APPLICATION OF THE IMMERSED BOUNDARY METHOD IN FLOWS THROUGH HERMETIC COMPRESSOR VALVES

Tadeu Tonheiro Rodrigues, tadeu.tonheiro@gmail.com
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. *In this work is presented the modeling of the flow through hermetic compressor automatic valves using the Immersed Boundary Method. The Immersed Boundary Method is attractive for modeling this problem once the displacement of the valve is ruled by the force applied on the valve reed by the refrigerant flow, establishing a fluid-structure interaction problem. Traditional methods, like body-fitted meshes, are computationally expensive for this kind of problem, due to re-meshing requirement in order to fit domain modifications. The Immersed Boundary Method is simple and computationally cheap when compared with body-fitted meshes. The governing equations are solved with a fixed orthogonal mesh, called eulerian mesh, while the solid immersed in the flow is modeled using a lagrangian mesh. These two meshes are independents and the coupling between them is made through the addition of a force term in the momentum equations. Using this approach, the displacement of the valve reed can be modeled only with the movement of the lagrangian mesh, without mesh adaptation. In this work, it is presented a preliminary study of the application of the Immersed Boundary Method together with the Virtual Physical Model (VPM) for modeling the flow through a static valve. The solution of the governing equations in cylindrical coordinates is accomplished by the Finite Volume Method, using the SIMPLEC algorithm to solve the velocity-pressure coupling problem. The Power-law is used to treat the advective-difusive terms and the three-time level is used to advance the time solution in a fully implicit approach. A staggered mesh for the velocity is used in relation to the pressure. The valve was simulated for Reynolds numbers equal to 500 and 1500, diameter ratio D/d = 1.5, and dimensionless gap s/d equal to 0.03, 0.05, and 0.07. The results were confronted with numerical data obtained through traditional methods. The main conclusion is that the Immersed Boundary Method is capable to model the valve flow and can be tested in future studies for considering the dynamic displacement of the valve reed.*

Keywords*: Automatic Valve, Immersed Boundary Method, Virtual Physical Model, Radial Diffuser.*

1. INTRODUCTION

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

Several studies have been made to characterize the main features of the flow through the valve. Numerical solutions for incompressible laminar flows have been obtained 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 obtained 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). Nevertheless, most of these studies are limited to static analysis of the problem, disregarding the movement of the valve reed and flow-structure interaction. Only few works in this area have taken into account the movement of the reed, although without considering the real problem of fluid-structure interaction (Matos et al., 2000 and Matos et al., 2001).

One of the main challenges for modeling the problem is the complex geometry of the valve. For this reason, simpler geometries have been adopted, specially the radial diffuser. Despite the valve geometry, the treatment of valve reed movement is challenging 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 is not an easy task and requires extensive computational resources.

The Immersed Boundary Method has been successfully used for different applications, especially for problems involving fluid-structure interaction. The Immersed Boundary 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 with the addition of a force field in the momentum equations for the cells located around the Lagrangian points. Therefore, the re-meshing procedure used in other methods is not necessary, which saves computation time.

The main purpose of this work is to apply the Immersed Boundary Method together with the Virtual Physical Model (VPM) for modeling the flow through compressor valves, in this first application in a static configuration, just to verify the applicability of the method.

In this work the flow through a radial diffuser representing the valve is solved numerically by using the Immersed Boundary Method with the Virtual Physical Model (Lima e Silva et al., 2003), used for calculating the force field. The radial diffuser is simulated using a two-dimensional approach, through the cylindrical coordinate system. Only the valve reed is modeled with the Immersed Boundary Method, while the valve seat is modeled by imposing infinity viscosity in the control volumes located inside the valve seat. The most important conclusion of the work is that the methodology showed to be suitable for studying the flow through the radial diffuser, indicating that it can be used in the future for the real geometry of the valve system, including the fluid-structure interaction between the reed and the flow.

