Solution of Navier-Stokes equations for fluids with magnetorheological compensation used in structures with energy dissipaters

N Balaguera Medina¹, M A Atuesta¹, O A Nieto², and P A Ospina Henao²
¹ Facultad de Ingeniería Civil, Universidad Santo Tomás, Bucaramanga, Colombia
² Departamento de Ciencias Básicas, Universidad Santo Tomás, Bucaramanga, Colombia

E-mail: paolo.ospina@ustabuca.edu.co

Abstract. The fixed-wall rectangular cavity flow problem is a classic problem that has been studied since the beginning of computational fluid mechanics. The present work aims to provide a numerical and computational solution of the Navier-Stokes equations using the finite difference method, applied to model the problem of a magnetorheological fluid in a rectangular cavity with a fixed wall in shock absorbers devices, used in civil structures that use energy dissipaters.

1. Introduction
Much of the Colombian population lives in an area formed by tectonic plates, under the threat of earthquakes of all degrees. The constructions in seismic zones increase day by day in our country, consequently, the structural security has an important value in the national development. The reduction of costs, with the consequent safety of the works in seismic zones is the central problem of construction in our country. The fundamental reason for this problem is the development of seismic calculation methodologies for buildings with passive energy dissipators [1].

In other countries, conventional energy dissipation techniques have been complemented with additional systems to the structural components of the building, which modify the dynamic characteristics of the structure, controlling or dissipating part of the energy imposed by the earthquake [2]. The use of these seismic response control techniques aims to reduce the seismic demand of the structure itself by controlling its deformation and, therefore, its damage [3].

A viscous dissipator consists of two main chambers separated by a small hole between them, one of the chambers contains a viscous fluid similar to an oil that is pushed by a piston towards the opposite chamber and forced to pass through the small hole; the resistance exerted by the viscous fluid when entering generates a force against the thrust movement of the piston, which allows stability in the mechanism. This type of dissipators are placed in specific places in the structure, generally in places where the elements are subjected to greater dynamic stress. The fluid inside the chamber must have a high relative viscosity, which must be higher as the stresses to which the system will be subjected increases [4].

The function of this type of dissipators is to generate forces contrary to the movement of the building, which are generated by sinusoidal seismic waves, all this without altering the center of
mass of the structure or its structural shape. The ideal for the behavior of a viscous dissipator is that it complies with the minimum shear at the base and the distortion between floors, this in order to act appropriately and not against the building in case of a very strong disturbance in the structure [4].

The article is distributed as follows; in the section 2, we present the Navier-Stokes equations and the numerical solution for the magnetorheological fluid present in the damper using the finite difference method. In section 3, the results of the simulation for the cavity and channel flow, respectively, as well as the simulation for the velocity field for the Reynolds number, are presented; finally, the conclusions are presented in section 4.

2. Navier-Stokes equations for the magnetorheological fluid

In Figure 1, the parts of the shock absorber that contain the magnetorheological fluid and that will be housed in civil structures are shown.

In the present work we limit ourselves to two-dimensional fluid flows; therefore, the approximation is made where the velocities and gradients of the fluid in the direction other than the plane are neglected; in the same way, we study incompressible Newtonian and isothermal fluids [5]. The Navier-Stokes equations arise from the application of Newton’s second law to the motion of a fluid, in our particular case, the fluid contained in a magnetorheological damper complies with the general form of these equations. The set of equations Equation (1), describe the vector expression of the Navier Stokes equations and the velocity gradient equation, which are written as [6].

\[
\rho \left( \frac{\partial \vec{v}}{\partial t} + \vec{v} \cdot \nabla \vec{v} \right) = -\nabla p + \mu \nabla^2 \vec{v} + \vec{f}, \quad \frac{\partial u}{\partial x} + \frac{\partial u}{\partial y} = 0,
\]

respectively, and the momentum equation in the \( x \) and \( y \) axis, take the following form, as expressed in the set Equations (2).

\[
\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = -\frac{1}{\rho} \frac{\partial p}{\partial x} + \eta \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right),
\]

\[
\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} = -\frac{1}{\rho} \frac{\partial p}{\partial y} + \eta \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right),
\]

where \( u = u(x,y,t) \) and \( v = v(x,y,t) \) are the instantaneous fluid velocities in the \( x \) and \( y \) directions respectively, \( p = p(x,y,t) \) is the instantaneous pressure, \( \rho \) is the magnetorheological
fluid density and \( \mu \) is the viscosity. Using the Equation (1) as a restriction for each moment Equation (2), obtained the Equation (3), the pressure-Poisson equation.

\[
\frac{\partial^2 p}{\partial x^2} + \frac{\partial^2 p}{\partial y^2} = -\rho \left[ \frac{\partial}{\partial t} \left( \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} \right) + \left( \frac{\partial u}{\partial x} \right)^2 + 2 \frac{\partial u}{\partial y} \frac{\partial v}{\partial x} + \left( \frac{\partial v}{\partial y} \right)^2 \right].
\]  

(3)

Therefore, it is a derived equation that relates the pressure to the moment equation which is similar to the Poisson equation \( \nabla^2 p = -f \).

