NUMERICAL SIMULATION OF THE FLOW THROUGH THE VALVE SYSTEM OF REFRIGERATION COMPRESSOR

Jônatas Ferreira Lacerda, jonatasflacerda@hotmail.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, Av. Brasil, 56-Centro, 15385-000, Ilha Solteira, SP, Brasil.

Abstract. *An experimentally validated analysis of the incompressible, laminar and isothermal flow in eccentric radial diffusers representing a laboratory model of refrigeration compressor valve system is numerically performed. The finite volume methodology is used to calculate the flow field. A three-dimensional bicylindrical coordinate system is used to obtain the pressure and velocity fields. Numerical results are obtained for three different Reynolds numbers: 500, 1500, and 3000; two different dimensionless gaps between disks: 0.01 and 0.03; and four horizontal eccentricity values: 0, 5, 10 and 15 mm. The numerical results showed good agreement with the experimental data. In addition, the results showed large influence of both the gap between disks and the Reynolds numbers on the dimensionless pressure distribution on the frontal disk (valve reed), especially for the dimensionless gap of 0.01. It was also observed that the influence of the eccentricity is larger for dimensionless gap of 0.01 and for lower Reynolds numbers.*

Keywords*: Finite Volume, Eccentric Radial Diffusers, Refrigeration Compressor Valves.*

1. INTRODUCTION

Reciprocating compressors are widely used in refrigeration, petrochemical, gas transportation and gas storage industries. This type of compressor is able to compress gases in a wide range of pressure ratios within a large range of flow rates. In all these systems the compressor is the main component and must operate reliably for several years (Habing, 2005).

The compression process in refrigeration reciprocating compressors is developed by the linear alternate piston displacement. The suction and discharge valves are responsible for the refrigerant gas retention and flow from the suction chambers to the compressor cylinder and from cylinder to the discharge chamber.

The appropriate design of the valve system is of fundamental importance for elevating the compressor efficiency. Valves with small opening and closing times, small pressure drops, and those restricting the gas backflow are required (Lopes, 1996; Souto, 2002). As the valve opening and closing movements are caused by the refrigerant gas flow force, a comprehensive understanding of the flow through the valve is essential in order to enhance the compressor efficiency.
The numerical simulation of this type of 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. Souto (2002) presents a good literature review about radial diffuser flows with application in refrigeration compressor valves. Works involving analytic solutions of steady, incompressible, and laminar flows have been performed by Woolard (1957), Livesey (1960), Savage (1964) and Killmann (1972). Numerical solutions for incompressible laminar flows have been obtained by Hayashi et al. (1975), Raal (1978), Piechna and Meier (1986), Ferreira et al. (1987), Deschamps et al. (1989), Ferreira et al. (1989), Langer et al. (1990), Gasche (1992) e Possamai et al. (1995). Numerical solutions for incompressible turbulent flows have been accomplished by Deschamps et al. (1988) and Deschamps et al. (1996). Experimental works about this subject have been performed by Wark and Foss (1984), Ferreira and Driessen (1986), Tabakabai and Pollard (1987), Ervin et al. (1989) and Gasche (1992). Some researches also have obtained numerical solutions for incompressible laminar flow including the front disk dynamic: Lopes (1994), Matos et al. (1999), Matos et al. (2000), Matos et al. (2001), and Salinas-Casanova (2001). Among these authors, only Gasche (1992) have used an eccentric geometry for the radial diffuser.

In the present work, an experimentally validated analysis of the incompressible, laminar and isothermal flow in eccentric radial diffusers representing a laboratory model of refrigeration compressors valve is numerically performed. Aiming to obtain better agreement with experimental results, finer meshes were used when compared with the mesh utilized by Gasche (1992).

The computational code is firstly experimentally validated using the experimental results obtained by Gasche (1992). After the validation process, numerical results were obtained for three different Reynolds numbers (500, 1500, and 3000), two different dimensionless gaps between disks (0.01 and 0.03), and four eccentricity values (0, 5, 10, and 15 mm), in order to compare the influence of the separation between disks and of the Reynolds numbers on the dimensionless pressure distribution.

2. MATHEMATICAL MODEL

Figure 1 depicts the radial diffuser together with the bicylindrical coordinate system (ψ, η, z), the computational domain, and the main parameters of the geometry: valve reed or front disk diameter (D), orifice diameter (d), eccentricity (e), valve seat height (l), and the gap between the valve reed and valve seat (h).