2. MATHEMATICAL FORMULATION

In this study a two-dimensional, unsteady, incompressible, and isothermal flow of a Newtonian fluid in cylindrical coordinates was considered to solve the flow through the radial diffuser. The governing equations are the mass conservation and the momentum equations written in the r direction (radial) and x direction (axial), given by:

$$\frac{1}{r}\frac{\partial(r\rho u)}{\partial r}+\frac{\partial(\rho w)}{\partial z}=0 \tag{1}$$

$$\rho\left(\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial r}+v\frac{\partial v}{\partial x}\right)=-\frac{\partial p}{\partial r}+\mu\left(\frac{\partial^2 v}{\partial r^2}+\frac{1}{r}\frac{\partial v}{\partial r}-\frac{v}{r^2}+\frac{\partial^2 v}{\partial x^2}\right)+F_r \tag{2}$$

$$\rho\left(\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial r}+v\frac{\partial u}{\partial x}\right)=-\frac{\partial p}{\partial x}+\mu\left(\frac{\partial^2 u}{\partial r^2}+\frac{1}{r}\frac{\partial u}{\partial r}+\frac{\partial^2 u}{\partial x^2}\right)+F_x \tag{3}$$

where u and v are de velocity components in the axial and radial directions, respectively; p is the pressure; and ρ and μ are the density and the viscosity, respectively. The terms F_r and F_x are Eulerian force density fields responsible for representing the immersed body inside the flow. The Eulerian forces are calculated through the distribution of the Lagrangian interfacial forces, f_i, with the following equation:

$$F_i\left(\vec{x}_{ij}\right)=\sum_k D_{ij}\left(\left\|\vec{x}_{ij}-\vec{x}_k\right\|\right)f_i\left(\vec{x}_k\right)\frac{\Delta V_k}{\Delta V_{ij}} \tag{4}$$

where ΔV_k is the Lagrangian volume, ΔV_{ij} the Eulerian volume, $\vec{x}_k$ the Lagrangian point position, $\vec{x}_{ij}$ the Eulerian point position, and D_{ij} is the distribution function having Gaussian function properties (Lima e Silva *et al.*,2003).

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 interfacial Lagrangian force, which is defined as:

$$\vec{f}\left(\vec{x}_k,t\right)=\underbrace{\rho\frac{\partial\vec{V}\left(\vec{x}_k,t\right)}{\partial t}}_{\vec{fa}}+\underbrace{\rho\left(\vec{V}\cdot\vec{\nabla}\right)\vec{V}\left(\vec{x}_k,t\right)}_{\vec{fl}}-\underbrace{\mu\nabla^2\vec{V}\left(\vec{x}_k,t\right)}_{\vec{fv}}+\underbrace{\vec{\nabla}p\left(\vec{x}_k,t\right)}_{\vec{fp}} \tag{5}$$

where $\vec{fa}$, $\vec{fi}$, $\vec{fv}$, and $\vec{fp}$ represent, respectively, the inertial, acceleration, viscous, and pressure forces (by unit volume) acting on the fluid particle at the interface. The terms of Equation (5) are calculated using the Eulerian field data. Details of the implementation are founded in Lima e Silva *et al.* (2003).

3. NUMERICAL METHOD

The Finite Volume Method is used to discretize the governing equations, using a staggered grid. The Power-law scheme is used for interpolating the convective-diffusive terms and the SIMPLEC algorithm is used to treat the pressure-velocity coupling. The discretization of the transient term is accomplished by using the three-time level scheme, a fully implicit second order scheme. The algebraic equation systems are solved iteratively using the TDMA algorithm. Figure 1 shows the computational domain, the boundary conditions, and also the geometric parameters of the valve.

