Turbulent Flow Through the Reciprocating Compressor Valve Using the Large Eddy Simulation Methodology

José Luiz Gasche, gasche@dem.feis.unesp.br, [1]
[1] Department of Mechanical Engineering, UNESP - São Paulo State University - College of Ilha Solteira,
Av. Brasil Centro, 56 – Postal Code 15385-000. Ilha Solteira - SP, Brazil.
Bruno Ferreira Pussoli, pussoli@polo.ufsc.br, [2]
[2] POLO – Research Laboratories for Emerging Technologies in Cooling and Thermophysics – Department of Mechanical Engineering,
Federal University of Santa Catarina, Postal Code 88040-900, Florianopolis – SC, Brazil.

Abstract: In the present work, an analysis of incompressible, turbulent and isothermal flow through a radial diffuser representing a refrigeration compressor valve is numerically performed using the finite volume and the large eddy simulation methodologies, associated with the classical Smagorinsky model. Numerical results are obtained for Reynolds numbers between 7000 and 14000, considering dimensionless gaps between disks equal to 0.05 and 0.10. Results for pressure distribution on the valve reed are obtained utilizing different Smagorinsky constants that are adjusted to fit the numerical results to experimental data available in the literature. The Smagorinsky model showed acceptable to analyze this kind of flow, mainly because its easy computational implementation.

Keywords: Compressor, Turbulent Flow, Radial Diffuser, Large Eddy Simulation, Smagorinsky Model.

1. INTRODUCTION

Reciprocating compressors are responsible for fluid flow and pressure increase in most refrigeration systems. This type of compressor is able to compress gas in a wide range of pressure ratios within a large range of flow rates, and can be found in domestic refrigeration systems as well as in industries (Souto, 2002).

The compression process in refrigeration reciprocating compressors is produced by the alternative linear movement of a piston. The suction and discharge valves are responsible for the refrigerant gas retention and flow from the suction chamber to the cylinder and from the cylinder to the discharge chamber. The appropriate design of the valve system is of fundamental importance for enhancing the compressor efficiency. The pressure difference between the cylinder and suction and discharge chambers is responsible for opening the valves. Once opened, the valve dynamics is controlled by the flow pressure field acting on the valve surface and by the reaction force of the valve. Therefore, a comprehensive understanding of the flow through the valves is essential in order to enhance the valve working and, consequently, the compressor efficiency. 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. In the present work the valve is also represented by a radial diffuser, whose geometry is represented in Fig. 1.

Souto (2002) presents a good literature review about radial diffuser flows with application in refrigeration compressor valves. Works involving numerical solutions of incompressible and laminar flows have been performed by Hayashi et al. (1975), Raal (1978), Piechna and Meier (1986), Ferreira et al. (1987), Deschamps et al. (1987), Ferreira et al. (1989), Langer et al. (1990), Gasche (1992) and Possamai et al. (1994). Numerical solutions for incompressible turbulent flows have been accomplished by Deschamps et al. (1988), Deschamps et al. (1996), and Colaciti et al. (2007). 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). From this literature review one can verify that the large eddy simulation methodology has not yet been used to analyze this type of flow.

In this work, the finite volume and large eddy simulation (LES) methodologies together with the classical Smagorinsky model are used to obtain the velocity and pressure fields for turbulent flows through the radial diffuser. Results for pressure profile on the front disk, an important data to analyze the acting force, are obtained for Reynolds number ranging from 7000 to 14000 and for dimensionless gaps between disks equal to 0.05 and 0.10. The numerical results are compared with experimental data obtained by Souto (2002), adjusting the Smagorinsky constant for obtaining the best results.

2. MATHEMATICAL MODEL

Figure 1 shows a scheme of a radial diffuser and the main parameters of this geometry: valve reed or front disk diameter (D), orifice diameter (d), valve seat height (l), and the gap between the valve reed and valve seat (h).