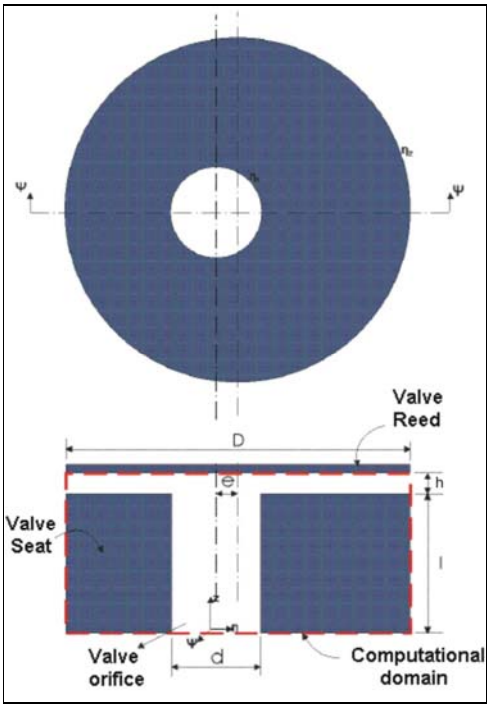


Figure 1. Geometry of the radial diffuser.

Using the bicylindrical coordinate system, the steady incompressible isothermal laminar flow of a Newtonian fluid through the diffuser is governed by the continuity equation and the Momentum equations, given by Eqs. (1) to (4).

Continuity

$$\frac{1}{h^2}\left[\frac{\partial}{\partial \psi}(\rho h u)+\frac{\partial}{\partial \eta}(\rho h v)+\frac{\partial}{\partial z}\left(\rho h^2 w\right)\right]=0 \tag{1}$$

Momentum in η direction (radial)

$$\begin{aligned}\frac{1}{h^2}\left[\frac{\partial}{\partial \psi}(h\rho uv)+\frac{\partial}{\partial \eta}(h\rho vv)+\frac{\partial}{\partial z}\left(h^2\rho wv\right)\right] &= -\frac{1}{h}\frac{\partial p}{\partial \eta}+\frac{\mu}{h^2}\left[\frac{\partial^2 v}{\partial \psi^2}+\frac{\partial^2 v}{\partial \eta^2}+h^2\frac{\partial^2 v}{\partial z^2}\right]+ \\ \frac{\mu}{h^2}\left[\frac{2}{h}\left(\frac{\partial h}{\partial \psi}\right)\left(\frac{\partial u}{\partial \eta}\right)-\frac{2}{h}\left(\frac{\partial h}{\partial \eta}\right)\left(\frac{\partial u}{\partial \psi}\right)-\frac{v}{h}\left(\frac{\partial^2 h}{\partial \eta^2}\right)-\frac{v}{h}\left(\frac{\partial^2 h}{\partial \psi^2}\right)\right] &-\frac{\rho}{h^2}\left(uv\frac{\partial h}{\partial \psi}-u^2\frac{\partial h}{\partial \eta}\right)\end{aligned} \tag{2}$$

Momentum in ψ direction (angular)

$$\frac{1}{h^2}\left[\frac{\partial}{\partial\psi}(h\rho uu)+\frac{\partial}{\partial\eta}(h\rho vu)+\frac{\partial}{\partial z}\left(h^2\rho wu\right)\right]=-\frac{1}{h}\frac{\partial p}{\partial\psi}+\frac{\mu}{h^2}\left[\frac{\partial^2 u}{\partial\psi^2}+\frac{\partial^2 u}{\partial\eta^2}+h^2\frac{\partial^2 u}{\partial z^2}\right]+$$
$$\frac{\mu}{h^2}\left[\frac{2}{h}\left(\frac{\partial h}{\partial\eta}\right)\left(\frac{\partial v}{\partial\psi}\right)-\frac{2}{h}\left(\frac{\partial h}{\partial\psi}\right)\left(\frac{\partial v}{\partial\eta}\right)-\frac{u}{h}\left(\frac{\partial^2 h}{\partial\psi^2}\right)-\frac{u}{h}\left(\frac{\partial^2 h}{\partial\eta^2}\right)\right]-\frac{\rho}{h^2}\left(uv\frac{\partial h}{\partial\eta}-v^2\frac{\partial h}{\partial\psi}\right) \tag{3}$$

Momentum in z direction (axial)

$$\frac{1}{h^2}\left[\frac{\partial}{\partial\psi}(h\rho uw)+\frac{\partial}{\partial\eta}(h\rho vw)+\frac{\partial}{\partial z}\left(h^2\rho ww\right)\right]=-\frac{\partial p}{\partial z}+\frac{\mu}{h^2}\left(\frac{\partial^2 w}{\partial\psi^2}+\frac{\partial^2 w}{\partial\eta^2}+h^2\frac{\partial^2 w}{\partial z^2}\right) \tag{4}$$

where h corresponds to the bicylindrical coordinate system metric, computed by Eq. (5):

$$h=\frac{a}{\cosh\eta-\cos\psi} \tag{5}$$

The geometric parameter a is defined by Eq. (6):

$$a=\sqrt{\frac{1}{2}\cdot\left[e^2-r_2^2-r_1^2+2\cdot\sqrt{\left(a^2\cdot r_2^2\right)\cdot\left(a^2\cdot r_1^2\right)}\right]} \tag{6}$$

where r_1 and r_2 are the eccentric circumference radiuses.

