Università di Napoli "Federico II"

Fluidodinamica Numerica

Ingegneria Aerospaziale

Driven Cavity flow in slender cavities with unsteady lid velocity

Author:
Lorenzo Fariello

Supervisor:
Prof. G. Coppola

Academic Year 2022-2023

Abstract

This report investigates the behavior of an incompressible viscous flow in a slender rectangular cavity with an unsteady specification of the tangential velocity of the upper lid. The approach adopted for the implementation and integration of the set of Incompressible 2D Navier Stokes equations is the Psi-Zita model, where 'Psi' is the stream function and 'Zita' is the vorticity. In order to study the same problem for different kinds of rectangular geometry of the cavity, the aspect ratio of the rectangle is easily tunable. Furthermore, the tangential lid velocity is assumed to be unsteady with the possibility of choice between sinusoidal or steady increasing from a minimum negative to a maximum positive.

1 Introduction

In this report we discuss how to discretize and solve the incompressible bidimensional Navier-Stokes equation using the vorticity-stream function approach. Our final goal is to simulate the driven cavity flow in a rectangular slender cavity which is produced from the tangential unsteady movement of the upper lid. In order to describe the evolution of a generic viscous fluid into a domain we need to discuss the mathematic model of Navier-Stokes equations. First of all, the fluid dynamic is described by the conservation equations for mass, momentum and energy. If we hypothesize that a generic particle of the fluid can neither vary its density during temporal evolution, nor can move in different portion of the space characterized by a varying density gradient, we can choose to study the incompressible form of the Navier-Stokes equation. This is a simplified mathematic model that is valid only if physically the fluid flows at very low Mach numbers (low-speed), because the heat transfer or significant propriety variations of the fluid are not present. In this model, we consider this hypothesis to be true, so, from the set of complete N-S equations we will not take in consideration the energy conservation equation. In addition, the continuity equation will turn into a constraint equation for the in-divergence of the flow velocity field, which will simplify also the momentum equation.

By manipulating the momentum equation with divergence and rotor operators, we can obtain two equations: the pressure evolution equation and the vorticity evolution equation. By the way, in this report, we are not interested in a primitive variable approach (velocity components and pressure), so we can apply a change of variables to the vorticity evolution equation in order to obtain, with the addition of bidimensional hypothesis, the vorticity transport equation. In order to complete the model, we have to add the stream function which give to us, after some analytic passages, the Poisson equation. So, as a result of the change of variables, we have been able to separate the mixed elliptic-parabolic 2D incompressible Navier-Stokes equation into one parabolic equation (the vorticity transport equation) and one elliptic equation (the Poisson equation).

The ultimate goal is to discretize and integrate these equations in a time marching procedure, adopting specific boundary conditions, in order to view the time evolution of flow field. It will also be noted that the variation of the velocity field has an instantaneous influence on the entire pressure field, being the model consisting of elliptic equations.

2 Numerical method

2.1 Mathematics

The governing equations for an incompressible fluid flow are:

$$\nabla \cdot \mathbf{V} = 0 \tag{1}$$

$$\partial \frac{\mathbf{V}}{\partial t} + \mathbf{V} \cdot \nabla \mathbf{V} + \nabla \mathrm{p} = \left(\frac{1}{\mathrm{Re}}\right) \nabla^2 \mathbf{V} \tag{2}$$

The vector **V** is the velocity, in 2D case are included in it the two velocity components (u,v) (in 3D there w is added), while p is the kinematic pressure.

By writing (2) in the Crocco's form and applying the rotor operator we obtain the vectorial vorticity transport equation:

$$\mathbf{w} = \nabla \mathrm{x} \mathbf{V} \tag{3}$$

$$\partial \frac{\mathbf{w}}{\partial t} + \mathbf{V} \cdot \nabla \mathbf{w} - \mathbf{w} \cdot \nabla \mathbf{V} = \left(\frac{1}{\mathrm{Re}}\right) \nabla^2 \mathbf{w} \tag{4}$$

Applying the 2D hypothesis to the equation, the vorticity vector (**w)** turns into the scalar Zita (ζ), and the third term, or the stretching term, become zero. From here we are able to obtain the vorticity transport equation:

$$\partial \frac{\zeta}{\partial t} + \mathbf{V} \cdot \nabla \zeta = \left(\frac{1}{\mathrm{Re}}\right) \nabla^2 \zeta \qquad (5)$$