The equations of momentum and continuity applied to an incompressible and isothermal flow, written in the cylindrical coordinate system are given by:

$$\frac{1}{r}\frac{\partial(rv_r)}{\partial r} + \frac{\partial(v_z)}{\partial z} = 0 \tag{1}$$

$$\rho\left(\frac{\partial v_r}{\partial t} + v_r\frac{\partial v_r}{\partial r} + v_z\frac{\partial v_r}{\partial z}\right) = -\frac{\partial p}{\partial r} + \mu\left(\frac{\partial}{\partial r}\left(\frac{1}{r}\frac{\partial}{\partial r}(rv_r)\right) + \frac{\partial^2 v_r}{\partial z^2}\right) \tag{2}$$

$$\rho\left(\frac{\partial v_z}{\partial t} + v_r\frac{\partial v_z}{\partial r} + v_z\frac{\partial v_z}{\partial z}\right) = -\frac{\partial p}{\partial z} + \mu\left(\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial v_z}{\partial r}\right) + \frac{\partial^2 v_z}{\partial z^2}\right) \tag{3}$$

In these equations ρ and μ are the density [kg/m^3] and the absolute viscosity [Pa.s] of the fluid, respectively; p is the pressure [Pa]; and v_r and v_z are the components of the velocity [m/s] in directions r and z, respectively.

The boundary conditions used to solve this problem are: uniform velocity at the valve inlet; zero velocity gradient at the diffuser outlet region; symmetry boundary condition at the centerline; non-slip boundary conditions at solid surfaces. The solid region of the valve seat is treated as a fluid with infinite viscosity.
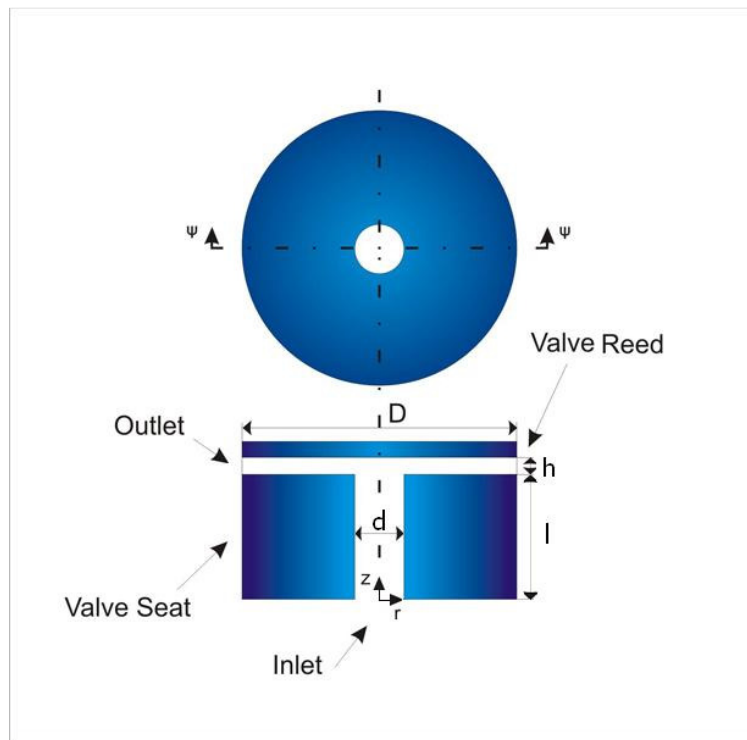

Figure 1. Geometry of the radial diffuser.

2.1 Numerical Solution Methodology

In this work, the finite volume methodology is used to solve the flow through the valve geometry. This method is extensively applied to solve fluid dynamics and heat transfer problems numerically. 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 each governing partial differential equation is integrated in these control volumes and the resulted equations are discretized resulting in a linear algebraic equation system. Finally, the algebraic equation system is solved to obtain the dependent variables. In this case, the components of the velocity field, v_r and v_z, and the pressure field, p.

The TDMA (*Tri-Diagonal Matrix Algorithm*) algorithm is used to solve the algebraic equation systems. The pressure-velocity coupling is solved by the algorithm SIMPLE-*Semi Implicit Method for Pressure-Linked Equations*.