In the above equations the symbols ρ and μ stand for the fluid density [kg/m^3] and the absolute viscosity [Pa.s], respectively; p is the pressure [Pa]; and u, v and w are the circumferential (ψ direction), radial (η direction), and axial (z direction) components of the velocity vector [m/s], respectively.

3. NUMERICAL SOLUTION METHODOLOGY

The finite volume method is used in this work to solve the refrigerant gas flow through the valve. This method is extensively applied to solve fluid dynamics and heat transfer problems numerically. A detailed overview of the finite volume method is presented by Patankar (1980), Versteeg and Malalasekera (1995), Ferziger and Peric (2002) and Maliska (2004).

Basically the finite volume method consists of three main steps. First, the domain is divided into several finite control volumes, forming the discrete computational mesh. Usually it is performed a mesh size refinement to obtain accurate solutions. In the second step the governing partial differential equations are integrated in these control volumes and the resulted equations are discretized resulting in a linear algebraic equation system. The spatial discretization of the equations is very important, considering that the governing equations are non-linear and should be linearized properly. The power-law scheme is used to interpolate the variables at the volume faces. Staggered meshes with respect to pressure for each velocity component are employed. The SIMPLE algorithm (Patankar, 1980) for pressure-velocity coupling is used. Finally, the algebraic equation system is solved to obtain the solution. In this work the algebraic equation system is solved by the TDMA methodology.

Besides these basic steps it is always necessary to formulate physically and numerically the appropriate boundary conditions. In this work, a uniform velocity is prescribed at the inlet of the valve orifice and the flow is considered locally parabolic at the exit of the gap between the valve reed and the valve seat, that is, at the exit of the diffuser region. The non-slip boundary condition is used in the rest of the computational domain. The region of the valve seat is treated as an infinite viscosity fluid.

4. SETUP OF THE NUMERICAL SIMULATIONS

This section describes the details of the mesh and all the computational conditions assumed in the numerical simulations. The valve orifice diameter is d = 30 mm and the valve reed is characterized by the diameter D = 90 mm. Thus, the valve configuration is characterized by a diameter relation equal to D/d = 3. The height of the valvem seat is l= 28 mm. A mesh refinement in the radial direction is employed at the inlet of the diffuser region due to the large gradient. In the axial direction, a mesh refinement is performed at the inlet of the diffuser region and in the diffuser region itself. In the angular direction a regular mesh is used. Three different meshes are used in this work, as can bee seen in Tab. 1.Figure 2 shows an example of the radial and axial direction mesh used in the numerical simulations.

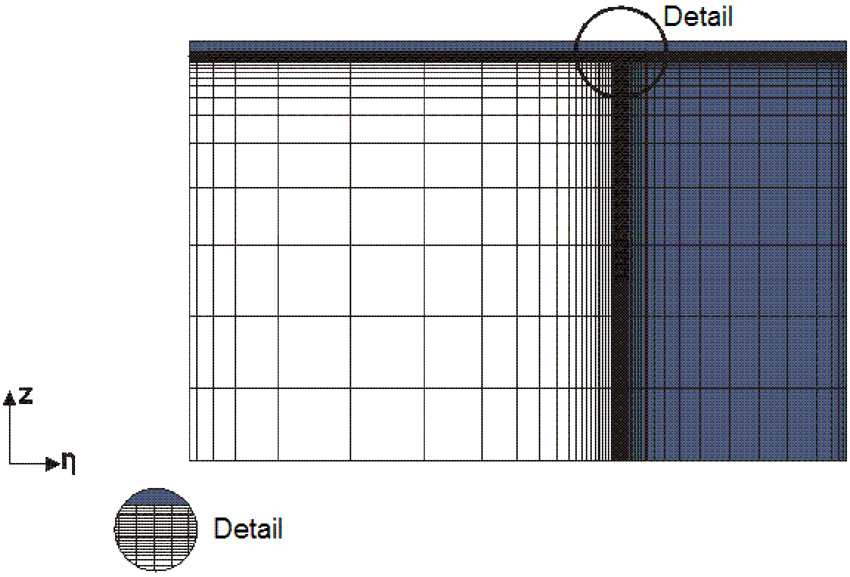


Figure 2. Example of the mesh used in the numerical simulations.

Table 1. Number of nodes used in the meshes.

