NUMERICAL SIMULATION OF THE FLOW THROUGH A COMPRESSOR VALVE MODEL USING IMMERSED BOUNDARY METHOD AND ADAPTIVE MESH REFINEMENT

Franco Barbi, frcbarbi@gmail.com
José Luiz Gasche, gasche@dem.feis.unesp.br
Thiago Andreotti; aga5thi@hotmail.com
Departamento de Engenharia Mecânica, Faculdade de Engenharia de Ilha Solteira, Universidade Estadual Paulista Júlio de Mesquita Filho, Av. Brasil, 56 – Centro, 15385-000, Ilha Solteira, SP, Brasil

Millena Martins Villar, millena.villar@gmail.com
Aristeu da Silveira Neto, aristeus@mecanica.ufu.br
Laboratório de Mecânica dos Fluidos Computacional, Universidade Federal de Uberlândia, Av. João Naves de Ávila, Bairro Santa Mônica, 38400-100, Uberlândia, MG, Brasil

Abstract. *In refrigeration compressors, the suction and discharge valves are responsible for the retention of the refrigerant fluid from the suction chamber to the cylinder and passage from the cylinder to the discharge chamber. Valve system designers seek for valves with small overall flow pressure drop in order to increase the compressor efficiency. As the opening and closing of the valves are caused by the forces produced by the refrigerant flow, the understanding of the flow through the valve is of fundamental importance in order to enhance the efficiency of the valve system. The numerical simulation of the flow is an efficient method to perform this task. Due to the complex geometry usually found in this type of valve, simplified geometries have been used to represent the valve, particularly the radial diffuser geometry. This work presents a numerical simulation of the flow through a more realistic geometric model for the suction valve. An Immersed Boundary Method is used to represent the valve geometry in two different angular positions during the opening movement of the valve. An adaptive mesh dynamically refined is used for representing the flow domain. The governing equations are solved by a projection method, using a semi-implicit second-order scheme for time integration. The systems of algebraic equations are solved by a Multigrid-Multilevel technique. Results are obtained for Reynolds of 500, for three different inclinations of the valve, 7.5, 15 and 30°. Besides providing interesting flow characteristics for these high inclination angles, the most important conclusion of this work is that the methodology showed to be suitable for studying the flow through this more realistic geometry, indicating also that it can be used in the future for studying the fluid-structure interaction between the valve reed and the flow, which is a current challenge for compressor designers.*

Keywords: *Compressor valve, Immersed Boundary, Adaptive Refinement*

1. INTRODUCTION

The valve is one of the main components of a hermetic compressor, because it controls the mass flow rate through the compressor. The opening and closing movements of the valve are governed by the pressure difference applied by the refrigerant flow over the reed, as shown in Figure 1. Therefore, it is essential to fully understand the flow through the valve in order to improve its design and to enhance the overall efficiency of the compressor.

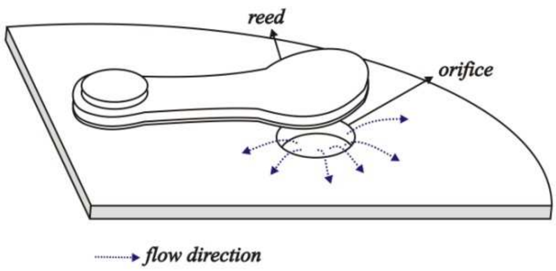


Figure 1. Scheme of the valve reed.

Several studies have been carried out to characterize the main features of the flow through the valve. Numerical solutions for incompressible laminar flows have been performed by Hayashi *et al.* (1975), Raal (1978), Piechna and Meier (1986), Ferreira *et al.* (1989), Gasche *et al.* (1992) and Possamai *et al.* (2001). Numerical solutions for incompressible turbulent flows have been accomplished by Deschamps *et al.* (1996) and Colaciti *et al.* (2007).

Experimental work on this subject has been performed by Wark and Foss (1984), Ferreira and Driessen (1986), Tabakabai and Pollard (1987), Ervin *et al.* (1989) and Gasche *et al.* (1992).

One of the main challenges for modeling this problem is the complexity of the valve geometry. Because of this reason, simpler geometries have been adopted, specially the radial diffuser. The treatment of valve reed movement is another challenge for the current computational fluid dynamics methods. The use of body-fitted meshes, where the computational mesh is set to fit to the body, introduces several computational penalties once that, for each displacement of the valve reed, the mesh must be updated for the discretization of the new computational domain. This procedure requires extensive computational resources.

The Immersed Boundary Method (IBM) developed initially by Peskin (1972) has been used successfully for solving flow problems in complex geometries presenting moving boundaries, especially for problems involving fluid-structure interaction. This method uses a simple fixed grid (Eulerian grid) for solving the flow equations and models the presence of an interface using a moving Lagrangian grid. The interface is recognized through the addition of a force field in the momentum equations.

Lacerda (2010) and Rodrigues (2010) have used the IBM to simulate the two-dimensional incompressible flow through a radial diffuser representing the valve system. Lacerda (2010) used the Virtual Physical Model introduced by Lima e Silva (2002) to calculate the Lagrangian force density in order to represent the valve seat. The results obtained by Lacerda agreed well with experimental data. Rodrigues (2010) used the same model used by Lacerda to calculate the force field, but to represent the movement of the valve reed. Rodrigues concluded that the method is able to represent well the movement of the reed and it is a promising tool for future works on fluid-structure interaction.