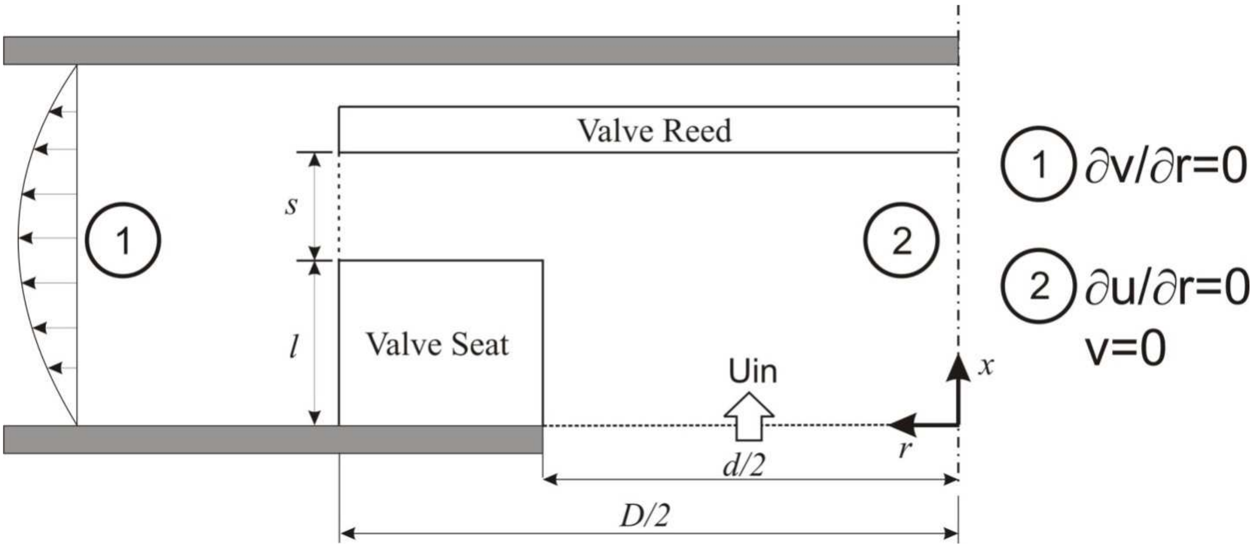


Figure 1. Computational domain and boundary conditions for the valve simulation.

In this work, the following geometric parameters were considered: the valve orifice diameter is d=3 cm and the diameter ratio is equal to D/d = 1.5; the height of the valve seat is l = 0.5 cm; dimensionless gaps, s/d, are equals to 0.03, 0.05 and 0.07; the dimensions of the computational domain are 1 cm and 2.75 cm in the x and r directions, respectively.

The mesh used to perform the simulation is showed in Figure 2. The region with uniform mesh configuration (shadow region) is the region where the valve reed is modeled. The mesh employed is composed by 127x491 volumes, totalizing 62357 elements. The valve seat is modeled by setting the viscosity to infinity in the correspondent control volumes.

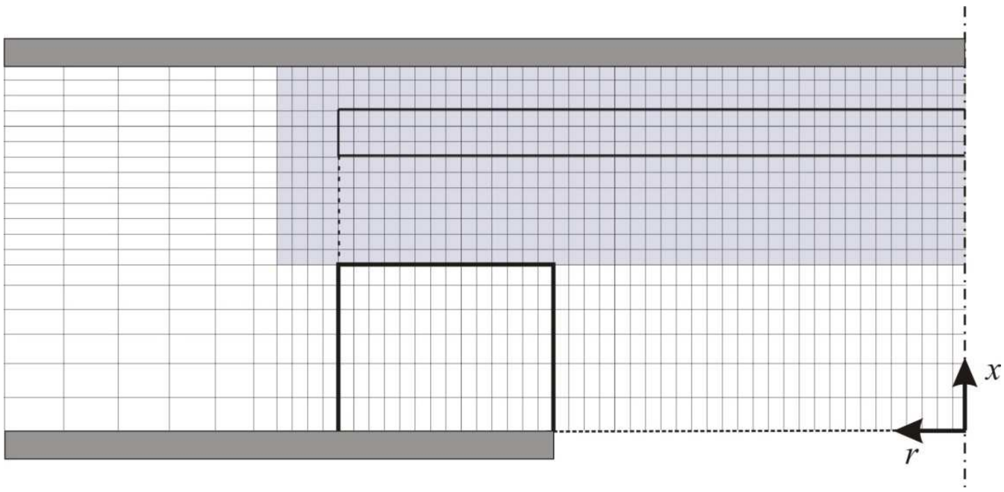


Figure 2. Mesh employed in the simulations.

4. RESULTS AND DISCUSSION

The simulations were performed for Reynolds numbers equals to 500 and 1500, which is defined as follows:

$$\mathrm{Re} = \frac{\rho U_{in} d}{\mu} \tag{6}$$

where U_{in} is the uniform velocity at the inlet of the flow. A important result for the radial diffuser flow is the dimensionless pressure profile on the valve reed surface, P*, which is defined by Eq. (7):

$$P^* = \frac{p}{\frac{1}{2}\rho U_{in}^{\ 2}} \tag{7}$$

Figures 3 and 4 present the dimensionless pressures profiles for Reynolds numbers equals to 500 and 1500, respectively, which are confronted with numerical results obtained with the traditional method (Finite Volume Method).