The LES-*Large Eddy Simulation* methodology, with the classical Smagorinsky model, was used to treat the turbulence. Basically, in this methodology a turbulent viscosity, v_t, given by Equation 4, is added to the absolute viscosity.

$$v_t = (C_s l)^2 \sqrt{S_{ij}S_{ij}} \tag{4}$$

In this equation, S_{ij}, represents the deformation rate of the flow. The characteristic length, l, is calculated as a function of the mesh size. The Smagorinsky constant, $C_S=0.18$, was analytically determined for a homogeneous and isotropic turbulence. However, this constant value has been adjusted according to the application.

The numerical simulations were accomplished considering two diameter ratios, D/d (1.66 and 3), and two dimensionless gaps between valve seat and reed, h/d (0.05 and 0.1). Four different meshes, shown in Figs. 2 and 3, were used to analyze turbulent flows with Reynolds numbers ranging from 7000 to 14000.
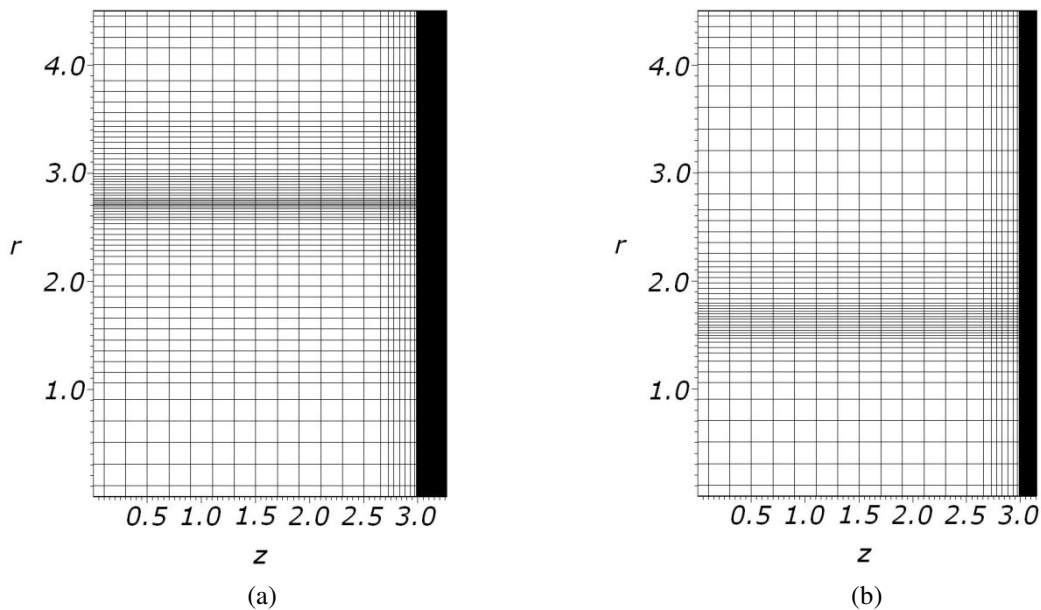

(a)   (b)