Although the results presented on the literature using the Immersed Boundary Method are satisfactory, the researchers have observed that the method cannot represent well thin bodies, because the mesh refinement required must be extended to the entire computational domain when uniform meshes are used. Thus, much efforts have been done in order to use the computational capacity more efficiently, and adaptive versions of the Immersed Boundary Method have been introduced.

Roma (1999) has introduced a version of the IBM and achieved enhanced the local accuracy by covering an immersed boundary with a sequence of nested, progressively finer rectangular grid patches, which follow dynamically the immersed boundary motion. The main conclusion of this work is that the solution for the mesh refined locally around the immersed boundary has no significant difference from the solution obtained in a uniform mesh, built with the resolution of the finest level.

The main motivation of this work is to simulate the flow through compressor valves considering the flow in a geometry nearest to the actual geometry of the valve system. In order to accomplish this task a three-dimensional model was used to solve the incompressible flow through a suction type valve, including the inclination of the reed. The method introduced by Wang (2008) is used to calculate the Lagrangian force density. Besides, the same refinement strategy proposed by Roma (2007) was used to build the mesh.

2. NUMERICAL METHOD

In this study a three-dimensional, unsteady, incompressible, and isothermal flow of a Newtonian fluid in cartesian coordinates is considered to solve the flow through the suction valve model. The governing equations are the mass conservation and the momentum equations, given by:

$$\vec{\nabla} \cdot \vec{u} = 0 \tag{1}$$

$$\rho\left(\frac{\partial \vec{u}}{\partial t} + \vec{u} \cdot \vec{\nabla}\vec{u}\right) = -\vec{\nabla}p + \vec{\nabla} \cdot \left[\mu\left(\vec{\nabla}\vec{u} + \nabla\vec{u}^T\right)\right] + \vec{f} \tag{2}$$

where $\vec{u}$ represents the vector velocity field, p is the pressure, and ρ and μ are the density and the viscosity, respectively. The term $\vec{f}$ corresponds to the Eulerian force density field, which models the immersed boundary, which is responsible for representing the body inside the flow. The Eulerian forces are calculated through the distribution of the Lagrangian interfacial forces, $\vec{F}$, with the following equation:

$$\vec{f}(\vec{x}) = \int_{\Gamma} \vec{F}\left(\vec{X}\right) D\left(\vec{x} - \vec{X}\right) \Delta V \tag{3}$$

where $\vec{x}$ is the position of the Eulerian point, $\vec{X}$ the position of the Lagrangian point, ΔV the discrete volume for each Lagrangian point, Γ the Lagrangian domain, and D is the distribution function having Gaussian function properties. In this work capital letters refers to the Lagrangian variables and the lowercase letters refers to the Eulerian variables.

Several models to calculate the Lagrangian interfacial force, $\vec{f}$, have been developed (Mittal and Iaccarino, 2005). In this work, the Multi-Direct Forcing proposed by Wang (2008) is used to calculate the interfacial Lagrangian force. This method iterates a direct forcing process on the Eulerian points close to the immersed boundary to guarantee the desired velocity on de boundary. The procedure involves the following steps:

- Calculus of the velocity on the Lagrangian point through the distribution function:

$$\vec{U}^{t,i} = \sum_{\Gamma} D\left(\vec{x} - \vec{X}\right)\vec{u}^{t,i}(\vec{x})\Delta V \tag{4}$$

- Calculus of the Lagrangian force density $\vec{F}$ for the iteration:

$$\vec{F}^{i}\left(\vec{X},t\right) = \rho\left(\frac{\vec{U}^{t+\Delta t} - \vec{U}^{t,i}}{\Delta t}\right) \tag{5}$$

- Distribution of the Lagrangian force density to the Eulerian points through the distribution function:

$$\vec{f}^{i}(x,t) = \sum_{\Gamma} D\left(\vec{x} - \vec{X}\right)\vec{F}^{i}\left(\vec{X},t\right)\Delta V \tag{6}$$

- Update of velocity field after forcing:

$$\vec{u}^{i+1} = \vec{u}^{i} + \frac{\Delta t}{\rho}\vec{f}^{i}(\vec{x},t) \tag{7}$$

In the second step, $\vec{U}^{t+\Delta t}$ refers to the desired velocity on the immersed boundary. These four steps are repeated until $\vec{U} = \vec{U}^{t+\Delta t}$. The force on the immersed boundary can be calculated by:

$$\vec{F}\left(\vec{X},t\right) = \sum_{i=1}^{N}\vec{F}^{i}\left(\vec{X},t\right) \tag{8}$$

where N is the number of Multi-Direct Forcing cycles.

The mesh used consists in sequences of nested, progressive finer rectangular grid patches, as exemplified by Fig. 2. The refinement strategy spots regions on the domain is based on the position of Lagrangian points and the vorticity field. For regions close to the Lagrangian points a finest level is set, while in other regions the refinement level is set considering the maximum vorticity value. The mesh dynamically adapts itself to the flow while the Immersed boundary is represented by the finest level. Figure 2 shows a body represented by the Lagrangian domain covered by the finest level of the Eulerian domain as an example.

