Department of Mechanical Engineering
Indian Institute of Technology Bombay

ME704: Computational Methods in TFE **Autumn 2022** **Instructor**: Prof. Atul Sharma

Term-Project: Stream-function Vorticity Method-based Solution of 2D Incompressible Flow

Date Posted: 19th Oct. (Wednesday) **Due Date: 2nd Nov. (Wednesday, Early Morning 2 AM)**

VIVA on: 2nd November, Wednesday, 04:00-07:00 PM, Room No. F-24, Mech. Engg. Dept.

ONLINE SUBMISSION THROUGH MOODLE ONLY (No late submission allowed): Create a single zipped file consisting on (a) filled-in answer sheet (this doc file—converted into a pdf file), and (b) all the computer programs. The name of the zipped file should be **rollnumber_TP**

NOTE: There will be no extension on the date of the SUBMISSION & VIVA – you need to get started at the earliest to avoid the struggle to finish within the deadline of 2nd November.

WITH BEST WISHES!

1. **CFD code-development and testing of the Flow Solver for a LDC Flow:** Lid driven cavity flow probably is the most commonly used problem for testing of an in-house flow solver. This is shown in Fig. 1, as a square cavity with the left, right and bottom wall as stationary. The top wall, called here as the lid, acts like a long conveyor-belt and is moving horizontally with a constant velocity U_0. The motion results in a lid driven recirculating flow inside the cavity. The cavity is represented by a closed 2D Cartesian square domain of size $L_1=L_2$, with all the boundaries as the solid-walls. Figure 1 also shows the boundary conditions for the non-dimensional computational set-up of the problem. The simplicity in the shape of the domain and the boundary conditions has led to the wide application of the LDC flow as the benchmark problem for a flow solver.

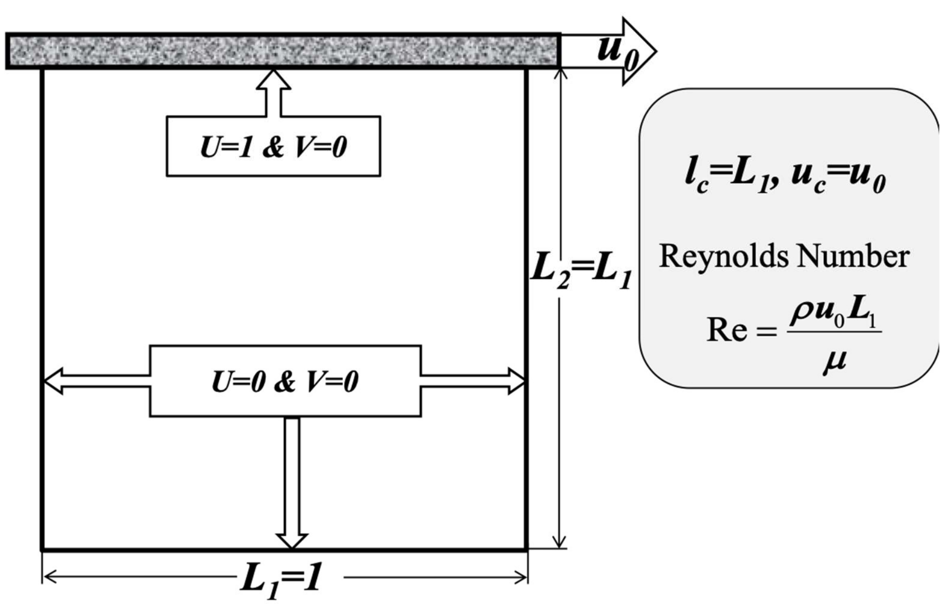


Fig. 1: Computational set-up (domain and boundary conditions) for the lid driven cavity flow.

Using the **stream function vorticity method** of the CFD development presented in the class for the FOU scheme, develop a computer program for the 2D steady flow solver on a *uniform* grid. The code should be written in a **non-dimensional** form, with the length of the cavity $\boldsymbol{L_1=1}$ as the characteristic length, and the lid velocity $\boldsymbol{U_0=1}$ as the characteristic velocity scale. They are shown in Fig. 1, along with the governing-parameter for the isothermal flow as Reynolds number $\boldsymbol{Re=\rho U_0 L_1/\mu}$. The Re is implemented in the code by a computational set-up as $\rho=U_0=\boldsymbol{L_2}=1$ and $\mu=1/Re$. You may use the developed and tested codes in the previous chapters (for steady state conduction and convection heat transfer) as the generic subroutines in the present code-development for the flow solver. After the CFD development, run the code with a convergence tolerance of $\varepsilon_{st}=10^{-3}$; and present the following results:

a) **Steady-State flow-patterns:** For a Reynolds number of 100 and a grid size of 51×51, present and discuss a figure for the (i) velocity-vector, (ii) streamlines (stream-function contours), and (iii) vorticity contours in the flow domain (3 figures).