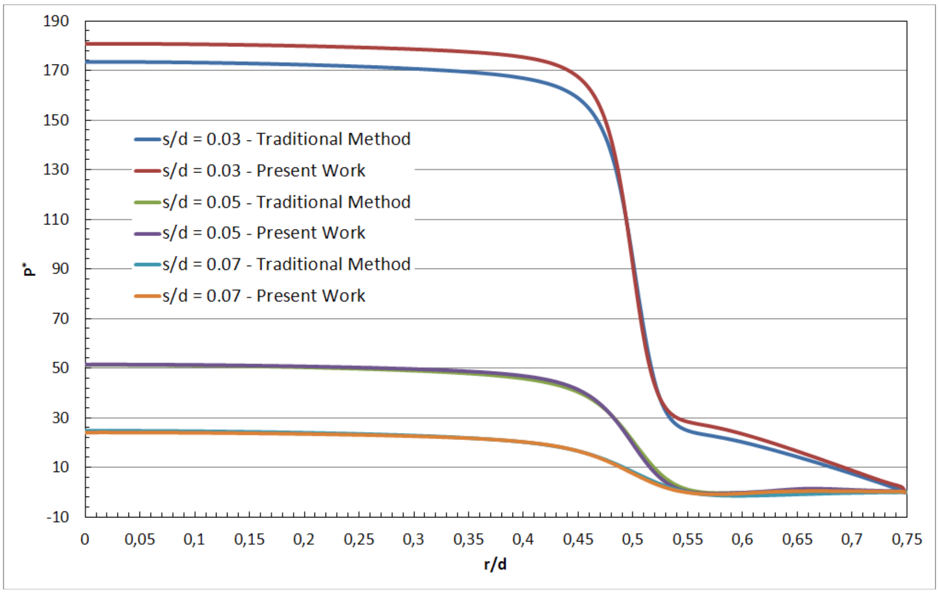


Figure 3. Dimensionless pressure profiles for Re = 500.

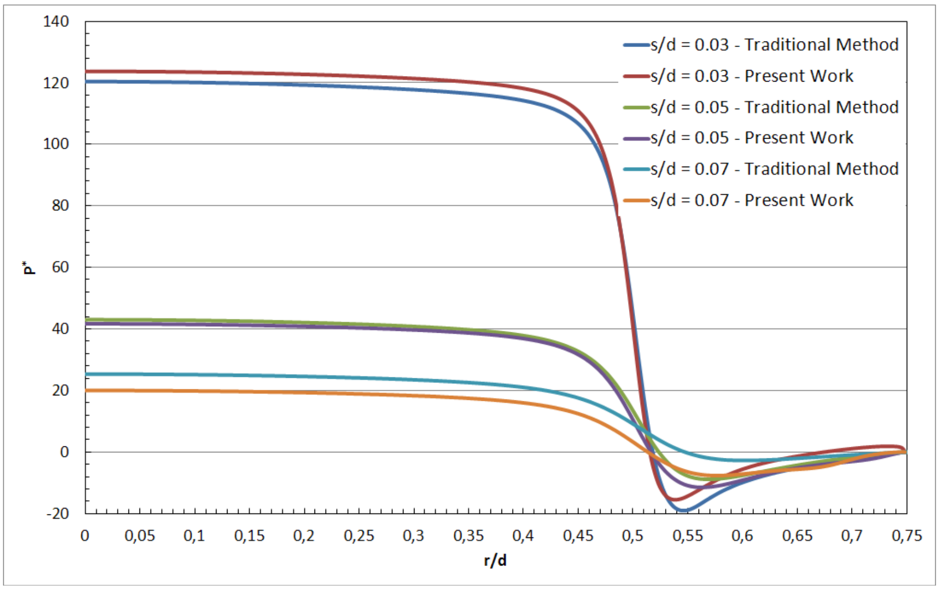


Figure 4. Dimensionless pressure profiles for Re = 1500.