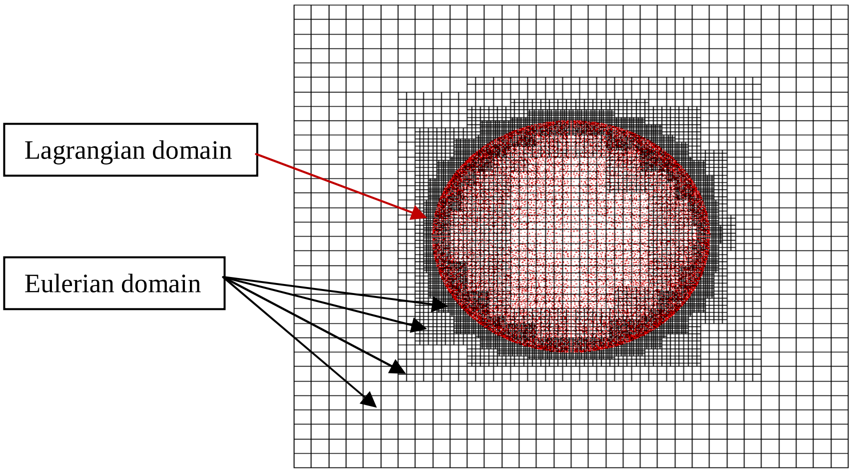
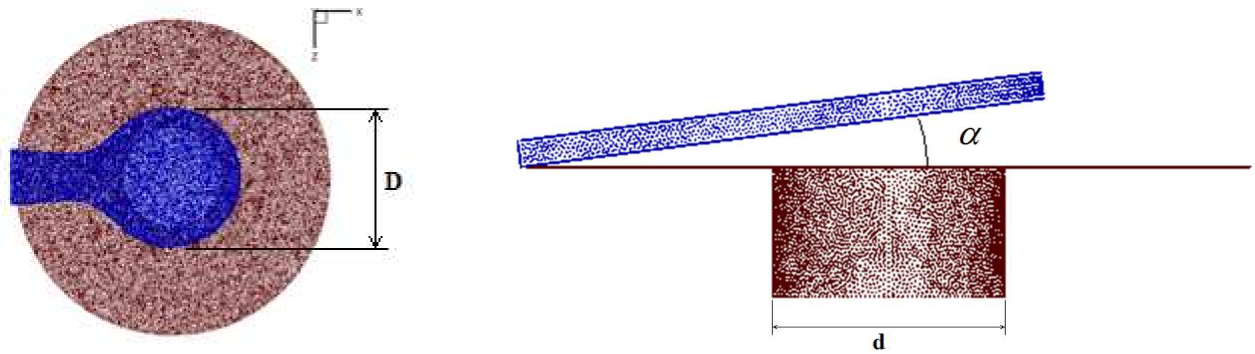


Figure 2. Example of the Lagrangian domain covered by the finest level of grid patches.

The temporal discretization of Eq. 2 is based on the IMEX (Implicit-Explicit) scheme, a second order scheme described by Ascher *et. al* (1997). This method allows the use of time-advance schemes as the SBDF (Semi-Backward Difference Formula), MCNAB (Modified Cranck-Nicolson Adams-Bashforth), CNAB (Cranck-Nicolson Adams-Bashforth) and CNLF (Cranck-Nicolson Leap-Frog). The central difference scheme (CDS) is used to dicretize the equations on space. The Projection Method is used for the solution of the pressure-velocity coupling, while the algebraic linear systems are solved by a Multigrid technique.

The geometry of the valve is depicted in Figure 3. The IBM can reduce substantially the difficulty to represent complex geometries because it is not necessary to adapt the numerical grid to the immersed body.



(a) – Upper view of the model

(b) – Side view of the model

Figure 3. Three-dimensional model of the valve used in the present work.

Figure 4 shows the computational domain and boundary conditions used in the present simulation.

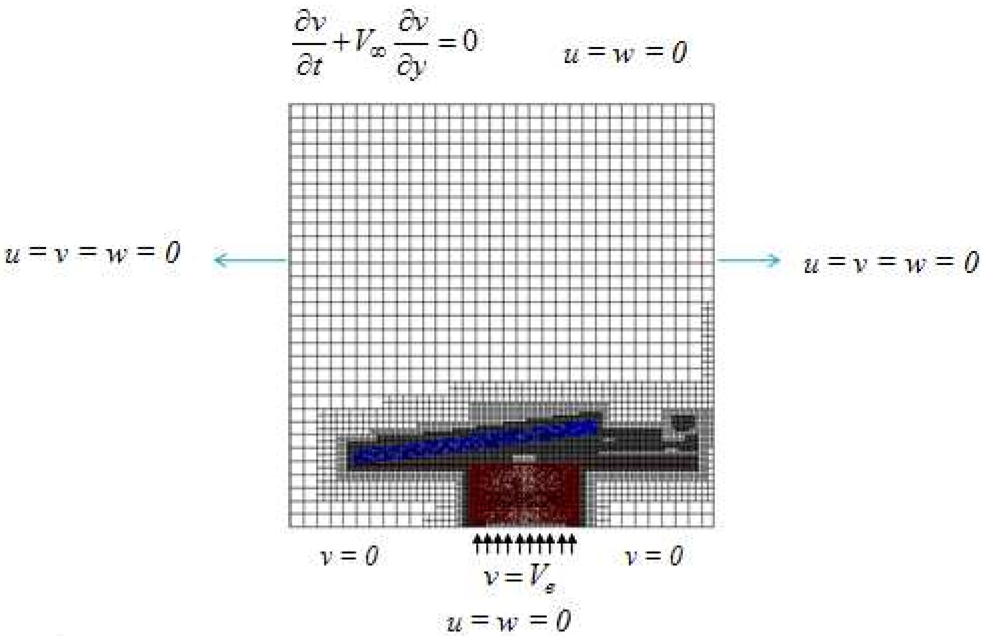


Figure 4. Computational domain and boundary conditions.