b) **Grid-independence and code-verification study:** For the Reynolds number of 100 and four uniform grid sizes (11×11, 31×31, and 51×51), draw an overlap plot of the results (on all the grid sizes) as follows (1 figure):

   a) Variation of U-velocity along the vertical centerline.

   b) Variation of V-velocity along the horizontal centerline.

   Discuss on the variation in the results with the change in the grid size. Along with the results for the grid independence study, overlap the benchmark results reported by Ghia et al. (1982) (find attached files: ghia_data_Y&U_Re100.dat & ghia_data_X&V_Re100.dat, with first column as the coordinate and the second column as the velocity); and discuss the accuracy of the present results (on the grid size of 51×51) as compared to the benchmark results.

2. **CFD Application and Analysis for the Allotted-Problem:** After modifying the computational set-up (domain and boundary conditions) for the allotted-problem, run the code with a convergence tolerance of $\varepsilon_{st}=10^{-3}$; and present the results for the steady state flow patterns (as mentioned above for the previous problem) at a constant value of *input* governing parameters (such as Re) and grid-size as marked (by red color) in the slide for your problem. Also, discuss the flow patterns.

References

1. **Sharma A., (2017)**, *Introduction to Computational Fluid Dynamics: Development, Application and Analysis*, Ane Books Pvt. Ltd., New Delhi, Chapter 9, pp. 302-306.
2. **Ghia U., Ghia K. N., and Shin C. T. (1982)**. High-Re solutions for incompressible flow using the Navier-Stokes equations and a multigrid method, *J. Comp. Phys.*, vol. 48, pp. 387-411.

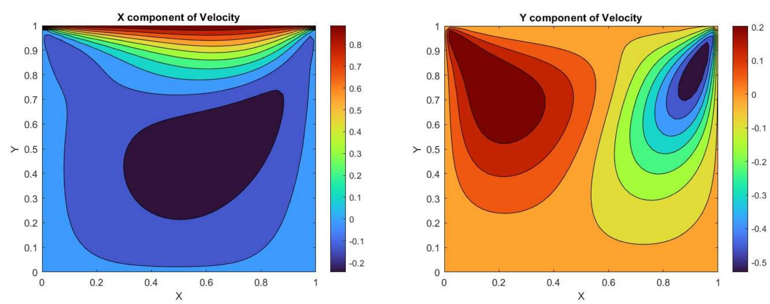
BEST OF LUCK

Keep Playing with the Codes in Future also.

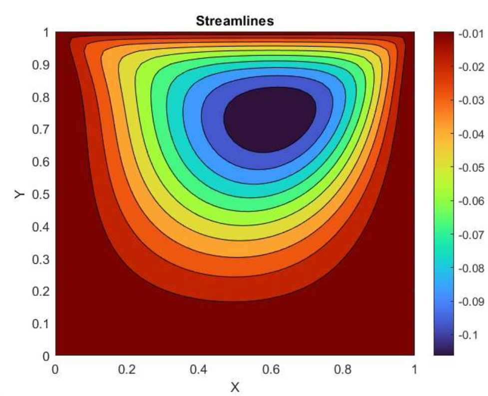
Answer Sheet

Problem # 1: CFD code-development and testing of the Flow Solver for a LDC Flow:

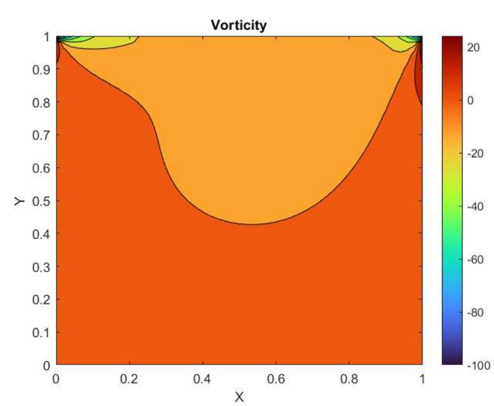
1. **Steady-State flow-patterns:** For a Reynolds number of 100 and a grid size of *51×51*, present and discuss a figure for the velocity-vector, U-velocity contour, and pressure-contour in the flow domain (3 figures).