It can be observed a good general agreement between the profiles obtained with both approaches. Table 1 shows the comparison of the maximum pressure (for r/d=0) obtained from both approaches. Except for Re = 1500 and s/d = 0.07, all the results showed a error lower than 5 %.

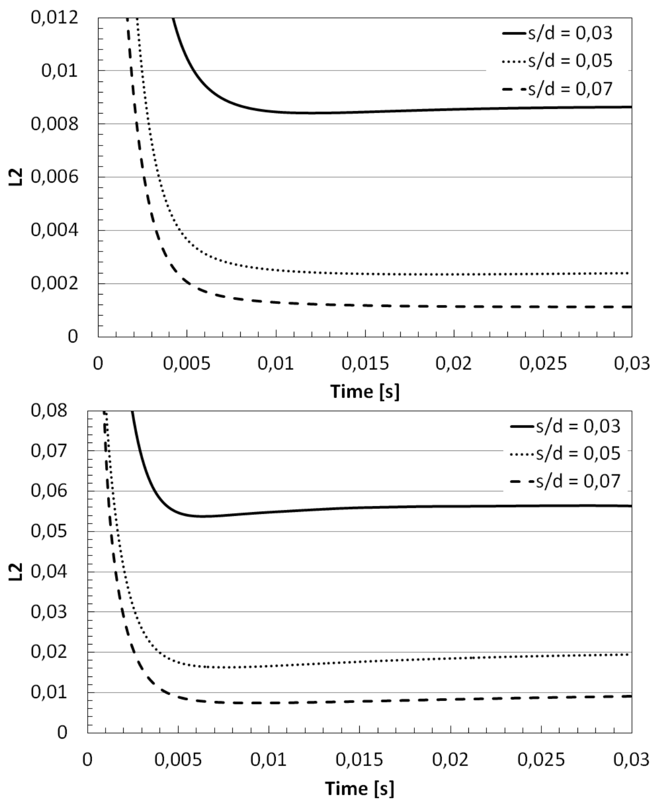
Table 1. Comparison between the maximum pressures obtained from both approaches.

Reynolds	s/d	P^* Traditional Method	P^* Present Work	Error [%]
500	0,03	173,45	180,79	4,63
500	0,05	51,27	51,46	0,38
500	0,07	24,75	24,07	2,82
1500	0,03	120,37	123,71	3,33
1500	0,05	42,99	41,66	3,02
1500	0,07	25,32	19,99	20,94

One important parameter used to evaluate if the non-slip and impermeability conditions are been satisfied at the immersed boundary surfaces is the L2 norm, defined as:

$$L_2 = \frac{\sqrt{\sum \left(u_k - u_{fk}\right)^2}}{n} \tag{8}$$

where u_k is the prescribed velocity at the immersed boundary surface, u_{fk} is the velocity of the fluid at the interface and n is the number of Lagrangian points. The time evolution of the L2 norm is presented in Fig. 5 for the simulated cases.

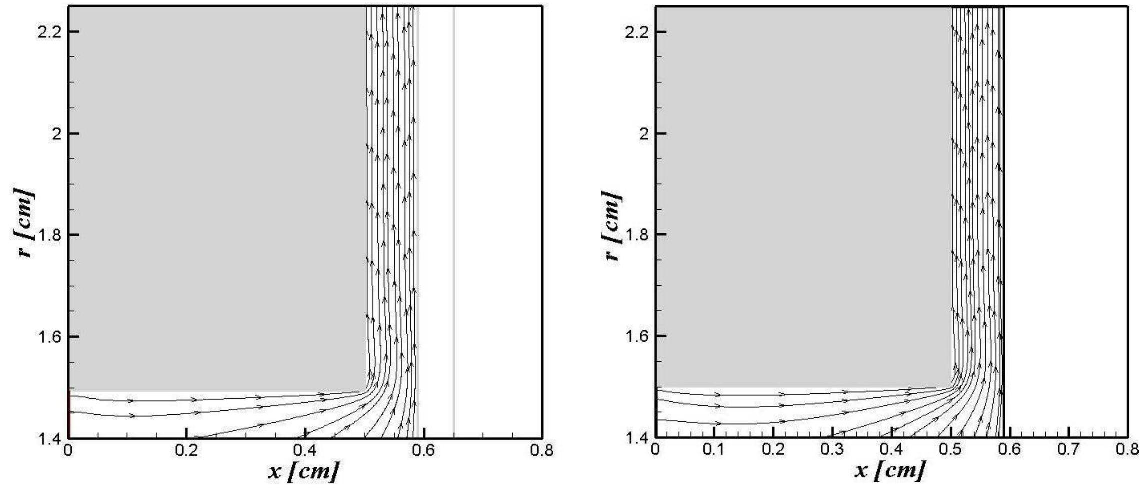


(a) (b)