The Lagrangian grid is formed by 73.038 nodes, while the Eulerian grid is composed of about 3.000.000 nodes. For the Eulerian domain, the number of nodes varies according with the adaptive refinement.

3. RESULTS

Velocity vectors and dimensionless pressure field acting on the reed surface, P*, are obtained for Reynolds number equal to 500. The Reynolds number based on the feeding orifice diameter and average velocity, Re_D, the dimensionless pressure (P^*), and the dimensionless time (t^*) are given by:

$$\mathrm{Re_D} = \frac{\rho \overline{U} d}{\mu} \tag{9}$$

$$\mathrm{P}^* = \frac{P}{\frac{1}{2}\rho \overline{U}^2} \tag{10}$$

$$\mathrm{t}^* = \frac{\overline{U} t}{d} \tag{11}$$

where $\overline{U}$ is the average velocity in the feeding orifice entrance and d is the feeding orifice diameter.

Figures 5 and 6 depict the velocity field for inclination angle, α , equal to 7.5º at two different times and Fig. 7 shows the dimensionless pressure on the reed for t*=2.84. Similar results for inclination angles of 15º and 30º are shown in Figs. 8 to 13.

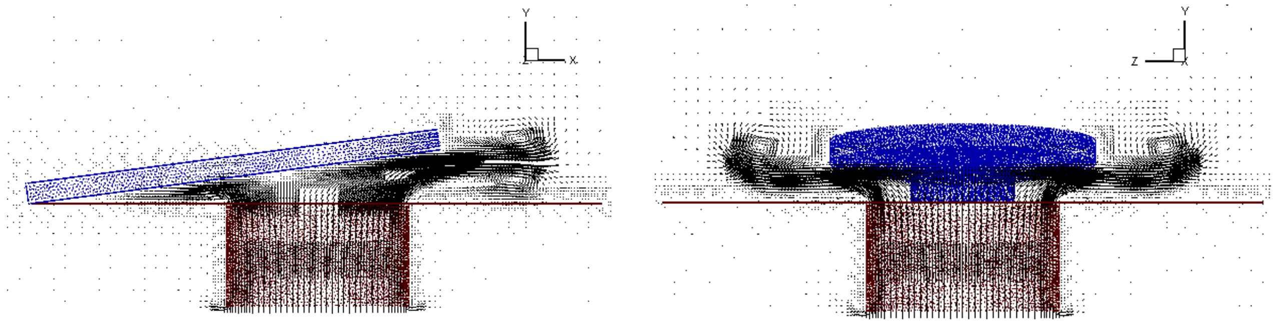


Figure 5 – Velocity vectors for t^*= 1.18 and α = 7.5º

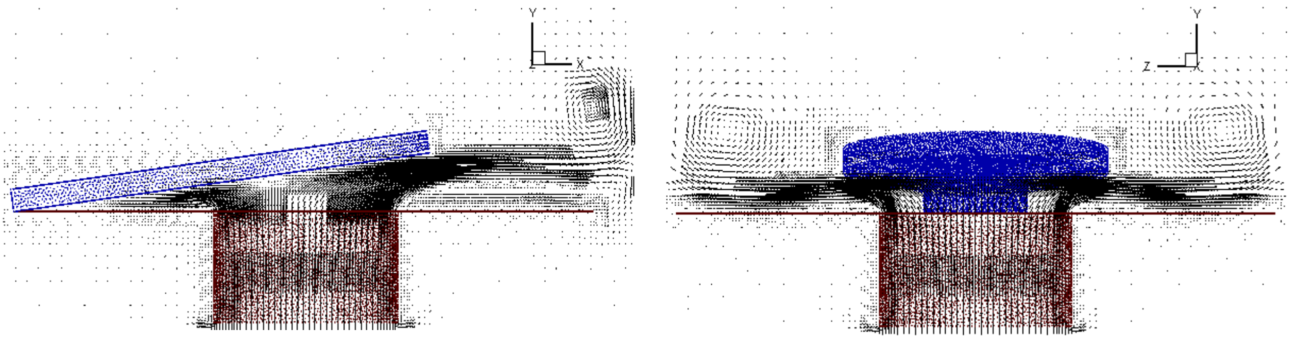


Figure 6 – Velocity vectors for t^*= 2.84 and α = 7.5º

It can be noted the formation of a vortex ring around the reed after the flow is deflected by the reed. This structure is dissipated after colliding with the boundaries of the computational domain. One can also observe that the flow is not symmetrical in relation to the longitudinal plane of the reed. The pressure distribution on the reed surface, shown in Fig. 7, is also not symmetrical, which was not expected for this low Reynolds number. This behavior could be due to the influence of the domain walls, but further studies have to be carried out to investigate this phenomenon. The lower pressure values are located on the left side of the reed owing to the higher restriction to the flow in this region.