2.1. Finite difference method

The discretization of the system of Equations (2) are [6] and [7], using the finite difference method allows writing the components of the fluid velocity field \( u \) and \( v \), such as Equation (4) and Equation (5).

\[
\frac{u_{ij}^{n+1} - u_{ij}^n}{\Delta t} + u_{ij}^n \left( \frac{u_{ij}^n - u_{i,j-1}^n}{\Delta x} \right) + v_{ij}^n \left( \frac{u_{ij}^n - u_{i,j-1}^n}{\Delta y} \right) = -\frac{1}{\rho} \left( \frac{p_{i+1,j}^n - p_{i-1,j}^n}{2\Delta x} \right) + \eta \left( \frac{u_{i+1,j}^n - 2u_{ij}^n + u_{i,j-1}^n}{\Delta x^2} + \frac{u_{i,j+1}^n - 2u_{ij}^n + u_{i,j-1}^n}{\Delta y^2} \right),
\]

(4)

\[
\frac{v_{ij}^{n+1} - v_{ij}^n}{\Delta t} + u_{ij}^n \left( \frac{v_{ij}^n - v_{i,j-1}^n}{\Delta x} \right) + v_{ij}^n \left( \frac{v_{ij}^n - v_{i,j-1}^n}{\Delta y} \right) = -\frac{1}{\rho} \left( \frac{p_{i+1,j}^n - p_{i-1,j}^n}{2\Delta y} \right) + \eta \left( \frac{v_{i+1,j}^n - 2v_{ij}^n + v_{i,j-1}^n}{\Delta x^2} + \frac{v_{i,j+1}^n - 2v_{ij}^n + v_{i,j-1}^n}{\Delta y^2} \right),
\]

(5)

while the discretized Equation (3), the pressure take the form like the Equation (6).

\[
\frac{p_{i+1,j}^n - 2p_{ij}^n + p_{i-1,j}^n}{\Delta x^2} + \frac{p_{i,j+1}^n - 2p_{ij}^n + p_{i,j-1}^n}{\Delta y^2} = -\rho \left[ \left( \frac{1}{\Delta t} \right) \left( \frac{u_{i+1,j}^n - u_{i-1,j}^n}{2\Delta x} + \frac{v_{i,j+1}^n - v_{i,j-1}^n}{2\Delta y} \right) \right]
\]

\[
+ \left( \frac{u_{i+1,j}^n - u_{i,j}^n}{2\Delta x} \right)^2 + 2 \left( \frac{u_{i,j+1}^n - u_{i,j-1}^n}{2\Delta y} \right) \left( \frac{v_{i+1,j}^n - v_{i,j}^n}{2\Delta x} \right) \left( \frac{v_{i,j+1}^n - v_{i,j-1}^n}{2\Delta y} \right)^2,
\]

(6)

where a second order central difference is used for the left hand side and a first order central difference for the right hand side of the equation. Finally, the components of velocity and pressure are found by transposing the terms [8], obtaining Equation (7), Equation (8) and Equation (9).

\[
u_{ij}^{n+1} = u_{ij}^n - u_{ij}^n \left( \frac{u_{ij}^n - u_{i,j-1}^n}{\Delta x} \right) - v_{ij}^n \left( \frac{u_{ij}^n - u_{i,j-1}^n}{\Delta y} \right) - \frac{1}{\rho} \left( p_{i+1,j}^n - p_{i-1,j}^n \right) \left( \frac{\Delta t}{\Delta x} \right) + \eta \left[ \left( u_{i+1,j}^n - 2u_{ij}^n + u_{i,j-1}^n \right) \left( \frac{\Delta t}{\Delta x^2} \right) + \left( u_{i,j+1}^n - 2u_{ij}^n + u_{i,j-1}^n \right) \left( \frac{\Delta t}{\Delta y^2} \right) \right],
\]