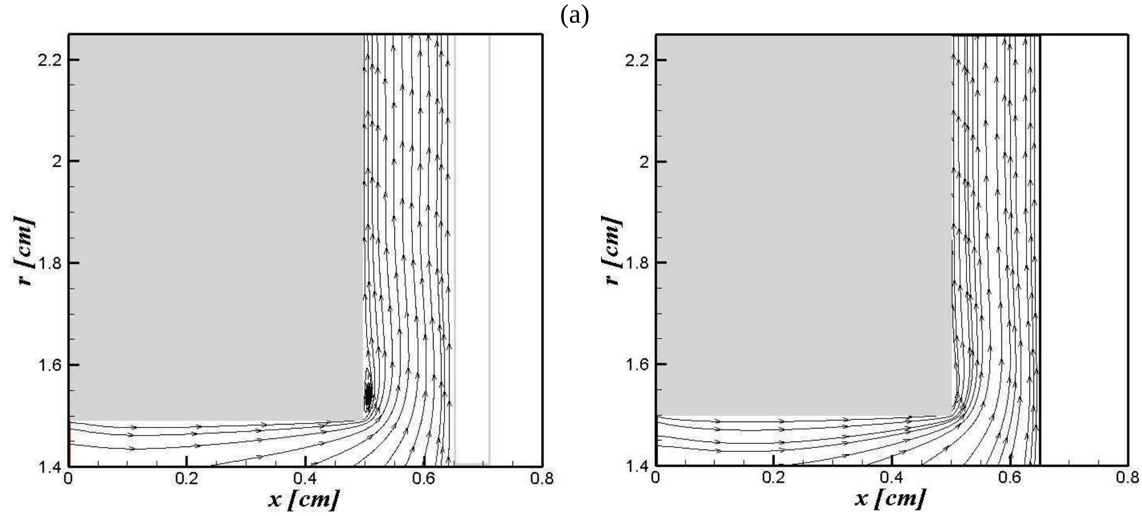
Figure 5. Time evolution of the L2 norm for (a) Re = 500 and (b) Re = 1500

It can be observed that the worst values for the L2 norm were obtained for Re = 1500. It is also clear a tendency for the L2 norm reaches better values for increasing gaps s/d. This means that the method deals better with velocity fields with smaller magnitudes, once that for increasing Reynolds numbers and decreasing relations s/d, the flow is highly accelerated in the diffuser region. Despite this, the values obtained for the L2 norm were satisfactory to produce good results for the dimensionless pressure profiles on the reed surface, which demonstrates a good performance of the method. In order to obtain better results a refined mesh must be used.