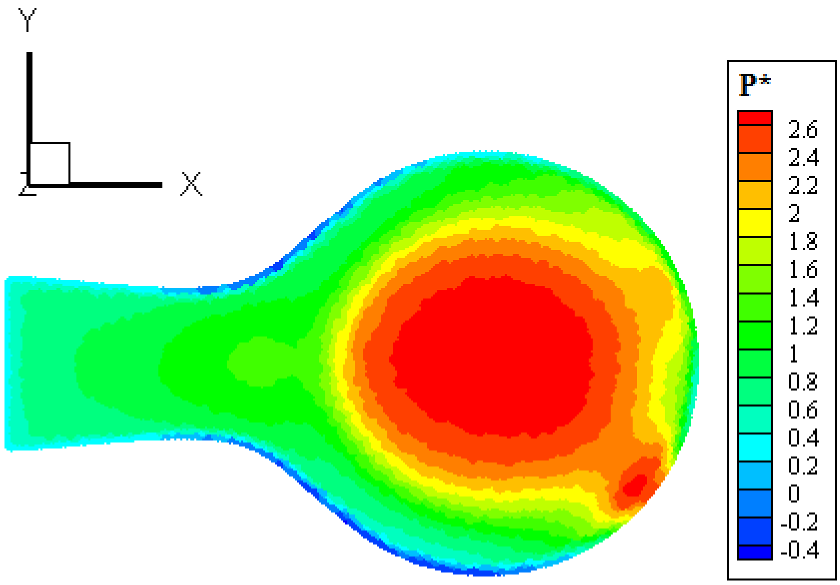


Figure 7. Dimensionless pressure distribution for $t^{*}= 2.84$ and $\alpha = 7.5^{o}$

As the inclination angle increases to 15º, the flow patterns are slightly modified. The pressure level on the reed surface decreases due to the lower restriction to the flow. It can be also noticed that the higher pressure region diminishes with α. For inclination angle even larger, α = 30º, one can see the formation of two major vortex in the xy plane. The pressure level is even lower for this case, as expected.

