MAE 6263 Computational Fluid Dynamics

Project 3

Instructions:

1. *Refer to the 'Academic Professionalism' and 'Academic Dishonesty' sections of the course outline document before you start working on this homework (HW) set.*
2. *The **typed** project report should be handed over by the end of 05/11/2018. These need to be submitted as a soft copy along with the source code, input files and plotting script.*
3. *The source codes need to be in one of the lower level languages like C, C++, F77, F90, java or Matlab/python (without leveraging inbuilt functionality not available in F90/C/C++) etc. Please discuss with me in case you have questions or need special accommodation.*
4. *Please write your full name, CWID and page number on the top right corner of each sheet of paper used in your HW assignment. The page numbers need to be written as 1/x,2/x,3/x….to ensure the integrity of your submitted assignments and neatly stapled.*

Motivation: This project will focus on your ability to apply the basic concepts of finite difference/finite-volume algorithms and pressure-velocity coupling to the solution of the incompressible Navier-Stokes equation to model a pressure driven channel flow in two dimensions. The presentation of the results from this project is intended to train you to develop professional reports of your work as you embark on your research career. So, please take this seriously.

Problem: Solve the 2-D NS equation in uniform collocated grid for a channel flow as shown.

$$\left.\begin{array}{l} \dfrac{\partial \vec{u}}{\partial t} + \vec{u}.\nabla\vec{u} = \upsilon\Delta\vec{u} - \dfrac{\nabla p}{\rho} \\ \nabla.\vec{u} = 0 \end{array}\right\} over\ \Omega$$

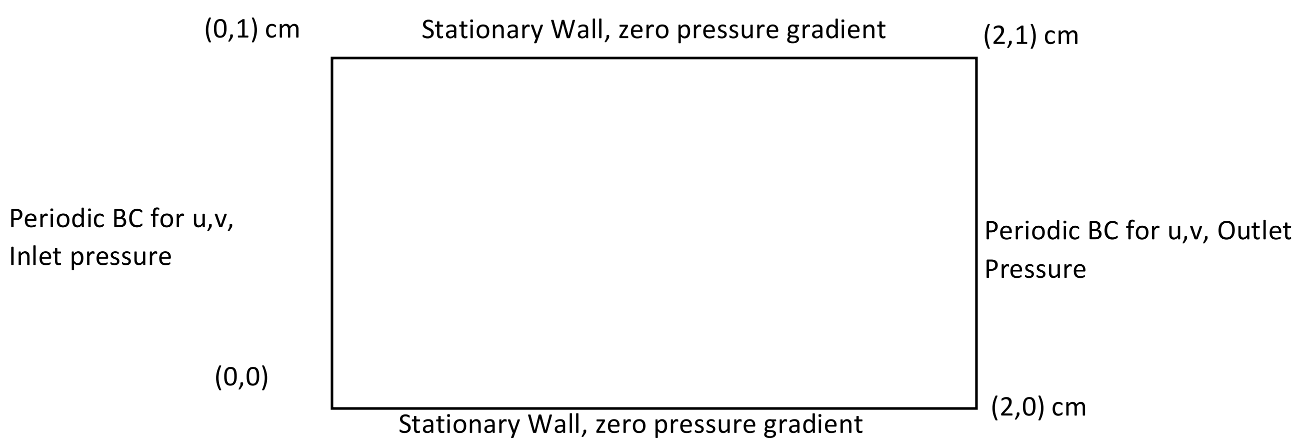
$$At\ t=0 \left\{\begin{array}{l} \vec{u}(x,y)=0\ \ for\ \ 0\le x\le 2, 0\le y\le 1; \\ p(x,y)=0\ \ everywhere\ \ except\ boundaries \end{array}\right\}$$

$$BC:\left\{\begin{array}{l} \vec{u}(x,y)=0\ \ for\ y=1; \\ \vec{u}(x,y)=0\ \ for\ y=0; \\ \vec{u}(x,y)=\vec{u}(x-2,y)\ \ for\ x=2; \\ \vec{u}(x,y)=\vec{u}(x+2,y)\ \ for\ x=0; \end{array}\right\}$$

$$BC:\left\{\begin{array}{l} p(x,y)=p_{outlet}\ for\ x=2; \\ p(x,y)=p_{inlet}\ for\ x=0; \\ \dfrac{\partial p}{\partial y}=0\ for\ y=0\ and\ \ y=1; \end{array}\right\}$$

Discretization Schemes: Forward Euler for time derivative, Central difference for the viscous term and first order upwind for the advection term (i.e. use FOU to estimate u_e in advection term). Use compact scheme for the pressure.

Divergence and Gradient Scheme: Use central difference or linear central scheme.



Assignment:

1. Neatly state the problem, boundary and initial conditions as you would in the project report.
2. Derive the pressure Poisson equation in continuous space.
3. Use a semi-discrete version of these equations to detail the explicit solution algorithm for the coupled set of equations.
4. Prove for a 1-d problem, whether the divergence and gradient schemes suggested will conserve kinetic energy or not.
5. Using the choice of discretization, divergence and gradient schemes suggested above, develop the appropriate 2D finite difference equation for the numerical method corresponding to a compact scheme for PPE without momentum interpolation. Explain how and why this method would fail based on what was taught in class?
6. Using the choice of discretization, divergence and gradient schemes suggested above, develop the appropriate 2D finite difference equation for the numerical method corresponding to a compact scheme for PPE with momentum interpolation. Why would the momentum interpolation address the issue that will be faced in the numerical method developed in 5? Note that for solving the PPE with momentum interpolation,

one would require boundary conditions at the node outside the boundary. Please treat this node as a ghost node whose value is to be extrapolated from the interior using a linear scheme.

7. <u>Code Development</u>: Develop a computer program (i.e. build on projects 1 & 2) to solve this coupled set of equations using both the strategies i.e. 5 and 6 – compact scheme for PPE with and without momentum interpolation. Even though the momentum equation is treated explicit, the pressure waves are handled implicitly which will require inversion of a coefficient matrix to solve $\bar{\bar{A}}\bar{p}^{n+1} = \bar{b}^{n}$. Use the Guass-Seidel method or another appropriate method to compute this matrix inversion and a convergence tolerance < 1.0e-5 for the iteration in the sense of an $L_{1,norm}$.
    a. Give the structure of tensors/vectors A,b.
    b. Include/print actual code text as an appendix to the project report along with submission of the actual source code.
8. <u>Designing Numerical experiments</u>: We will design a simulation of a 2D channel flow for $\mathrm{Re} = V_{peak}L_y / \upsilon = 100$. Use viscosity & density of water - $\upsilon = 1.0\times10^{-2}\,cm^2/s$ and $\rho = 1g/cc$. Do the following: (i) Identify the peak centerline velocity V_{peak} for the given Reynolds number ii) Use analytical expression for the parabolic profile for a 2D channel flow to obtain the pressure difference across the channel - $P_{inlet} - P_{outlet}$. Use a value for the outlet pressure of 100 to compute the inlet pressure. Choose a Peclet number, $Pe_{\Delta x} = V_{peak}\Delta x / \upsilon = 1.5$, $\Delta x = \Delta y$ and $\gamma = \upsilon\Delta t / \Delta x^2 = 0.2$.
9. <u>Numerical Simulations and Analysis</u>: Please carry out two simulations – one with momentum interpolation and the other without and plot the following for each of these runs.
    a. At steady state, isocontours of pressure and velocity field.
    b. At x=1.0, plot p vs x and u vs x.
    c. Plot the mass in the system vs time from t=0 to steady state.
    d. Plot the energy in the system vs time from t=0 to steady state.