Figure 2. Coarse meshes used in the numerical simulations (45x49): (a) D/d=1,66 and (b) D/d=3.
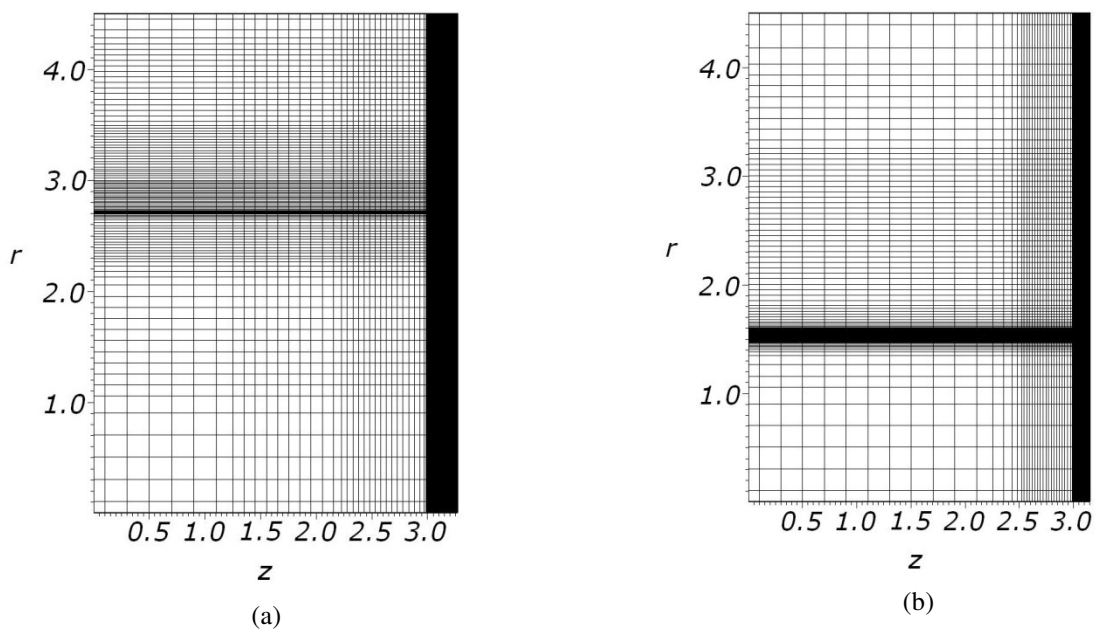

(a)   (b)

Figure 3. Fine meshes used in the numerical simulations (59x93): (a) D/d=1,66 and (b) D/d=3.

3. NUMERICAL RESULTS

The figures in this section show comparisons between numerical results and experimental data of dimensionless pressure distribution on the surface of the valve reed, P_{adm}, for two diameters rations, D/d=1,66 and D/d=3, and two dimensionless gaps, s/d=0.05 and s/d=0.1, considering Reynolds numbers varying from 7000 to 14000. The dimensionless pressure is given by the Equation (5).

$$P_{adm} = \frac{P}{1/2 \rho U_{in}^2}$$ (5)

where U_{in} is the uniform velocity prescribed at the inlet of the flow.

Firstly, experimental data are compared with numerical results obtained with and without turbulence modeling (direct simulation). In these cases, the Smagorinsky constant for isotropic and homogenous turbulence, that is, C_s=0.18, was used for obtaining the numerical results with turbulence modeling. After that, the same experimental data are compared with numerical results obtained with turbulent modeling utilizing different Smagorinsky constants.

In the following, the same comparisons were performed considering finer meshes. Considering these finer meshes, the present numerical results were compared with numerical results obtained by Souto (2002). Finally, velocity field is shown for Re=14000.

3.1. Comparison between experimental results and numerical results with and without turbulence modeling.

Firstly, the experimental data from Souto (2002) were compared with numerical results obtained by direct simulations, that is, without turbulence modeling, and with turbulence modeling using a Smagorinsky constant equal to 0.18. The coarse meshes depicted in Fig. 2 were employed in all cases. Figs. 4 and 5show results for D/d=1.66 and D/d=3, respectively.
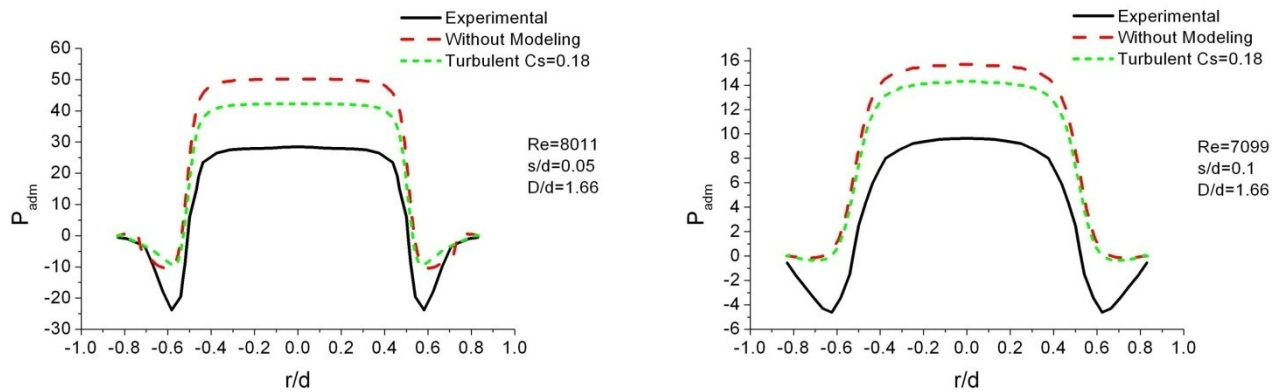

Figure 4. Dimensionless pressure distribution on the reed for D/d=1.66.