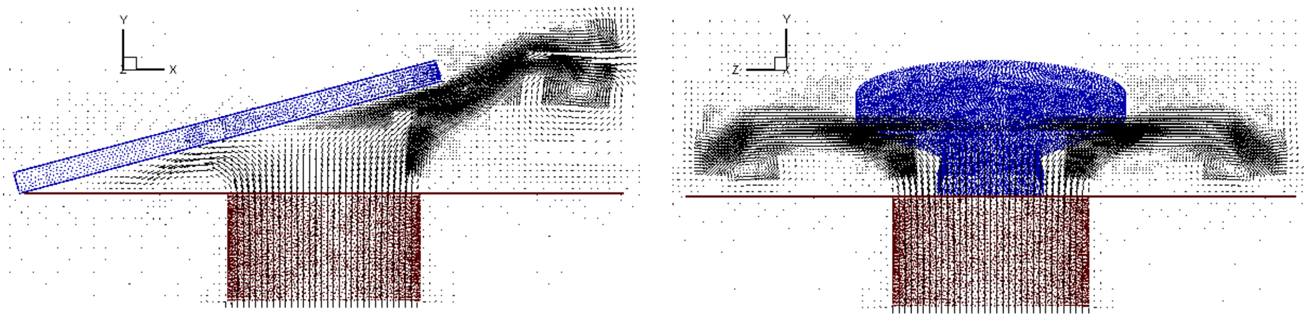


Figure 8 – Velocity vectors for $t^{*}= 2.73$ and $\alpha = 15^{o}$

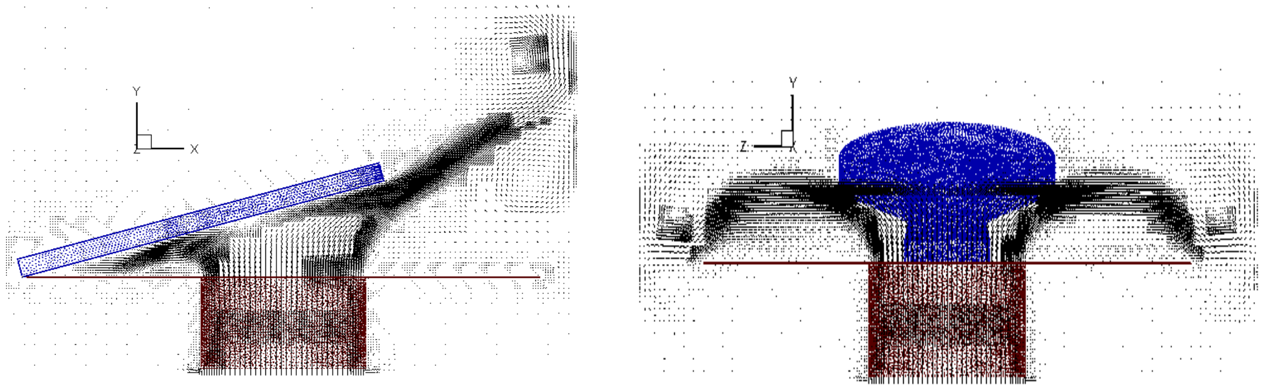


Figure 9 – Velocity vectors for $t^{*}= 5.62$ and $\alpha = 15^{o}$

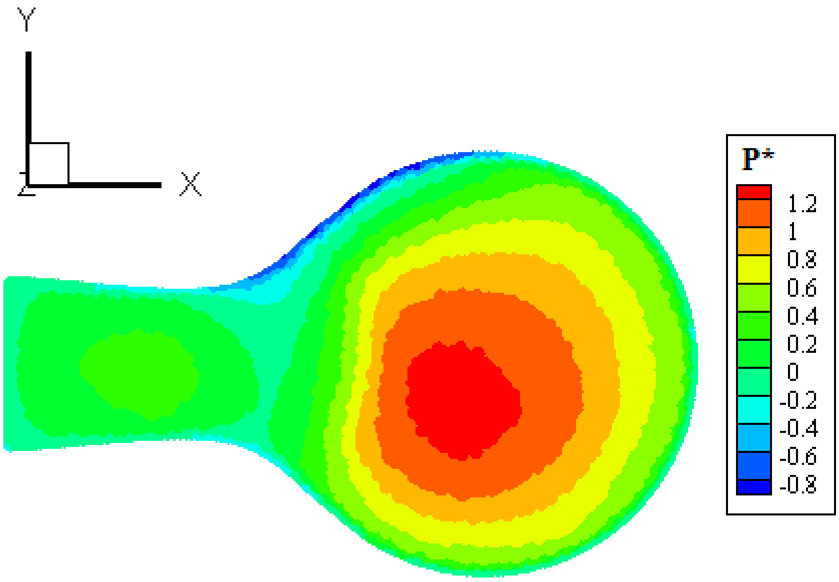


Figure 10. Dimensionless pressure distribution for $t^{*}= 5.62$ and $\alpha = 15°$