Mesh Type	Total number of nodes in ψ direction	Total number of nodes in η direction	Total number of nodes in *z* direction	Total number of nodes
Mesh 1	14	38	27	14364
Mesh 2	14	54	40	30240
Mesh 3	22	70	53	81620

To validate the computational code, several numerical simulations are performed considering different valve reed gaps (h), eccentricity (e), and inlet Reynolds numbers (Re), according to experimental data obtained by Gasche (1992). The Reynolds number is defined by Eq. (7):

$$\mathrm{Re} = \frac{\rho W_{in} d}{\mu} \tag{7}$$

where W_{in} is the uniform velocity at the inlet of the valve orifice.

In addition, numerical results are obtained for three different Reynolds numbers: 500, 1500, and 3000; two different dimensionless gaps between disks (h/d): 0.01 and 0.03; and four horizontal eccentricity values: 0, 5, 10 and 15 mm using mesh 2.

5. NUMERICAL RESULTS

5.1 Code validation using experimental data

In order to validate the computational code, the experimental data obtained by Gasche (1992) shown in Tab. 2 are used.

Table 2. Computational parameter for numerical simulations for comparison with experimental results

Flow	Re	e (cm)	h (cm)
1	2032.99	0	0.06
2	3035.55	1.73	0.06
3	1491.00	0	0.075
4	1483.70	1.06	0.075

Figures 3 to 6 show the comparisons between experimental and numerical results obtained for the three meshes. It is shown the dimensionless pressure profile on the surface of the valve reed for ψ=0 and ψ=180^0, given by Eq. (8) .