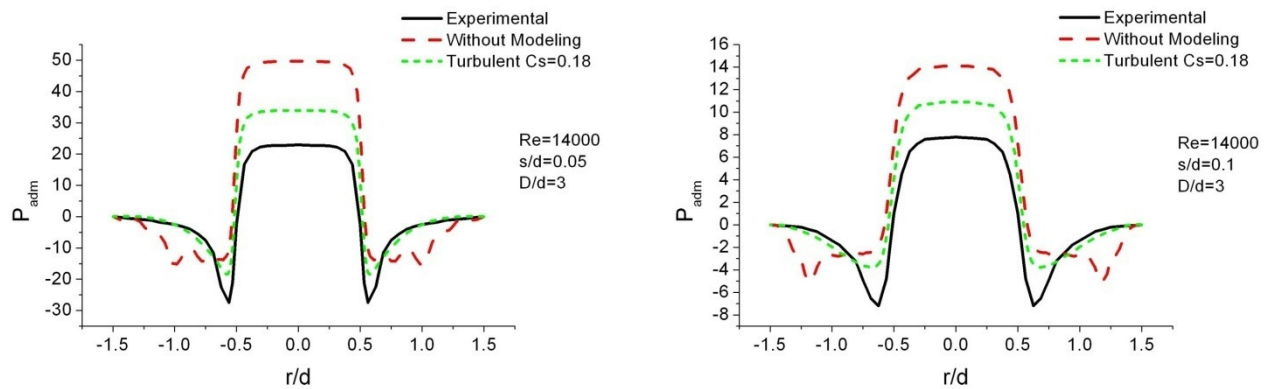

Figure 5. Dimensionless pressure distribution on the reed for D/d=3.

From these figures one can observe that the numerical results are qualitatively similar to the experimental data. Three main characteristics can be noticed in the pressure distribution: there is a central region in which the pressure is practically constant due to the stagnation of the fluid; in the inlet region of the diffuser (region between the reed and the

valve seat) the pressure decrease abruptly due to the fluid acceleration; after that, the pressure increase again because the reduction of the velocity caused by the increase of the cross section area. Analyzing the results showed on Fig. 4, to Reynolds number equals 8011 and 7099, it is observed that the quantity analysis revels that the obtained numerically results do not get near from the experimental data, considering or not the use of turbulence modeling.

It can be also observed that in some cases (Re=8011 and 1400) there are pressure oscillations in the diffuser region when direct simulation was used. This was so because in these cases the same convergence criterion was not reached. Converged numerical results were obtained only when the turbulence modeling was employed. However, the numerical results obtained by using the turbulence modeling with C_s=0.18 were not accurate compared to the experimental data. In order to improve the numerical results, different Smagorinsky constants were used and the results are presented in the following item.