(7)

\[
v_{ij}^{n+1} = v_{ij}^n - v_{ij}^n \left( \frac{v_{ij}^n - v_{i,j-1}^n}{\Delta x} \right) - v_{ij}^n \left( \frac{v_{ij}^n - v_{i,j-1}^n}{\Delta y} \right) - \frac{1}{\rho} \left( p_{i+1,j}^n - p_{i,j-1}^n \right) \left( \frac{\Delta t}{\Delta y} \right) + \eta \left[ \left( v_{i+1,j}^n - 2v_{ij}^n + v_{i,j-1}^n \right) \left( \frac{\Delta t}{\Delta x^2} \right) + \left( v_{i,j+1}^n - 2v_{ij}^n + v_{i,j-1}^n \right) \left( \frac{\Delta t}{\Delta y^2} \right) \right],
\]

(8)
\[ p_{ij}^n = \left[ \frac{(p_{i+1,j}^n + p_{i-1,j}^n)(\Delta y^2) + (p_{i,j+1}^n + p_{i,j-1}^n)(\Delta x^2)}{2(\Delta x^2 + \Delta y^2)} \right] + \left( \frac{\rho \Delta x^2 \Delta y^2}{2 \Delta t (\Delta x^2 + \Delta y^2)} \right) \left( \frac{u_{i+1,j}^n - u_{i-1,j}^n}{2 \Delta x} + \frac{v_{i,j+1}^n - v_{i,j-1}^n}{2 \Delta y} \right) + \left( \frac{\rho \Delta x^2 \Delta y^2}{2 (\Delta x^2 + \Delta y^2)} \right) \left( \frac{u_{i+1,j}^n - u_{i-1,j}^n}{2 \Delta x} \right)^2 + \left( \frac{\rho \Delta x^2 \Delta y^2}{2 \Delta y^2} \right) \left( \frac{v_{i,j+1}^n - v_{i,j-1}^n}{2 \Delta x} \right) + \left( \frac{v_{i,j+1}^n - v_{i,j-1}^n}{2 \Delta y} \right)^2 \] \tag{9}

3. Numerical solution

For the development of the numerical solution, a code was implemented for rectangular geometry of cavity flow and channel flow, where the real variables of the damper with magnetorheological fluid compensation were implemented, such as particle mass \( m = 1.43 \times 10^{-8} \) (kg); fluid density \( \rho = 872 \) (kg/m\(^3\)) and dynamic viscosity \( \mu = 7.29 \times 10^{-2} \) (kg/m s) [9].

3.1. Cavity flow

The cavity flow problem is a very common problem for incompressible flow solvers, mainly due to its simplicity. The problem configuration is a square domain with walls at each boundary where the upper wall moves at a constant tangential velocity, see Figure 2. Although the cavity flow problem is common, an obvious problem is the physical singularities at each of the upper corners [10].

The problem of corner singularities can be completely avoided by modifying the tangential velocity at the upper limit so that the speed at the corners is zero. Here, the upper limit condition for the velocity component is equal to 1, thus any discontinuity introduced is small enough to be negated by the dissipative properties of the system. It can be see in the Figure 3, that in this case the vortices in the lower corners are not appreciated, it is because the lower viscosity makes the movement of some particles and others more independent, and the restrictions imposed by the boundary and others are not so decisive in the fluid flow at the top over the bottom.

In the Figure 4, we see that due to the stationary motion can not be reached, but, for more time in the simulation and a finer mesh to increase the approximation, the vortices would be clearly visible. Finally, in the Figure 5, the important result is underlined that the difference in fluid pressures depends exclusively on the difference in depths of the same in the shock absorber and, obviously, on the density of the fluid.

![Figure 2. Boundary conditions cavity flow.](image1)

![Figure 3. Contour velocity u.](image2)
3.2. Channel flow

The boundary conditions propose periodic solutions at the ends of the mesh and allowing a pipe region with zero velocity profiles to transition to a non-zero region [11], as can be seen in Figure 6. In Figure 7, it is possible to notice the changes in the velocity profiles in the walls where periodic conditions were applied and how in its middle part it reaches the stationary level.