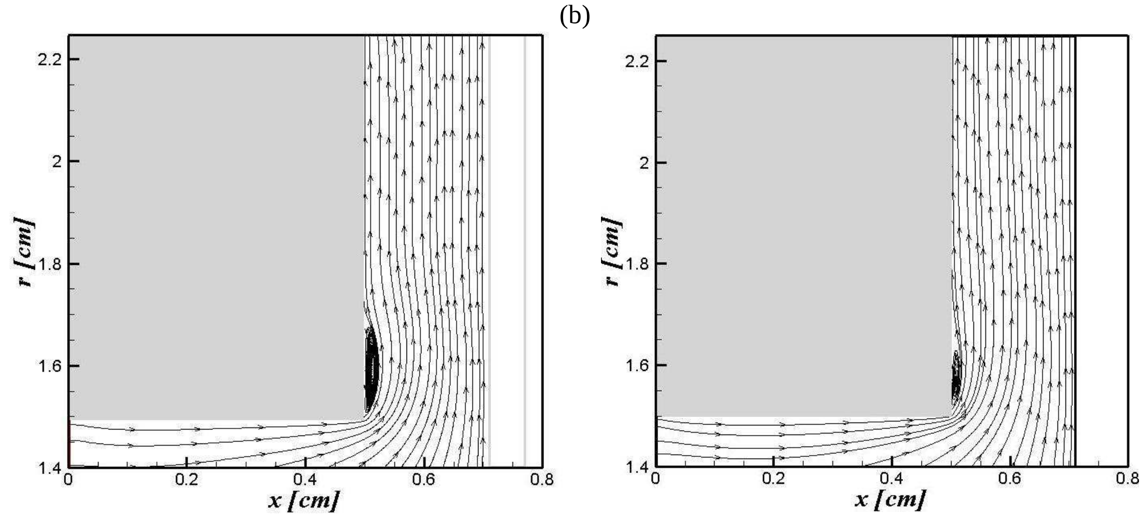
Figures 6 and 7 present the streamlines for Re = 500 and 1500, respectively, in the diffuser region.



(a)



(b)



(c)

Figure 6. Streamlines for Re = 500 considering the Immersed Boundary Method (left side) and the traditional method (right side) approaches for (a) s/d = 0.03, (b) s/d = 0.05 and (c) s/d = 0.07.