3.2. Comparison between experimental results and numerical results for different Smagorinsky constants.

Figures 6 and 7 depict results for the same cases analyzed previously. However, several Smagorinsky constants were used in order to have better numerical results.
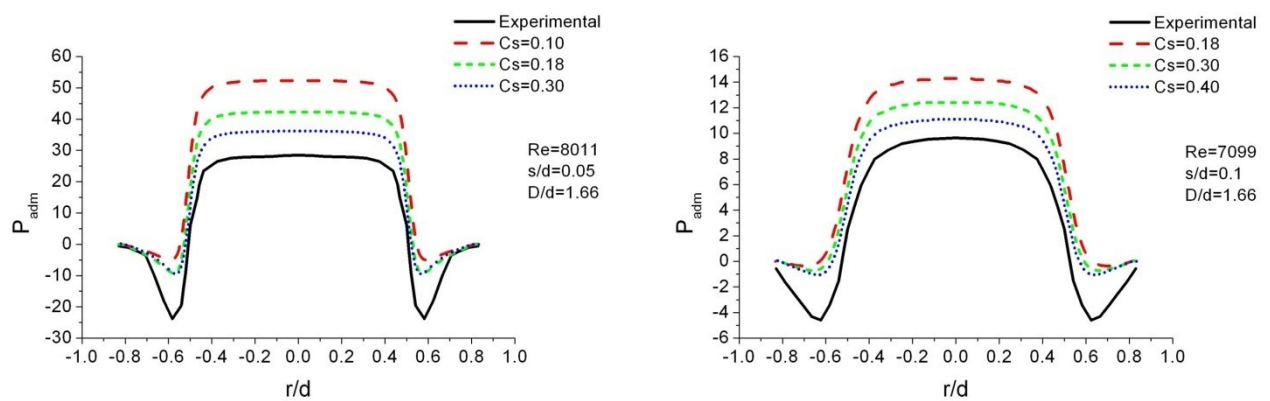

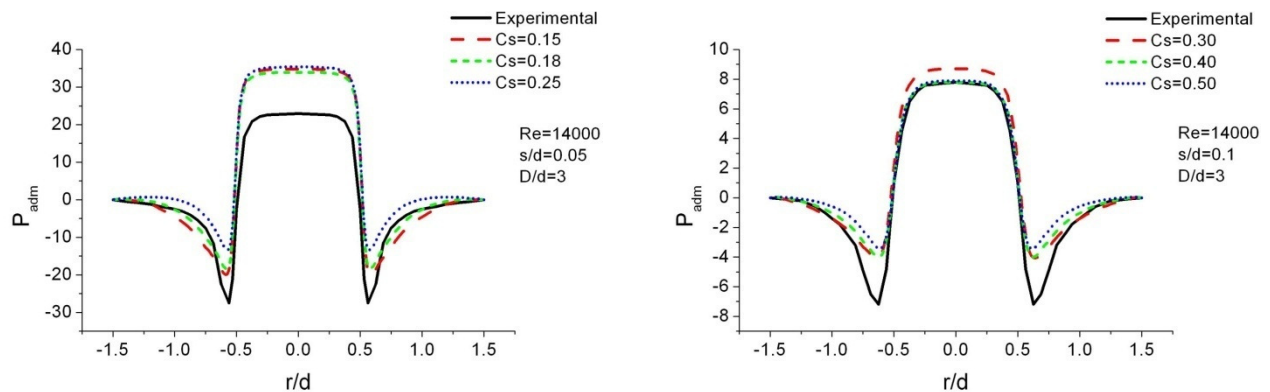Figure 6. Influence of the Smagorinsky constant in the dimensionless pressure on the reed for D/d=1.66.

Figure 7. Influence of the Smagorinsky constant in the dimensionless pressure on the reed for D/d=3.

As can be seen from these figures, it is possible to vary the Smagorinsky constant in order to enhance the comparison between the numerical results and the experimental data, mainly for the stagnation region of the flow. However, at the inlet region of the diffuser, the pressure drop does not reach the low values of the experiments. Part of this discrepancy should be attributed to the numerical diffusion caused by interpolation scheme used in the numerical code (power-law).

In addition, it can be noticed that for each flow configuration there is a specific Smagorinsky constant that produces the best agreement between the numerical results and the experimental data. Therefore, it can be concluded that there is not an optimal Smagorinsky constant valid for all flow configurations.