In Figure 8, the upper right corner shows a decrease in speed while the upper left corner shows an increase in speed and the stationary regime is reached in the central region. In Figure 9, the pressure profile rises and falls due to this transition and therefore we can see a non-physical loss in these regions. In addition, the pressure results are difficult to verify since in the bibliography it is usually a data that is omitted to give preference to the velocity fields and the motion of the fluid. Flow profiles can act as inlet-outlet profiles that do not cause unrealistic spikes in pressure profiles and can also reduce the number of grid points required by a boundary.
3.3. Reynolds number
Due to the limitations described for test the pressure data for other Reynolds numbers, it is consider the incompressible Navier-Stokes equations in two space dimensions, and the incompressibility condition. The last one is incorporate by using a projection approach [12], that evolve the momentum equations neglecting the pressure, then project onto the subspace of divergence-free velocity fields. Now, we present the graphs of the evolution of the velocity field, and the isolines of the fluid current for an incomprehensible fluid with a Reynolds number \( R_e < 2300 \), in our case, we choose \( R_e = 400 \) and the evolution at \( t = 4 \) (s), but any value can be choose in that interval and the behavior remains the same trend.

It can be seen that for the case \( R_e = 400 \), the temporal evolution in the Figure 10, Figure 11, Figure 12 and Figure 13 the vortices in the lower corners are not appreciated, this is due to the following reasons: (i) the fact that the lower viscosity makes the motion of some particles more independent, (ii) the restrictions imposed by the border and (iii) the motion of the fluid in the upper part on the lower part. We can also appreciate that for this viscosity a time of (iv) is not enough for the stationary movement to be obtained, however it is possible to notice the tendency of it.

4. Conclusions
An accurate and efficient numerical procedure has been developed that can simulate many types of gravity-driven flows. Computational efficiency allowed us to handle large free surface deformations directly without entanglement of the mesh by decoupling the pressure and velocity fields, respectively, allowing us to calculate them sequentially. Although a simple mesh geometry was used, the results are satisfactory and future comparisons can be made in the case of laminar and turbulent flow simulations with more complex mesh geometries that require more rigorous validation for these complex systems.
The previous study allows us to verify the effectiveness in the design and the safety in the implementation of these energy dissipators in civil structures. The behavior of the magnetorheological fluid is well controlled when the structure is under the action of external disturbances that put its stability at risk. It also allows to achieve the reduction of the accumulated energy transferred to the highest mezzanines of the structure, thus avoiding the increase in acceleration and it is possible to reduce the value of drifts and torsion, thus leaving the structure stable and in operation after said disturbances, reduces the efforts in columns.

Structurally, one of the reasons for the non-existence of this type of systems in Colombia is due to the lack of experience of the professionals who are in charge of the designs and costs are generated when applying or implementing the use of said technological advances; making this tool unprofitable for the construction industry. Finally, as future work, a control strategy will be implemented for this type of energy dissipators to structures with $N$ degrees of freedom, as well as a numerical study of the damping equations when the structures are attacked by external disturbances of the environment.

Acknowledgments
We are grateful for the permanent support of the Vicerrectoría Académica de la Universidad Santo Tomás, Bucaramanga, Colombia.

References
[1] Lafuente C 2016 Introducción al Uso de Aisladores y Dissipadores en Estructuras (Caracas: CAF-Banco de Desarrollo de América Latina)
[2] Navarro F, Castillo C 2017 Comparación de las Respuestas Dinámicas en Estructuras con y sin Dissipadores de Energía Pasivos del Fluído Viscoso en la Zona Sísmica Cuatro (Piura: Universidad Nacional de Piura)
[3] Oviedo J, Duque M 2007 Sistemas de control de respuesta sísmica en edificaciones Revista EIA 3(6) 105
[4] Chopra Anil K 1995 Dynamics of Structures: Theory and Applications to Earthquake Engineering (New Jersey: Prentice Hall)
[5] Chorin A J, Marsden J E 2000 A Mathematical Introduction to Fluid Mechanics (New York: Springer-Verlag)
[6] Lukaszewicz G, Kalita P 2016 Navier–Stokes Equations. An Introduction with Applications (Switzerland: Springer International Publishing)
[7] Lewis R W, Nithiarasu P, Seetharamu K N 2004 Fundamentals of the Finite Element Method for Heat and Fluid Flow (Chichester: John Wiley & Sons)
[8] Arfken G B, Weber H J, Harris F E 2012 Mathematical Methods for Physicists. A Comprehensive Guide (Waltham: Elsevier Inc.)
[9] LORD Corporation 2009 RD-8040-1 and RD-8041-1 Dampers (North Carolina: LORD Corporation)
[10] Yuan S W, Finkelestein A B 1955 Laminar Pipe Flow with Injection and Suction Through a Porous Wall (Princeton University: James Forrestal Research Center)
[11] Berman A S 1953 Laminar flow in channels with porous walls J. Appl. Phys. 24 1232
[12] Rieutord M 2015 Fluid Dynamics: An Introduction (Switzerland: Springer International Publishing)