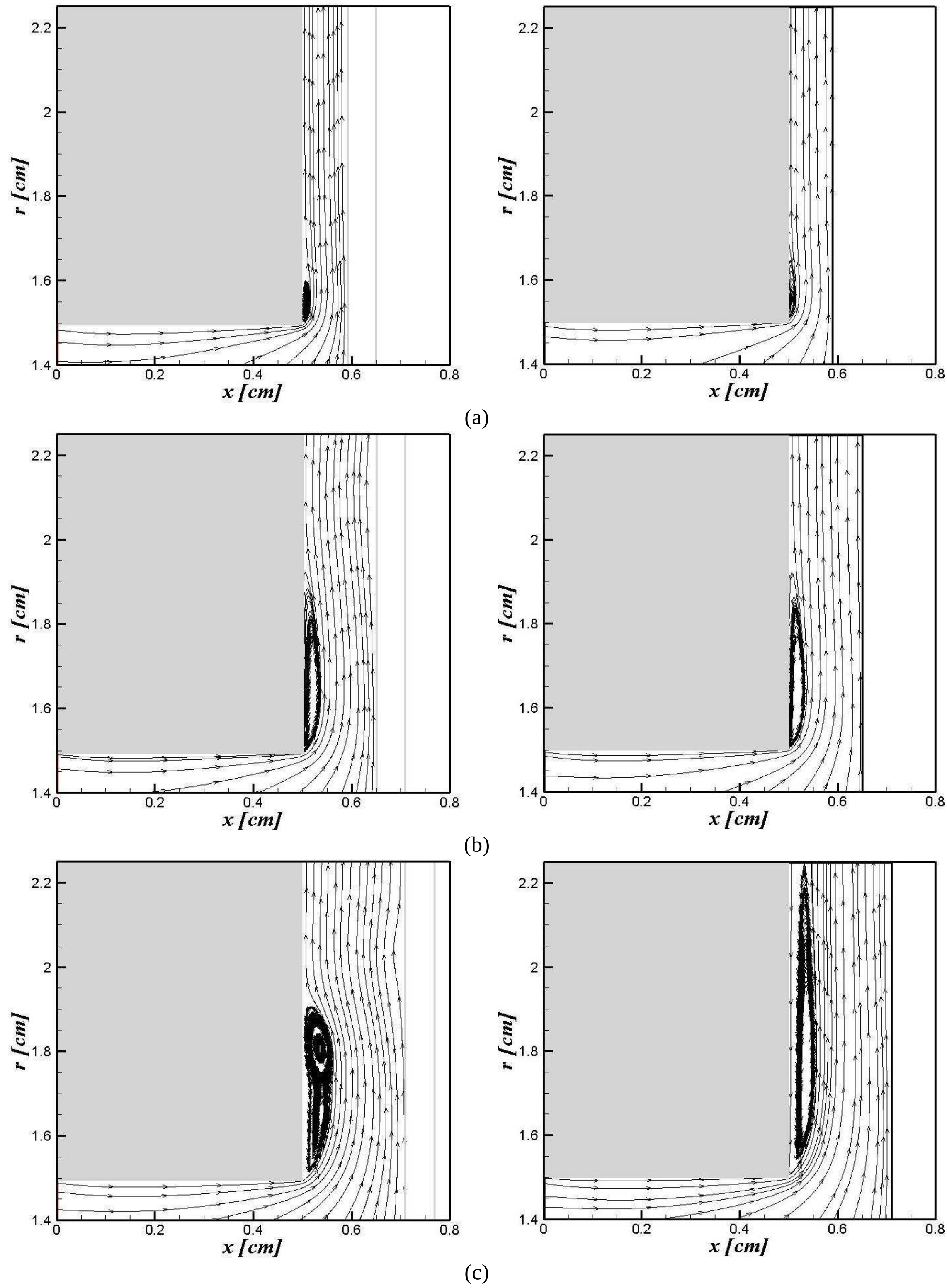


Figure 7. Streamlines for Re = 1500 considering the Immersed Boundary Method (left side) and the traditional method (right side) approaches for (a) s/d = 0.03, (b) s/d = 0.05 and (c) s/d = 0.07.

One can note that the streamlines obtained from both approaches are slightly different, mainly for s/d=0.07, where the vortex obtained from the traditional method is much larger. This explains the higher pressure for this case, once that the consequent reduction of the cross section area of the flow in the diffuser region increases the flow resistance. According to Roma et al., (1999), the Immersed Boundary Method has low resolution at the vicinity of the immersed boundary. The resolution can be improved by increasing the mesh density at the vicinity of the immersed boundary. In this work, the coarse mesh at the region of the immersed body can be the explanation for the different flow patterns observed.

5. CONCLUSIONS

In the present work the Immersed Boundary Method, together with the Virtual Physical Model, was used for modeling the flow through valves of refrigeration compressors. The main aim of the work is to evaluate the reliability of the method for modeling the valve in a static configuration with future purposes to consider the simulation of the real problem of fluid–structure interaction present in the valve dynamics.

The dimensionless pressure profiles on the valve reed surface showed a good agreement with numerical results obtained from the Finite Volume Method. As a consequence, one can conclude that the method can be used in further studies considering the motion of the valve reed, including also the fluid–structure interaction for modeling the valve dynamics.

6. ACKNOWLEDGEMENTS

The authors acknowledge the financial support given by Tecumseh do Brasil Ltda and Fepisa–Fundação de Ensino, Pesquisa e Extensão de Ilha Solteira.

7. REFERENCES

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, p. 201-207.

Gasche, J.L., Ferreira, R.T.S., Prata, A.T., 1992, “Pressure distribution along eccentric circular valve reed of hermetic compressors”, 12th Purdue Int. Compressor Technology Conference, West Lafayette, USA, pp. 1189-1198.

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

Lima e Silva, A.L.F., Silveira-Neto, A., Damasceno, J.J.R., 2003, “Numerical simulation of two-dimensional flows over a circular cylinder using the immersed boundary method”, Journal of Computational Physics, Vol. 189, pp. 351-370.

Matos, F.F.S., Prata, A.T. e Deschamps, C.J., 2000, “A Numerical Methodology of Valve Dynamics”, International Compressor Engineering Conference at Purdue, p. 383-390.

Matos, F.F.S., Prata, A.T. e Deschamps, C.J., 2001, “Modeling of Dynamics of Reed Type Valves”, International Conference on Compressor and Coolants, Slovak, p. 24-31.

Mittal, R.; Iaccarino, G., 2005, “Immersed Boundary Methods”, Annual Review of Fluid Mechanics, Vol. 37, pp. 239-261.

Peskin, C. S., 1972, “Flow patterns around heart valves: a numerical method”, Journal of Computational Physics, vol. 10, pp. 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, New York, vol. 22, n. 4, p. 440-449.

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

Roma, A., M.; Peskin, C., S.; Bergery, M., J. An adaptive version of the immersed boundary method. Journal of Computational Physics, New York, v. 153, p. 509-534. 1999.

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.

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.

8. RESPONSIBILITY NOTICE

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