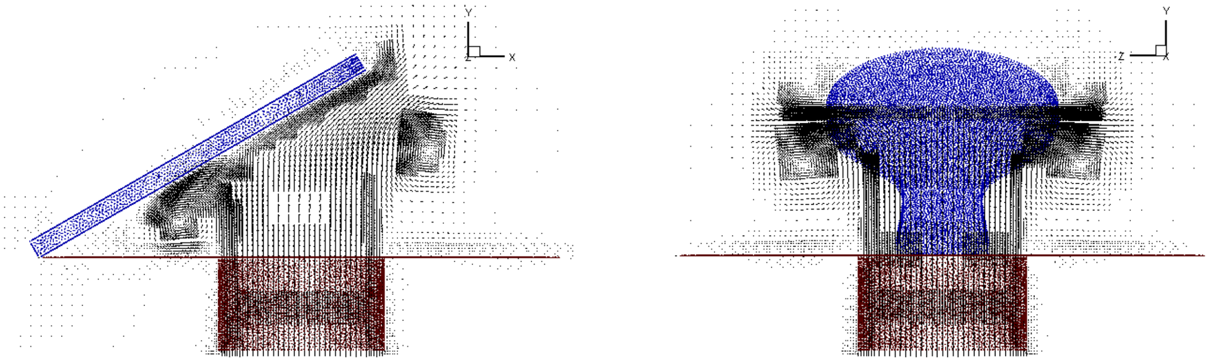


Figure 11 – Velocity vectors for $t^{*}= 2.3$ and $\alpha = 30°$

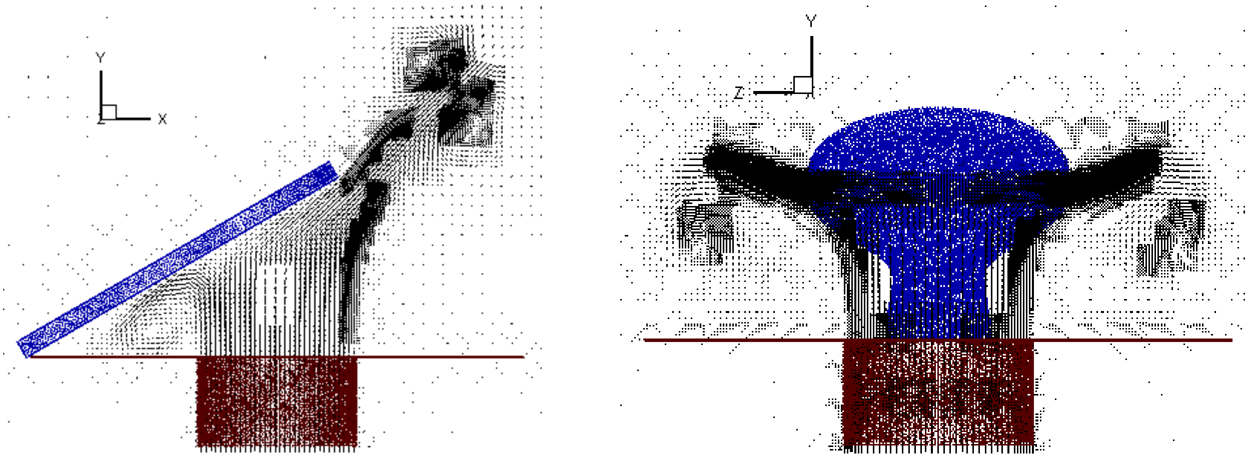


Figure 12 – Velocity vectors for $t^{*}= 4.43$ and $\alpha = 30°$

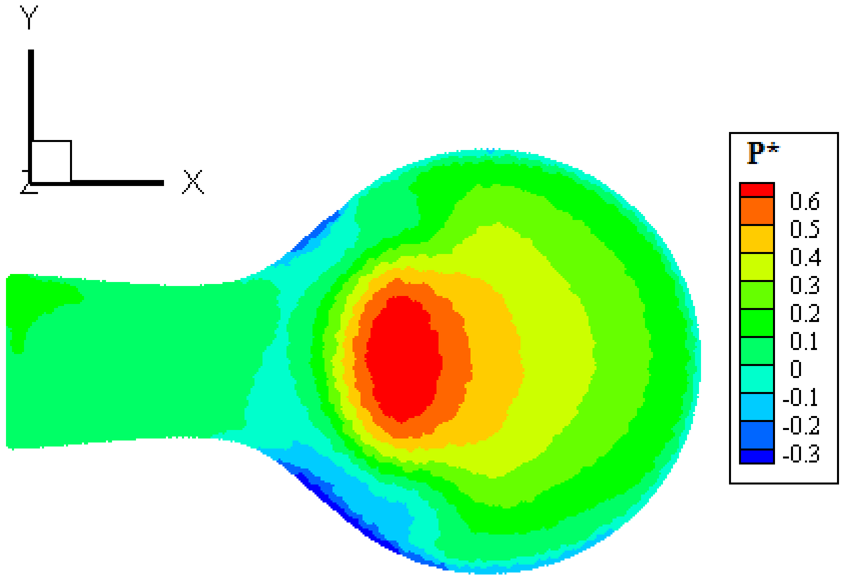


Figure 13. Dimensionless pressure distribution for $t^{*} = 4.43$ and $\alpha = 30°$

Figures 14-16 shows a tri-dimensional view of iso-vorticity surfaces from results to highlight the structures formed by the flow.

