NUMERICAL ANALYSIS OF LID-DRIVEN CAVITY FLOWS BY THE IMMERSED BOUNDARY METHOD USING THE VIRTUAL PHYSICAL MODEL

Tadeu Tonheiro Rodrigues, tadeu.tonheiro@gmail.com
Jônatas Ferreira Lacerda, jonataslacerda@hotmail.com
Departamento de Engenharia Mecânica, Faculdade de Engenharia de Ilha Solteira, Universidade Estadual Paulista Júlio de Mesquita Filho, 15385-000 Ilha Solteira, SP, Brazil

José Luiz Gasche, gasche@dem.feis.unesp.br
Departamento de Engenharia Mecânica, Faculdade de Engenharia de Ilha Solteira, Universidade Estadual Paulista Júlio de Mesquita Filho, 15385-000 Ilha Solteira, SP, Brazil

Abstract. The study of flows over complex geometries represents a great challenge due to the difficulty in representing the geometry. The body fitted mesh methods, like non-orthogonal and unstructured grids, are efficient to represent complex geometries and provide consistent results. Nevertheless, the complex calculations to build the mesh make these methods computationally expensive when the problem involves moving boundaries due to the re-meshing process. Another approach to solve this class of problems is the Immersed Boundary Method developed by Peskin (1977). This method is based on two distinct meshes: a fixed Eulerian mesh used to solve the fluid governing equations, and a Lagrangian mesh used to represent the immersed body. The coupling between the two meshes is made by adding a force term in the momentum equations. Using this approach the no-slip condition is imposed virtually by the force term, which makes the numerical simulation with moving boundaries less expensive. The objective of the present paper is to show results obtained from a computational code based on the Finite Volume Method developed to solve incompressible, bi-dimensional and transient flows using the Immersed Boundary Method. The Physical Virtual Model proposed by Lima e Silva et al. (2003) was chosen to calculate the force term. In order to validate the code, the lid-driven cavity was simulated for low Reynolds numbers, Re = 100 and Re = 400. The results were analyzed using dimensionless velocity profiles, maps of velocity vectors, vorticity and force field showing good agreement with literature data.

Keywords: Immersed Boundary Method, Virtual Physical Model, Finite Volume, Lid-driven cavity

1. INTRODUCTION

The constant progress of computational fluid dynamics techniques has provided the solution of a great diversity of flow and heat transfer problems. Nevertheless, the study of problems involving complex geometries is still a great challenge due to the numerical issues involving implementation and computational cost, inducing the adoption of simplified geometries to describe the computational domain.

The finite element method was the first methodology capable to treat complex geometries with the use of body-fitted meshes, whose most familiar example is the unstructured mesh. The use of body fitted meshes methods is quite efficient to represent complex geometries, which explains its extension to other methods, e.g. the Finite Volume Method, and also its application in commercial codes.

Although quite efficient, the body-fitted meshes methods are expensive to solve problems with moving boundaries, due to the complex calculations to build the mesh at each time step in order to adjust the computational domain to the new configuration of the domain.

As alternative to handle with complex geometries, the Immersed Boundary Method proposed by Peskin (1977) has become an attractive method due to a special approach, because involves two different meshes: a fixed Eulerian mesh for solving the fluid equations, and a Lagrangian mesh, that is used to describe the immersed body. The two meshes are independent, and the coupling between them is established by adding a force term in the momentum equations. This singular characteristic, where the boundary conditions are imposed virtually, allows the simulation of flow problems with moving complex geometries without re-meshing, which can save computational resources.

The main goal of this work is solve the lid-driven cavity problem in order to validate a computational code developed to be applied to general fluid flow problems, using the Immersed Boundary Method together with the Virtual Physical Model proposed by Lima e Silva (2002).

2. MATHEMATICAL FORMULATION

The two-dimensional, unsteady, incompressible and isothermal flow in a lid-driven square cavity is governed by the mass conservation and the Navier-Stokes equations, given by:

\[
\frac{\partial u_i}{\partial x_j} = 0
\]
\[ \frac{\partial u_i}{\partial t} + \frac{\partial (u_i u_j)}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] + F_i \] 

where \( \rho \) and \( \mu \) are the density and the dynamic viscosity of the fluid, respectively, \( p \) is the pressure, and \( u_i \) represents the velocity components. The Eulerian force density field, \( F_i \), in Eq. (2), models the immersed boundary, being responsible to represent the body inside the flow. The Eulerian force is calculated by distributing the Lagrangian interfacial force, \( f_i \), through the following equation:

\[ F_i(x_{ij}) = \sum_k D_{ij} \left( |x_{ij} - \bar{x}_k| \right) f_i(x_k) \frac{\Delta V_k}{\Delta V_{ij}} \] 

where \( \Delta V_k \) is the Lagrangian point volume, \( \Delta V_{ij} \) is the Eulerian point volume, \( \bar{x}_k \) is the Lagrangian point position, \( \bar{x}_{ij} \) is the Eulerian point position and \( D_{ij} \) is the distribution function with of a Gaussian function properties.