In order to complete the model, we introduce the stream-function Psi (ψ) which is defined by the equations:

$$\partial \frac{\psi}{\partial y} = \mathrm{u} \qquad (6)$$

$$\partial \frac{\psi}{\partial x} = -\mathrm{v} \qquad (7)$$

The stream function is so called because the lines where this function is constant are the streamlines. These streamlines are, in every point, tangent to the velocity vector (**v**). Along a streamline the normal velocity is zero, so the mass flow through a current line is zero (these properties will be used in the code for the implementation of the boundary conditions).
By substituting the equations (6) and (7) into the (3) we can obtain the Poisson equation:

$$\nabla^2 \psi = \zeta \qquad (8)$$

2.2 Numerical method

2.2.1 Boundary conditions

The equations (5) and (8) are discretized on a collocate mesh, this provides that all the variables are collocated on the nodes of the mesh grid. The boundary conditions provide a constant value equal to zero for the stream function for all the four walls. In fact, as discussed before, the walls are considered streamlines so that there can be neither inflow nor outflow. On the other hand, for the vorticity, the boundary conditions are obtained from the Thom's formula which estimates the production of wall vorticity. Mathematically Thom's formula is a Taylor's series development of the stream function. The only alteration of the Taylor formula is due to the upper surface (North boundary), because is the only wall which has an unsteady velocity different from zero. The boundary conditions formulas are:

$$\psi = 0 \tag{9}$$

$$\zeta w = 2 * \frac{\psi i - \psi w + uw * h}{h^2} \tag{10}$$

where the subscript 'i' stand for internal point and the subscript 'w' stand for wall point. For the other three walls (south, west, east) the equation (10) is the same with the modification for the wall speed (u_w) that is zero.

2.2.2 Convective term discretization

Before introducing the time integration of the equation (5), in this subsection we will discuss the best way in order to discretize the convective term. There are several ways for the discretization of the convective term such as the advective form, the divergence and the skew-symmetric. All of them have the propriety of preserving the linear invariant, in our case, the mass and momentum integral, if a central scheme is chosen for the discretization. By the way, the first two forms (advective and divergence) are not able to preserve either the quadratic invariant or enstrophy which is the square of the vorticity, and is correlated to dissipation effects in the fluid. In order to preserve also the quadratic invariant, we can use the skew-symmetric form, that is obtained by the average value of the other two forms.

2.2.3 Equation integration and system resolution

The time-marching procedure adopted for the resolution of the vorticity transport equation (5) is the fourth order Runge-Kutta, which is one of the most popular procedures for the resolution of ordinary differential equations. Once the vorticity transport equation is integrated, the problem is solved with the calculation of the stream function obtained from the Poisson equation (8) with a direct method of resolution, applying in MATLAB the "Backsplash". After this step, the system composed of the two equations (5) and (8) is solved and we can finally visualize the plot generated by MATLAB.

3 Plot analysis

The final plot shows the temporal evolution of the flow field. We can distinguish two types of vortices identified by two different colors:

- The red one identifies the counterclockwise vortex, that is generated by a negative (left-hand movement) of the lid.
- The black one identifies the clockwise vortex, that vice versa, is generated by a positive (right-hand movement) of the lid.

The flow field background shows, thanks to colormap MATLAB's native command, how the pressure field is simultaneously affected by the unsteady value of the lid velocity. In the Figure (1) and (3) we can see some examples of the final simulation time plots, generated respectively by a linear varying lid velocity figure (2) and a sinusoidal varying lid velocity figure (4).

