ME 605 | Computational Fluid Dynamics

Project 4

Due: 11:59 pm, 24 November 2024

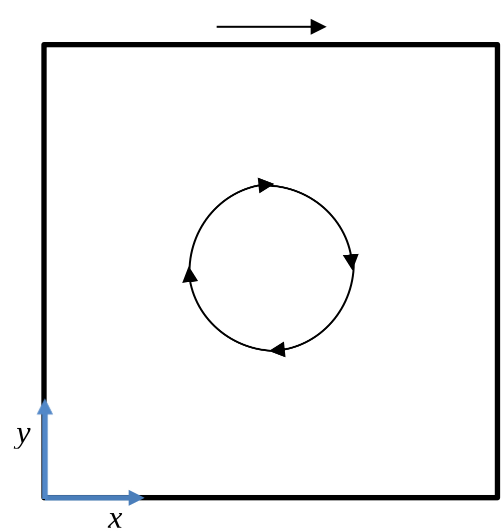
Instructions

1. You can choose any programming language of your choice.
2. Do not use any in-built or intrinsic functions. You are expected to write your computer program (including for solving the system of algebraic equations)
3. You are permitted to work in groups of a maximum of two students. The responsibility of forming groups lies with the students. Each group is expected to submit the project report and the code (one submission per group). Discussion among students (even across groups) is permitted.
4. Your report must consist of (1) a problem statement, (2) mesh details and approach for discretization, (3) derivation and presentation of the final form of the discretized equations, (4) solution methodology, (5) results and discussion; (6) concluding remarks. Note that an in-depth analysis and discussion of results is required.
5. The report must be prepared using WORD or LaTeX. Handwritten reports will not be accepted.
6. Submit the project report and the code in Google Classroom.

Project Statement

Lid Driven Cavity Flow

The flow domain is a unit square of length 1 unit. Let u and v denote the fluid velocities along x and y directions, respectively. The flow in the square cavity is driven by the top wall moving to the right at a constant velocity $u(x, 1) = 1$.This case is commonly referred to as the lid driven cavity flow. The schematic of the problem is given below:



The boundary conditions are given below:

$$u(0, y) = 0 \text{ and } v\ (0, y) = 0$$
$$u(1, y) = 0 \text{ and } v\ (1, y) = 0$$
$$u(x, 0) = 0 \text{ and } v\ (x, 0) = 0$$
$$u(x, 1) = 1 \text{ and } v\ (x, 1) = 0$$

The kinematic viscosity, ν, can be taken as 0.01. Note that you are required to derive the pressure boundary conditions by yourself.

You must solve the steady-state incompressible 2-D Navier-Stokes equations. Apply Finite Volume Method for discretization. You should use staggered mesh and SIMPLE algorithm.

Specifically, perform the following exercises:

1. Write a computer code to computationally solve the governing equations and obtain the steady-state solution.
2. Choose various grid sizes of your choice, and demonstrate grid-independence of the results. You will need at least three grid sizes for establishing grid independence.
3. Plot the *x*-velocity, *y*-velocity, and pressure fields as contous plots for the finest grid case.
4. Plot the variation of *x*-velocity, *y*-velocity, and pressure along the $x = 0.5$ and $y = 0.5$ centerlines for the finest grid case.
5. Show the velocity streamlines for the various grid sizes you have considered. Discuss the effects of grid refinement on the streamlines.