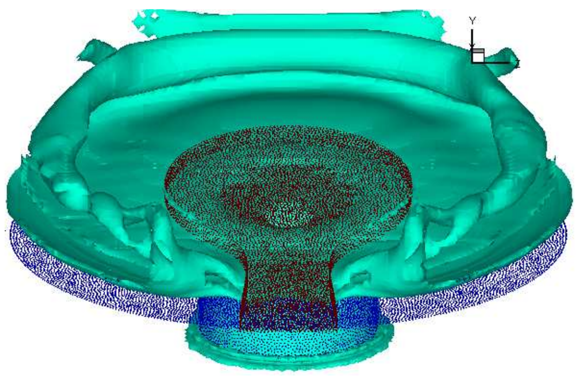


Figure 14. Iso-vorticity field for $t^{*} = 2.84$ and $\alpha = 7.5°$

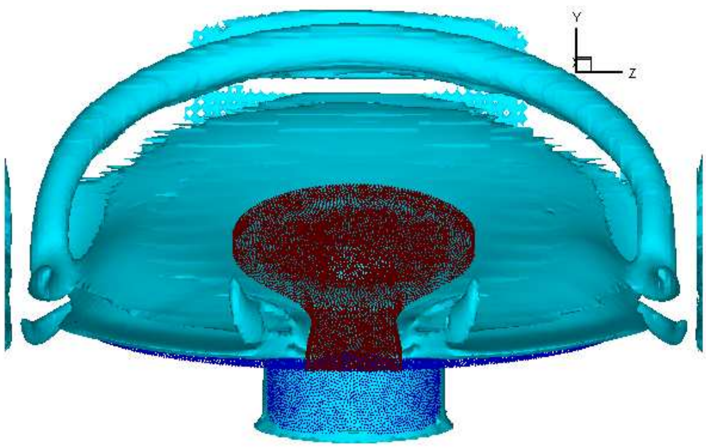


Figure 15. Iso-vorticity field for $t^{*} = 5.62$ and $\alpha = 15°$

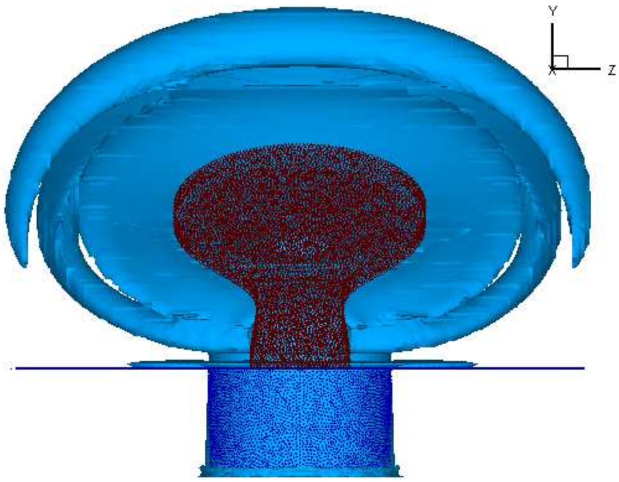


Figure 16. Iso-vorticity field for $t^* = 4.43$ and $\alpha = 30º$

4. CONCLUSIONS

This work presented a numerical simulation of the incompressible flow through a valve reed model used in refrigeration compressors. The Immersed Boundary Method was applied along with an adaptive mesh refinement algorithm for better computational efficiency. Results for Reynolds number equal to 500 and valve inclinations of 7.5º, 15º, and 30º were obtained. The velocity field and the dimensionless pressure distribution on the reed surface were analyzed for all cases. The results showed the formation of a vortex ring around the reed, just after the reed region. The pressure distribution on the reed surface is not symmetrical, as expected for the low Reynolds number analyzed. The level of the pressure on the reed surface decreases as the inclination angle increases due to the reduction of the friction forces of the flow. Based on these results, one can also conclude that the Immersed Boundary Method is a promising tool for studying the flow through the valve of refrigeration compressors, considering the actual geometry of the reed. In addition, the fluid-structure interaction between the reed and the flow, which is a current challenge for compressor design, can also be considered for future studies using this methodology.

5. REFERENCES

Ascher, U. M., Ruuth, S. J., Wetton, B., 1995, Implicit-Explicit Methods for Time Dependent Pde's., SIAM (Soc. Ind. Appl. Math.) J. Numer. Anal., vol. 32, p. 797.

Colaciti, A. K, López, L. M. V., Navarro, H. A., Cabezas-Gómez, L., 2007, Numerical Simulation of a Radial Diffuser Turbulent Airflow, *Applied Mathematics and Computation*, vol. 189, p. 1491-1504.

Deschamps, C.J., Ferreira, R.T.S. e Prata, A.T., 1996, Turbulent Flow Through Reed Type Valves of Reciprocating Compressors, ASME, *International Mechanical Engineering Congress*, Atlanta, EUA.

Ervin, J.S., Suryanarayana, N.V. e Chai NG, H., 1989, Radial, Turbulent Flow of a Fluid Between Two Coaxial Disks, *Journal of Fluid Engineering*, vol. 111, p. 378-383.

Ferreira, R. T. S., Driessen, J. L., 1986, Analysis of the influence of valve geometric parameters on the effective flow and force areas, International Compressor Engineering Conference at Purdue, p. 632-646.

Ferreira, R. T. S., Prata A. T., Deschamps C. J., 1989, Pressure distribution along valve reeds of hermetic compressors, Experimental Thermal and Fluid Science, vol. 2, no. 2: p. 201-207.

Gasche, J. L., 1992, Laminar flow through excentic valves of refrigeration compressors, MS. Dissertation, (in Portuguese), Federal University of Santa Catarina, Florianópolis-SC, Brazil, 149 p.

Hayashi, S., Matsui, T., Ito, T., 1975, Study of Flow and Thrust in Nozzle-Flapper Valves, *Journal of Fluids Engineering*, vol. 97, p. 39-50.

Lacerda, J., Gasche, J. L., 2010, Analysis of the Flow in Hermetic Compressor Valves Using the Immersed Boundary Method, Interrnational Compressor Engineering Conference at Purdue.

Lima e Silva, A. L. F., Silveira-Neto, A., Damasceno, J. J. R., 2004, Numerical simulation of two-dimensional flows over a circular cylinder using the immersed boundary method, Journal of Computational Physics, vol. 189, no. 2: p. 351-370.

Mittal, R., Iaccarino, G., 2005, Immersed boundary methods, Annual Review of Fluid Mechanics, vol. 37: p. 239-261.

Peskin, C. S., 1972, Flow pattern around heart valves: a numerical method, Journal of Computational Physics, vol. 10, no. 2: p. 252-271.

Piechna, J.R. and Meier, G.E.A., 1986, Numerical Investigation of Steady and Unsteady Flow in Valve Gap, *International Compressor Engineering Conference at Purdue*.

Possamai, F. C., Ferreira R. T. S., Prata, A. T., 2001, Pressure distribution in laminar radial flow through inclined disks, International Journal of Heat Fluid Flow, vol. 22, no. 4: p. 440-449.

Raal, J.D., 1978, Radial Source Flow Between Parallel Disks, *Journal of Fluid Mechanics*, vol. 85, p. 401-416.

Rodrigues, T. T., Gasche, J. L., Militzer, J., 2010, Flow Simulation Through Moving Hermetic Compressor Valves Using the Immersed Boundary Method, Interrnational Compressor Engineering Conference at Purdue.

Roma, A. M., Peskin, C. S., Berger, M. J., 1999, An Adaptive Version of the Immersed Boundary Method, Journal of Computational Physics 153, p. 509-534.

Tabatabai, M., Pollard, A., 1987, Turbulence in Radial Flow Between Parallel Disks at Medium and Low Reynolds Numbers, *Journal Fluid Mech*., vol. 185, p. 483-502.

Wang, Z., Fan, J., Luo, K., 2008, Combined Multi-Direct Forcing and Immersed Boundary for Simulating Flows with Moving Particles, International Journal of Multiphase Flow 34, p. 283-302.

Wark, C.E., Foss, J.F., 1984, Forces Caused by the Radial Out-Flow Between Parallel Disks, *Journal of Fluids Engineering*, vol. 106, p. 292-297.

6. RESPONSIBILITY NOTICE

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