$$Padm = \frac{p}{\frac{1}{2}\rho W_{in}^2} \quad (8)$$

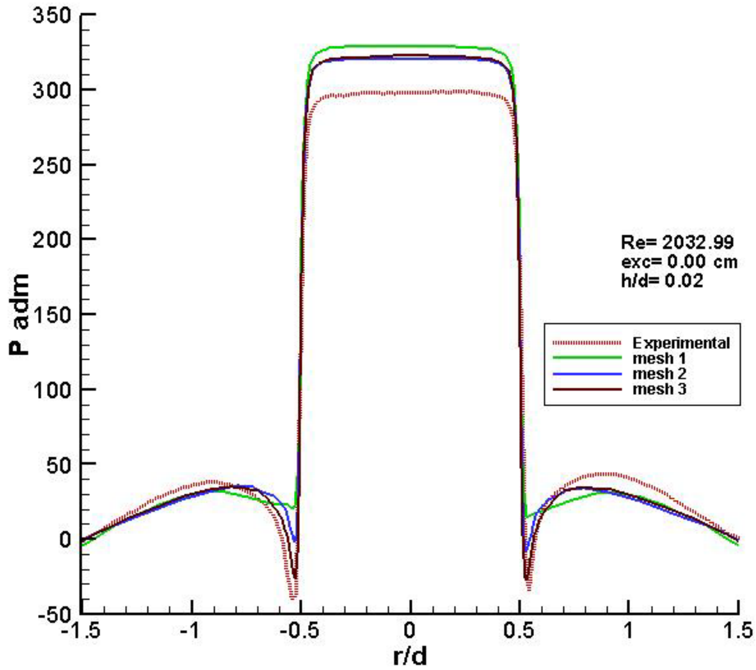


Figure 3. Validation for Re = 2032.99, eccentricity equal to zero and h=0.06 cm

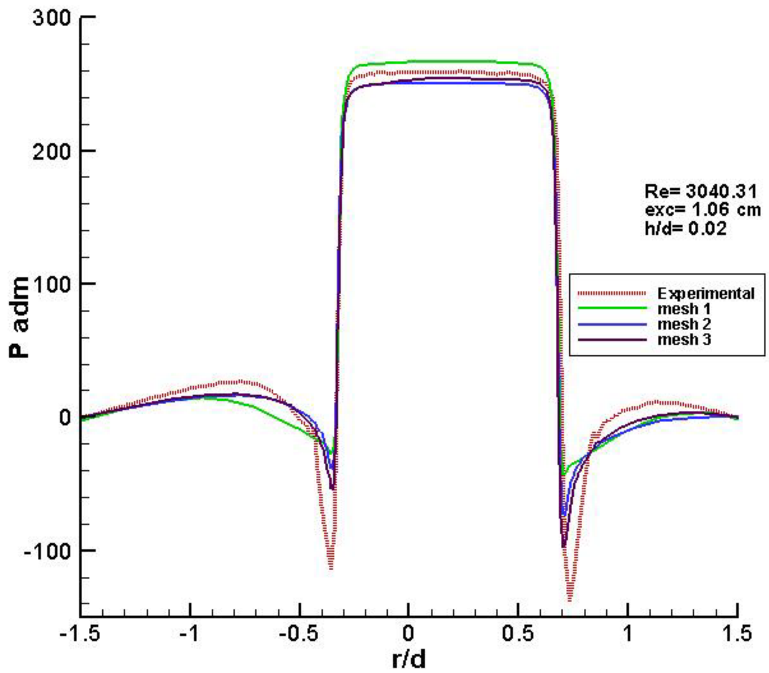


Figure 4. Validation for Re =3040.31, e=1.06 cm and h=0.06 cm

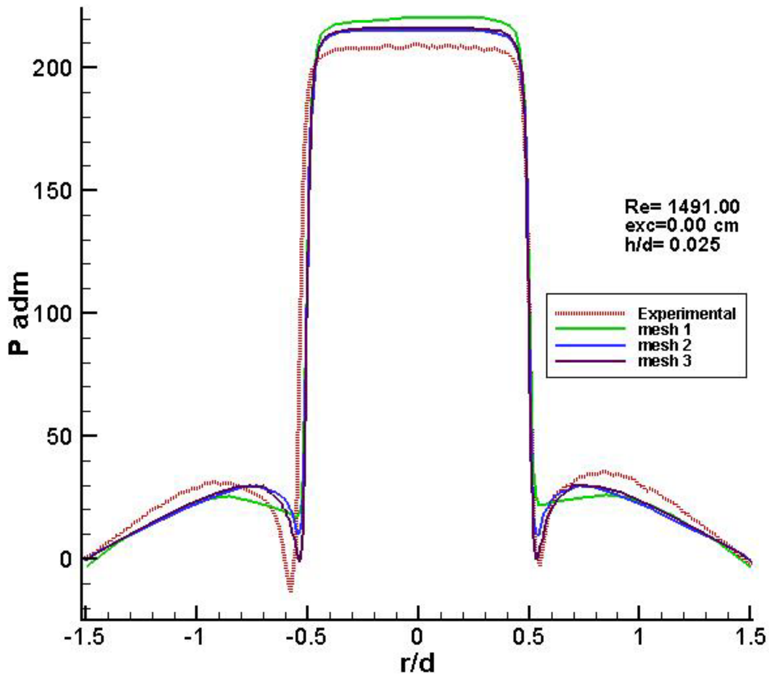


Figure 5. Validation for Re = 1491.00, eccentricity equal to zero and h=0.075 cm

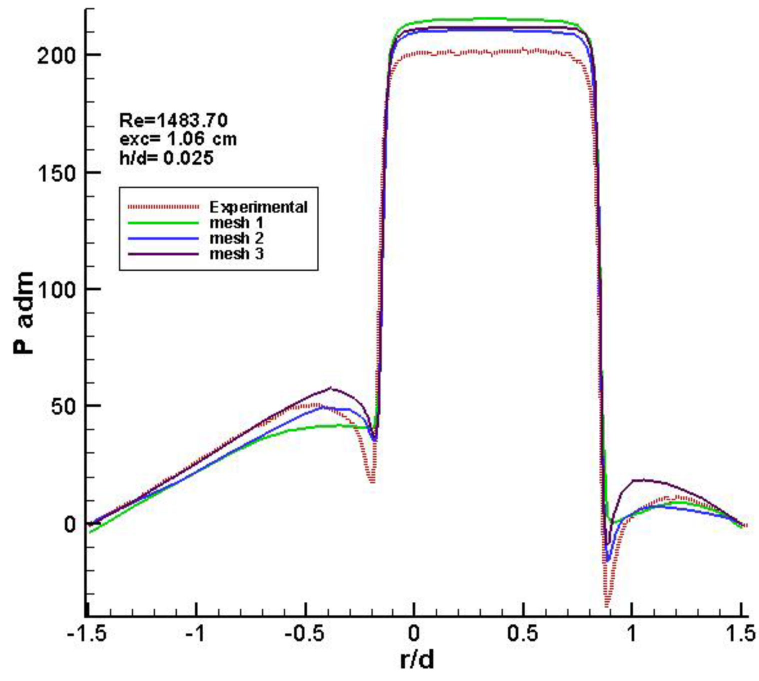


Figure 6. Validation for Re = 1483.70, e=1.06 cm and h=0.075 cm

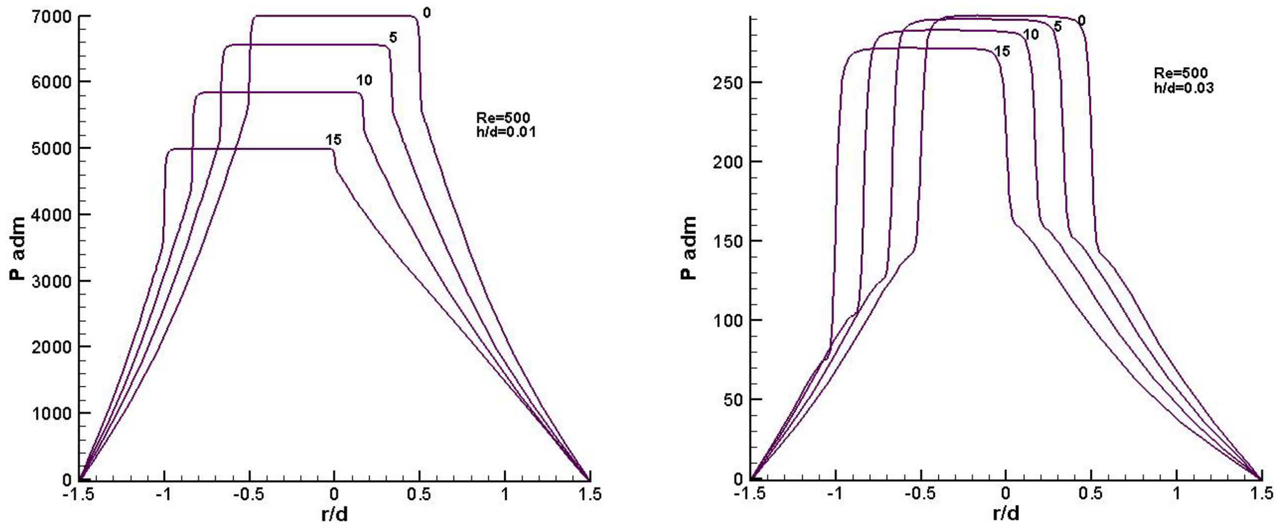
Observing the numerical results obtained with all meshes, one can say that they show good agreement with experimental results in all cases. Notice that at the inlet diffuser region there is a large pressure variation, especially for high Reynolds numbers. In this region there is a high fluid acceleration, which produces a high pressure gradient. Downstream this region pressure increases again due to the diffuser cross section area increasing. At the valve orifice region, on the surface of the valve reed, one can notice a uniform pressure distribution because the velocity is practically equal to zero, that