(a)



(b)



(c)

Fig. 1: Steady state results obtained for a lid-driven cavity, on a grid size of *51×51,* at Re=100: (a) velocity-vector, (b) streamlines (stream-function contours), and (c) vorticity contours.

a) Plot and discuss the variation of U-velocity along the vertical and V-velocity along the horizontal centerline of the cavity and its comparison with the benchmark results, on a grid size of 11×11, 31×31, and 51×51 at $Re=100$. Overlap the results obtained by Ghia et al. (1982), with symbols for published results and line for present results.

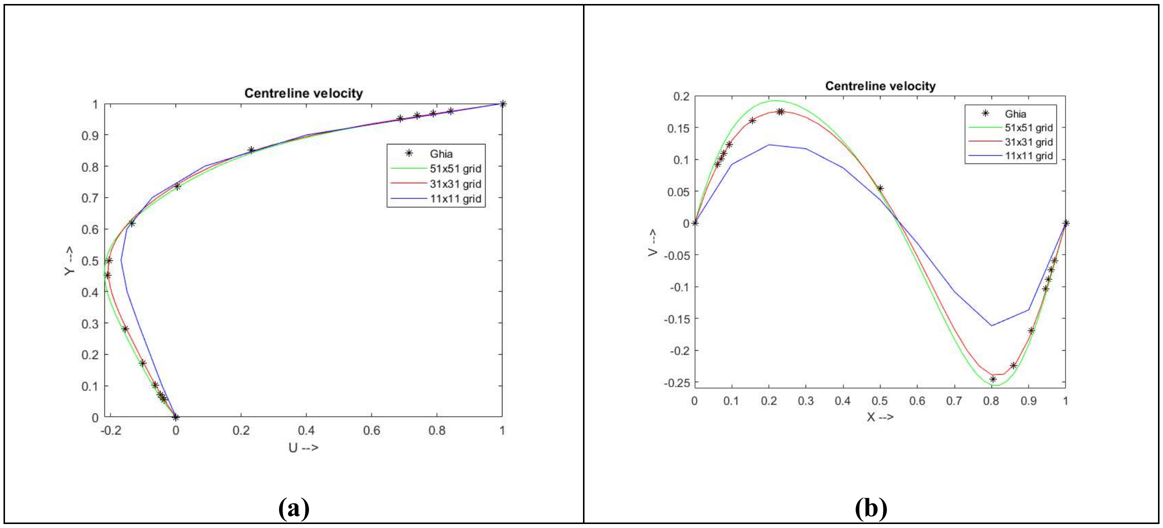


Fig. 2: Steady state results obtained for a lid-driven cavity at Re=100: (a) U-velocity along the vertical centerline (U-velocity in x-axis and y-coordinate in y-axis), (b) V-velocity along the horizontal centerline (x-coordinate in x-axis and V-velocity in y-axis); considering the three different uniform grid sizes. Here, the symbols and lines represent published and present results, respectively; similar to that shown in the lecture slide#10.28.

Discuss **Figs. 1** and **2** here, limited inside this text box only

The u and v velocity contours have been plotted for the grid size of 51x51. Similarly, the streamline and the vorticity contours have been plotted.

Streamlines give the idea how the flow is moving. In the present case rotation inside the Lid driven cavity can be visualized through the streamlines. There is no flow across the streamlines as the velocity vector at any point is tangential to the streamlines.

u velocity along the vertical centerline and v velocity along the horizontal centerline have been plotted for various grid size.

The result with 11x11, 31x31 and 51x51 grid are also plotted. It can be seen that for 51x51 grid the result has converged very nicely to the benchmark results (Ghia) for Re = 100. Thus it also shows in a way grid independence study for the given flow condition with Re = 100.

Problem # 2: CFD Application and Analysis for the Allotted-Problem (as per allotment-list):

Steady-State flow-patterns: For a input governing parameters and grid size, marked by red color in the slide for your problem, present and discuss a figure for the velocity-vector, streamlines (stream-function contours), and vorticity contours in the flow domain (3 figures).

