny = 51$ ,  $\nu = 1e - 2$ ,  $\Delta t = 1e - 2$ , and  $t_{max} = 10s$ .

Include two parallel walls that are separated by 2 units in the  $y$  direction, and are centered in the domain (i.e., one wall is above the  $y = \pi$  line, and the other is below it). The walls are rigid and immobile. The wall thickness is 0.5 units in  $y$ , and their length in  $x$  spans the entire domain. When using periodic BCs, this configuration represents an infinitely long channel.

For including the influence of the walls on the fluid, use the Brinkman penalization approach we discussed in class. The relevant forcing term may be taken as:

$$F_{penalty} = \lambda \chi (\mathbf{u}_s - \mathbf{u}) \quad (5)$$

where  $\chi$  is the characteristic function,  $\mathbf{u}_s$  is the velocity vector corresponding to the solid, and  $\mathbf{u}$  is the velocity of the fluid. For the penalty parameter, set  $\lambda = 1/\Delta t$ .

- (b) Run your simulation until  $t_{max} = 10s$ , and create a video showing the velocity vectors. Make sure to indicate the presence of the walls in some manner (shaded box, or any other suitable method).
- (c) What should the velocity profile inside the channel look like? Is this indeed the case for your simulation? Show this by making a plot of  $u$  along an appropriate line cut through the domain. What is the name given to this type of flow (*Hint: It's French*)?
- (d) Create a video showing how vorticity evolves in the domain. Does your computed vorticity value make sense close to the wall? Explain.