is, it is a stagnation region. When the eccentricity is not zero, fluid flows preferentially to the region of less resistance, that is, to the side of lower length. For this reason, pressure gradient is larger in this region.

It can be observed that mesh 3 presents better results than meshes 1 and 2 as would be expected for finer meshes. As the distance between nodal points diminishes the errors of the interpolation scheme and false diffusion decrease as well, which tends to represent the physical problem more realistically. However, the CPU time and memory storage necessary to obtain a converged solution were largely augmented by the mesh refinement. In some cases this may represent restrictive factor in numerical simulations.

5.2 Influence of Reynolds numbers, dimensionless gaps and eccentricity

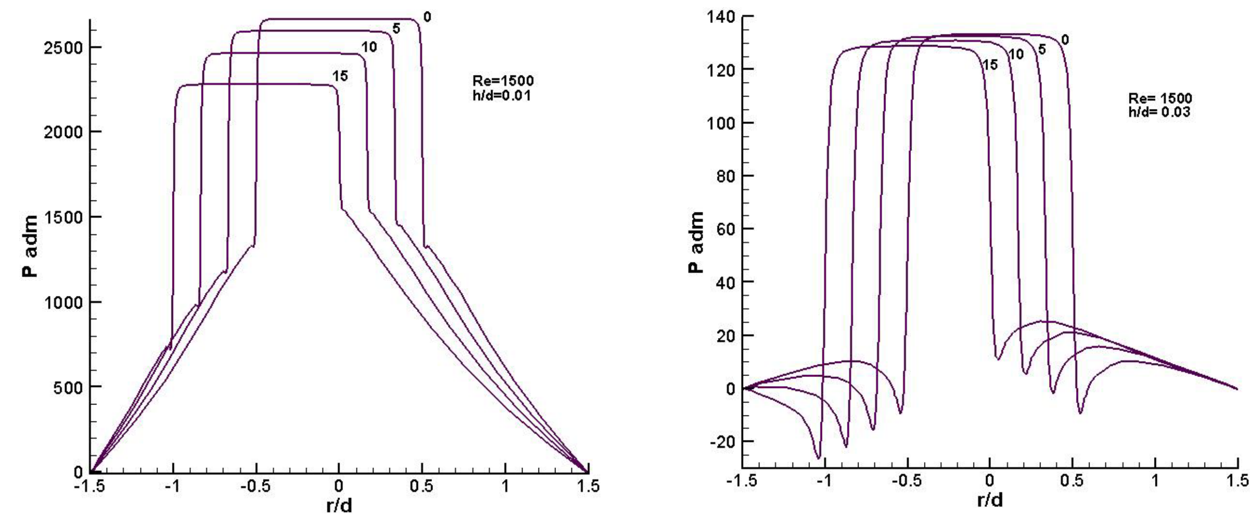
After the computational code has been validated one can study the problem for other flow configurations. As mesh 2 also presented good agreement with experimental results, it is used in the next section to obtain numerical results in order to analyze the influence of some geometrical and flow parameters on the flow behavior.