3.3. Comparison between experimental results and numerical results for the finer meshes.

The same flow configurations were simulated using the finer meshes depicted in the Fig. 3, and the results are presented in Figs. 8 and 9.
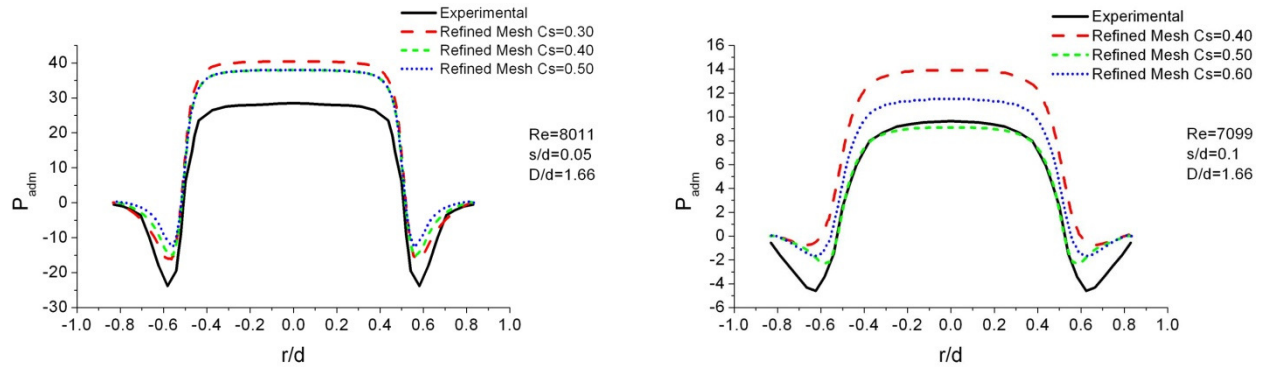

Figure 8. Dimensionless pressure on the reed for D/d=1.66, using the finer meshes.

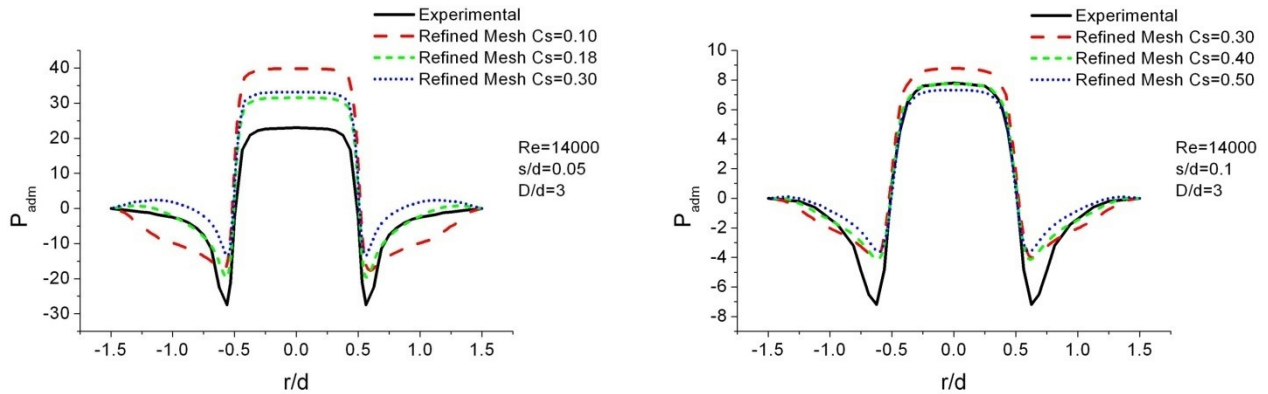
Figure 9. Dimensionless pressure on the reed for D/d=3, using the finer meshes.

Analyzing the results of Figs. 8 and 9, one can conclude that the mesh refining produced numerical results closer to experimental data, except at the inlet region of the diffuser, where the results still need to be improved. In addition, it can be noticed that the optimal Smagorinsky constants changed with the mesh.

3.4. Comparison between the present numerical results and the numerical results obtained by Souto (2002).

Figures 10 and 11 show numerical results obtained by using Large Eddy Simulation methodology compared with numerical results obtained by Souto (2002), which uses RNG k-ε model for treating the turbulence. Experimental results are also included for comparison reasons.