Several models to calculate the Lagrangian interfacial force, \( f_i \), have been developed (Mittal and Iaccarino, 2005). In this work, the Virtual Physical Model proposed by Lima e Silva et al. (2003) is used to calculate the Lagrangian interfacial force, which is defined as:

\[ \bar{f}(\bar{x}_i, t) = \rho \frac{\partial \bar{V}(\bar{x}_i, t)}{\partial t} + \rho \left( \bar{V} \cdot \nabla \right) \bar{V}(\bar{x}_i, t) - \mu \nabla^2 \bar{V}(\bar{x}_i, t) + \bar{V} \left( \frac{\partial p}{\partial \bar{x}_i} \right) \] 

where \( \bar{f}_a, \bar{f}_f, \bar{f}_v \), and \( \bar{f}_p \) represent, respectively, the acceleration, inertial, viscous and pressure forces (by unit volume) acting on the fluid particle at the interface. These terms are calculate by interpolating the Eulerian field data. Details about this procedure can be found in Lima e Silva et al. (2003) and Lima e Silva (2002).

3. NUMERICAL METHOD

The Finite Volume Method using a staggered grid is used to discretize the governing equations. The central differencing scheme is used for interpolating the convective-diffusive terms and the SIMPLExC (Van Doormal and Raithby, 1984) algorithm is applied to treat the pressure-velocity coupling. The three-time level second order scheme is used as time interpolation scheme. The velocity algebraic equation systems are solved iteratively using the TDMA algorithm (Thomas, 1949) and the algebraic equation system of the pressure correction is solved using the SIP - Strongly Implicit Procedure Method (Stone, 1968).

Figure 1 presents a scheme of the geometry and boundary conditions of the problem. As can be noted in Fig. 1, the cavity walls are modeled with the Immersed Boundary Method, i.e. the no-slip condition and the prescribed velocity at the top of the cavity are obtained through the Eulerian force field calculated with Eqs. (3) and (4).

The Lagrangian mesh is built with 90 points at each wall of the cavity. According to Lima e Silva (2002), the size of both Eulerian and Lagrangian meshes must be equal. As a result the interior of the cavity has 90x90 volumes and the total number of volumes for the Eulerian field is 137x137 volumes.
4. RESULTS AND DISCUSSIONS

The flow in the lid-driven square cavity was simulated for low Reynolds numbers, Re=100 and Re=400, where the Reynolds number was calculated using the top wall velocity and the cavity height. Dimensionless velocity profiles are presented in Figs 2 and 3, for Re=100 and Re=400, respectively, in comparison with the data Ghia et al. (1982) from.
One can note that the dimensionless velocity profiles present good agreement with the data from Ghia et al. (1982), demonstrating that the computational code was correctly built and, at least, partially validated. The maps of velocity vectors and vorticity field are presented in Figs. 4 and 5, for Re = 100 and Re = 400, respectively.

Figure 3. Dimensionless velocity profiles for Re = 400.

Figure 4. Maps of (a) velocity vectors and (b) vorticity for Re = 100.

Figure 5. Maps of (a) velocity vectors and (b) vorticity for Re = 400
The typical flow patterns of this type of flow can be visualized in Figs. 4 and 5: large vortex moving to the center of the cavity for increasing Reynolds and small secondary vortex in the lower right corner for larger Reynolds numbers (Re = 400 in this case).

In addition, one can observe that the force term added in the Navier-Stokes equations, in order to model the walls of the cavity, also produces an external flow in the opposite direction of the internal flow.

According to Mohf-Yusof (1997), this is a peculiar result always obtained when the Immersed Boundary Method is applied. As the entire domain is composed by the fluid, the forcing term has to produce a velocity field near the virtual walls in order to satisfy the prescribed velocities at the boundaries.

One important parameter used to evaluate if the virtual boundary conditions are satisfied is the L2 norm, defined as:

$$L_2 = \frac{\sqrt{\sum (u_k - u_{jk})^2}}{n}$$

where $u_k$ is the prescribed velocity at the immersed boundary, $u_{jk}$ is the velocity of the fluid at the interface and $n$ is the number of Lagrangian points.

Figure 6 presents the temporal evolution of the L2 for the simulated cases. As can observed, after the first time steps a relatively constant large values for L2 norm are reached. A similar behavior was observed by Arruda (2004) for the simulation of the lid-driven square cavity using the Virtual Physical Model. Even for these L2 norm values, the results for the velocity fields are in agreement with the literature data.

Figure 7 presents the Eulerian force field for the both Reynolds numbers.
The Eulerian force fields act as source or sink of momentum, in order to define the immersed body. This characteristic is easily observed in Figs. 7.a and 7.b, where the force field values at the top of the immersed body are larger, acting as a source of momentum, for modeling the prescribed velocity at the top of the cavity. It can also be noted that the Eulerian force field values increase for increasing Reynolds numbers, as expected.

5. CONCLUSIONS

In the present paper a computation code based on the Finite Volume Method together with the Immersed Boundary Method and the Virtual Physical Model was validated through the solution of the lid-driven cavity for two Reynolds numbers, Re = 100 and Re = 400. The good agreement with the data from Ghia et al. (1982) shows that the computational code was built correctly and can be tested for other types of problems in order to be further validated.

6. REFERENCES


7. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.