The influence of Reynolds number, dimensionless gap and eccentricity on the dimensionless pressure on the surface of the valve reed is performed. For this purpose, numerical results were obtained using mesh 2 for three different Reynolds numbers: Re=500, 1500, and 3000; two different dimensionless gaps between disks: h/d=0.01 and 0.03; and four horizontal eccentricity values: e=0, 5, 10 and 15 mm. The dimensionless pressure distribution on the valve reed for these situations are displayed in Figs. 7 to 9.



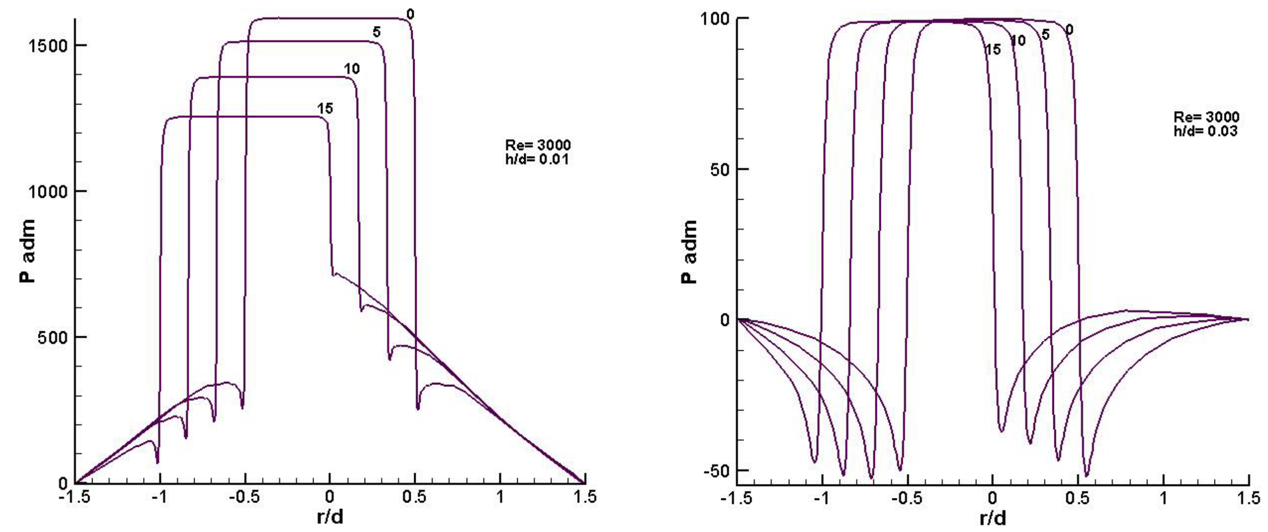
(a) (b)

Figure 7. Numerical results for: Re = 500; eccentricity 0, 5, 10 e 15 mm; h/d= (a) 0.01 and (b) 0.03



(a) (b)

Figure 8. Numerical results for: Re = 1500; eccentricity 0, 5, 10 e 15 mm; h/d= (a) 0.01 and (b) 0.03



(a) (b)

Figure 9. Numerical results for: Re = 3000; eccentricity 0, 5, 10 e 15 mm; h/d= (a) 0.01 and (b) 0.03

First of all, analyzing Fig. 7 one can notice the large influence of both eccentricity and dimensionless gap on the pressure profile. For eccentricity equal to zero, for example, as the dimensionless gap increase from 0.01 to 0.03 the stagnation pressure decreases about 24 times. In addition, one can observe that the influence of eccentricity is larger as the dimensionless gap increases. For h/d=0.01 the stagnation pressure diminishes about 29% as the eccentricity increases from 0 to 15 mm, while for h/d=0.03 the stagnation pressure diminishes only about 7%. The same tendencies, although in smaller scale, are noticed for Reynolds numbers of 1500 and 3000. For Re=3000 and h/d=0.03, for example, one can say that there is no influence of the eccentricity on the stagnation pressure.

Another interesting observation of the numerical results, for larger Reynolds numbers (Re=1500 and 3000) and dimensionless gap of 0.03, is the region of negative pressure at the inlet of the diffuser region. In these conditions, the fluid acceleration at this region is large enough to produce the negative pressures, mainly because the stagnation pressure is not much large than for the other flow configurations. In some flow conditions, not shown here, this negative pressure region is sufficient to produce negative force on the valve reed, which could instantaneously close the valve of the compressor.