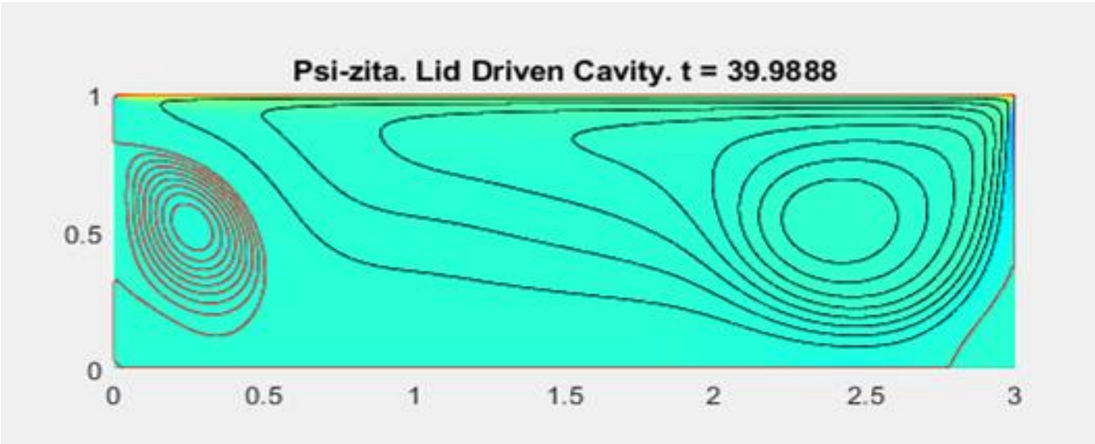

Figure 1 - Flow field generated by an unsteady linear varying lid velocity

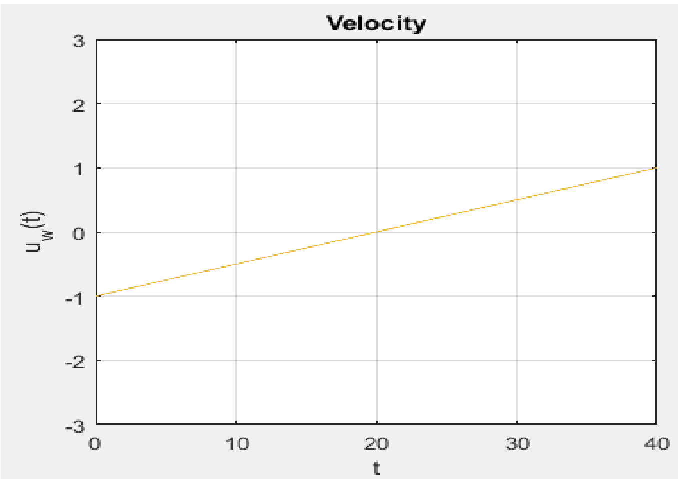

Figure 2 - Linear varying lid velocity from a negative minimum to a positive maximum

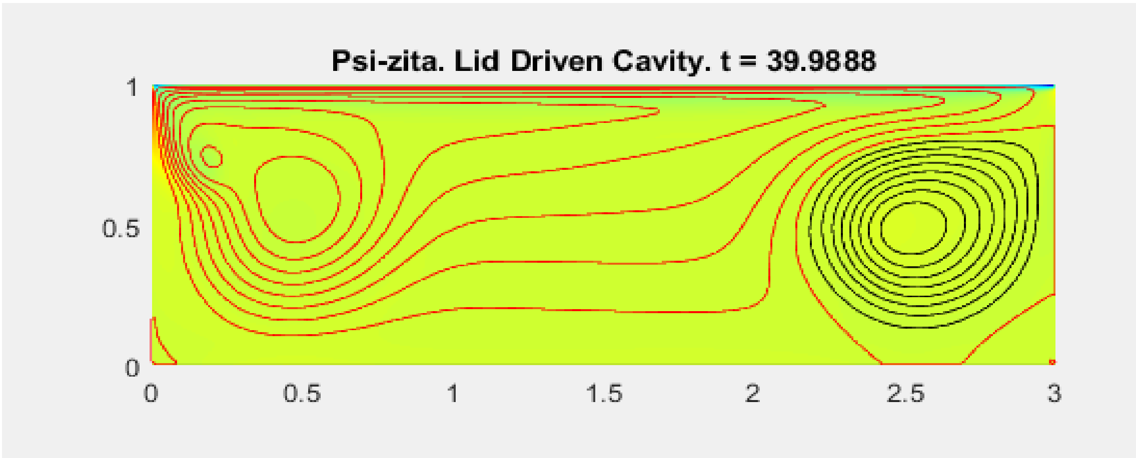

Figure 3 - Flow field generated by an unsteady sinusoidal lid velocity

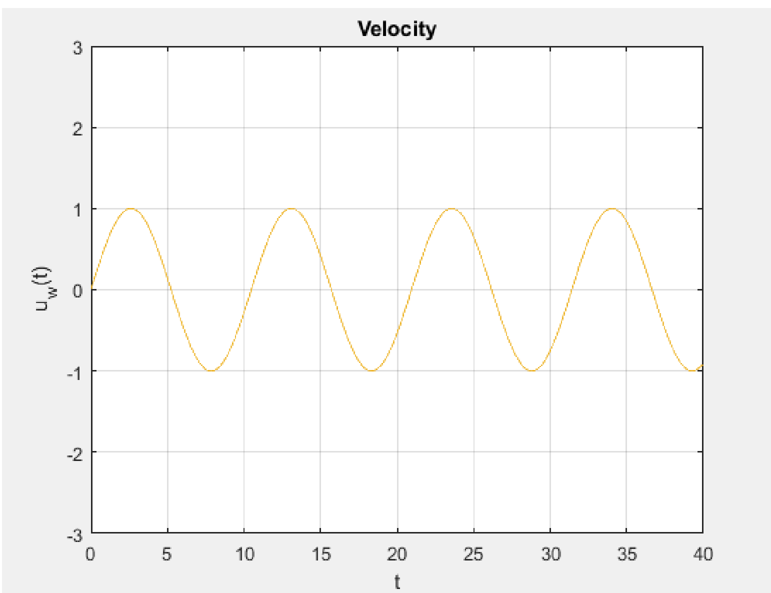

Figure 4 - Sinusoidal varying lid velocity

Conclusions

In this report we have analyzed the numerical discretization model for a physical fluid-dynamic problem, of a viscous incompressible fluid inside a slender rectangular cavity with an unsteady lid velocity. The physic behind this problem is strictly related to the shear stress of a viscous fluid which tend to follow the unsteady movement of the lid. In fact, the lid motion tends to carry the adjacent particles of fluid with it, this generates a non-uniform velocity distribution on the particles and this results in vorticity production. Depending on the geometry of the cavity, and the type of unsteady velocity chosen for the lid, we can visualize different type of the flow fields thanks to the numerical integration of the equations implemented on the MATLAB solver. In this specific case the problem has been solved using the vorticity – stream function approach because of the hypothesis of incompressible and bidimensional fluid.