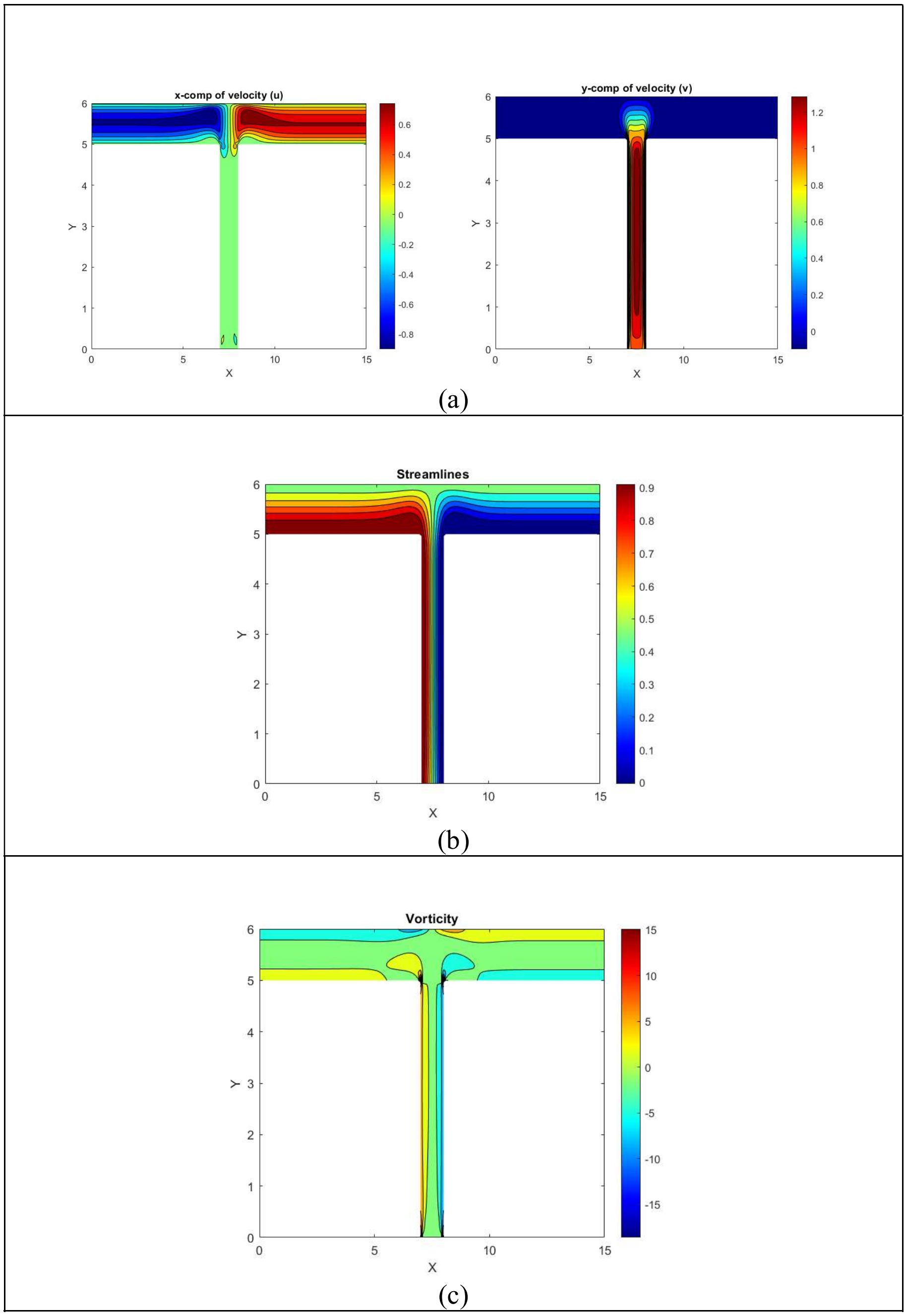


Fig. 3: Steady state results obtained for the Re and grid-size as mentioned for your problem: (a) velocity-vector, (b) streamlines (stream-function contours), and (c) vorticity contours.

Discuss **Fig. 3** here, limited inside this text box only

Fig.3(a) show the x component of the velocity which is nearly zero in the vertical channel. As the fluid turns at the junction in the horizontal channels. u component of velocity can be seen to develop. Initially the flow seems to develop and thus the u comp of velocity varies with x. By the time flow moves out through the outlet the u velocity is not changing with x signifying the flow is developed.

Similarly, in the vertical channel the v velocity is predominant but as the fluid moves into the horizontal channel, the v velocity vanishes.

Also from the conservation of mass the inlet mass is equally divided into the outlet channel. The contour plots are symmetric about the vertical centerline thus consistent with flow physics.

The velocity at the inlet is uniform which develops in the vertical channel initially and then in the horizontal channel.

Due to the no slip boundary condition it can be seen that the u and v velocity are zero at the walls.

Again the streamlines provide a visual understanding of how the flow is moving.