6. CONCLUSIONS

The incompressible, laminar and isothermal flow in eccentric radial diffusers representing a laboratory model of refrigeration compressor valve system is numerically performed. The governing equations are writen in the three-dimensional bicylindrical coordinate system, a coordinate system that adapts perfectly to the valve geometry, and the finite volume methodology is used to solve the equations. Firstly, the computational code is validated using the experimental results obtained by Gasche (1992). After the code validation, numerical results are obtained for three different Reynolds numbers: 500, 1500, and 3000; two different dimensionless gaps between disks: 0.01 and 0.03; and four horizontal eccentricity values: 0, 5, 10 and 15 mm.

The main conclusions of the work are:

- the Reynolds number based on the orifice diameter and inlet velocity has large influence on the valve reed pressure distribution;
- the gap between valve reed and valve seat also influences largely the pressure profile on the valve reed;
- the eccentricity is another parameter affecting the pressure distribution on the valve reed, mainly for smaller gaps between de valve reed and valve seat;
- for higher Reynolds numbers and gaps between disks there is a negative pressure region on the valve reed, which diminishes the force on the valve.

7. ACKNOWLEDGEMENTS

The authors acknowledge the support given by FAPESP – São Paulo State Research Foundation

8. REFERENCES

Deschamps, C.J, Ferreira, R.T.S. e Prata, A.T., (1988), The Effective Flow and Force Areas in Compressor Valves, *International Compressor Engineering Conference at Purdue,* p. 104-111.

Deschamps, C.J., Ferreira, R.T.S. e Prata, A.T., (1989), Modelagem de um Escoamento Turbulento em Presença de estagnação, Recirculação, Aceleração e Gradientes de Pressão Adversos, *X Congresso Brasileiro de Engenharia Mecânica*, Rio de Janeiro-RJ, p. 57-60.

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. e 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. e Deschamps, C.J., (1987), Laminar Fluid Flow in Compressor Valves: Numerical and Experimental Results, *Fluid Flow and Heat Transfer in Reciprocating Machinery*, p.33-38.

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., (1992), Escoamento Laminar Através de Válvulas Excêntricas de Compressores de Refrigeração, Dissertação de Mestrado, Universidade Federal de Santa Catarina, Florianópolis-SC.

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.

Habing, R.A., 2005, "Flow and Plate Motion in Compressor Valves", PhD. Thesis, University of Twente, Enschede, Netherlands.

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.

Killman, I.G., (1972), Aerodynamic Forces Acting on Valve Discs, *International Compressor Engineering Conference at Purdue*, p. 407-414.

Langer, C.A., Prata, A.T. e Ferreira, R.T.S., (1990), Separação e Bifurcação de Escoamentos em Difusores Radiais, *III Congresso Brasileiro de Engenharia e Ciências Térmicas*, Itapema-SC, p.357-362.

Livesey, J.L., (1960), Inertia Effects in Viscous Flows, *Int. J. Mech. Sci*, vol. I, p. 84-88.

Lopes, M.N., 1994, "Uma Metodologia Numérica Para a Análise do Comportamento Dinâmico de Válvulas Tipo Palheta em Escoamentos Periódicos", Dissertação (Mestrado em Engenharia Mecânica) – Curso de Pós-Graduação em Engenharia Mecânica, UFSC, Florianópolis, SC.

Maliska, C.R., 2004, "Transferência de calor e mecânica dos fluidos computacional", Rio de Janeiro: LTC.

Matos, F.F.S., Prata, A.T. e Deschamps, C.J., (1999), Numerical Analysis of the Dynamic Behaviour of the Plate Valves in Reciprocating Compressors, *International Conference on Compressor and Their Systems*, London, p. 453-462.

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.

Patankar, S.V., 1980, "Numerical Heat Transfer and Fluid Flow", Hemisphere Publishing Corporation.

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

Possamai, F.C., (1994), Escoamento Laminar Através de Válvulas Inclinadas de Compressores de Refrigeração, Dissertação de Mestrado, Universidade Federal de Santa Catarina, Florianópolis-SC.

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

Salinas-Casanova, D.A., (2001), Análise Numérica do Escoamento Turbulento em Válvulas Automáticas de Compressores, Tese de Doutorado, Universidade Federal de Santa Catarina, Florianópolis-SC.

Savage, S.B., (1964), Laminar Radial Flow Between Parallel Plates, *Journal of Applied Mechanics*, dezembro, p. 594-596.

Souto, L.E.M., (2002), Investigação Experimental do Escoamento Turbulento em Válvulas de Compressores, Dissertação de Mestrado, Universidade Federal de Santa Catarina, Florianópolis-SC.

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

Versteeg, H.K., Malalasekera, W., 1995, "An introduction to computational fluid dynamics: The finite volume method", Harlow: Prentice Hall.

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

Woolard, H.W., (1957), A Theorical Analysis of the Viscous Flow in Narrowly Spaced Radial Diffuser, *Journal of Applied Mechanics*, march, p. 9-15.

9. RESPONSIBILITY NOTICE

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