One can observe from these results that the turbulent modeling methodology used by Souto (2002) produced better agreement with the experimental data, mainly for the diffuser region, that is, the region between the reed and the valve seat. Souto (2002) have used QUICK-Quadratic Upwind Interpolation for Convective Kinetics as interpolation scheme, which possesses less numerical diffusion than the power-law interpolation scheme. This can explain the better results of Souto (2002) at the inlet region of the diffuser region. In the flow stagnation region, however, the Large Eddy Simulation methodology produced better agreement with the experimental data.
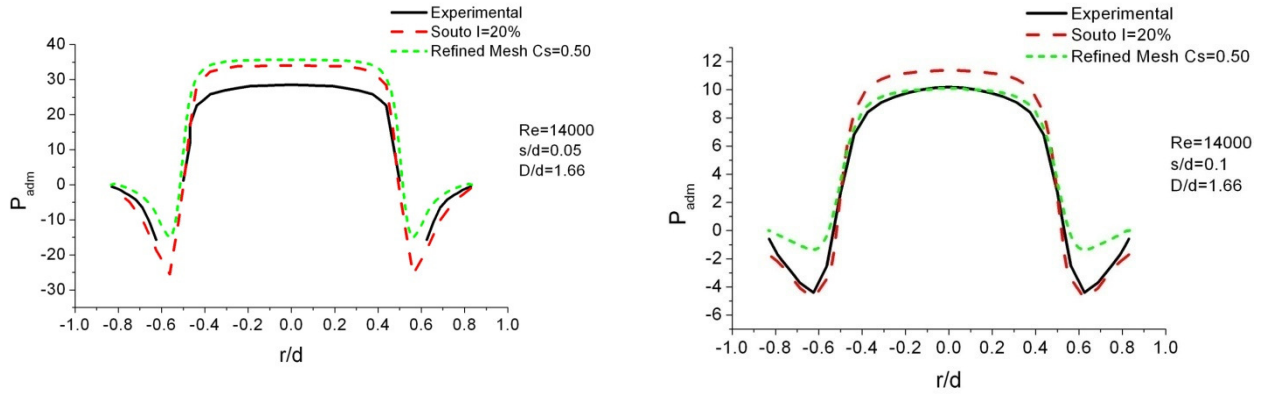
Figure 10. Comparison of the dimensionless pressure on the reed between the numerical results obtained by Souto (2002) and the present methodology for D/d=1.66.
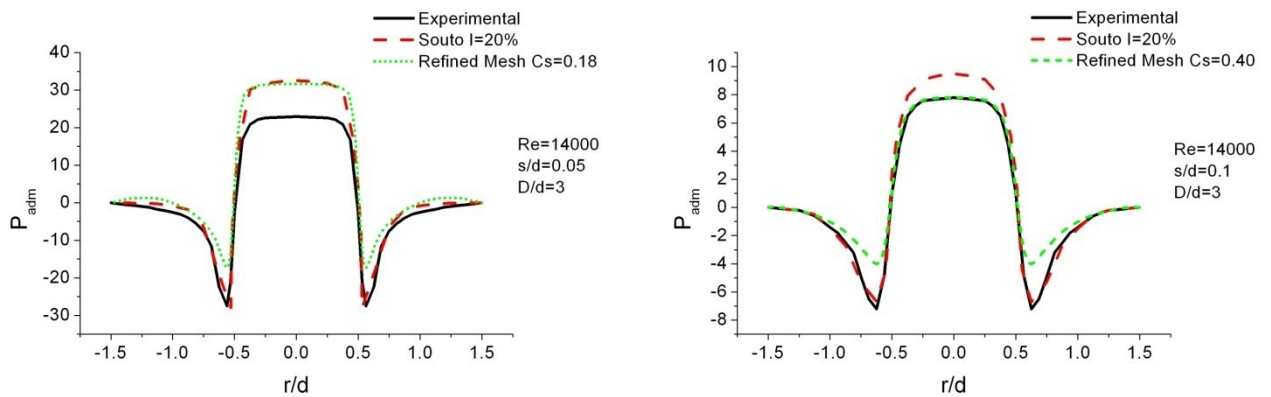

Figure 11. Comparison of dimensionless pressure profiles on the reed of numerical results for D/d=3.

3.5. Velocity and turbulent viscosity fields.

Figure 12 shows the velocity field in the radial diffuser for Re=14000. One can observe that at the inlet of the diffuser region, the velocity gradients are higher, as expected. It can also be noticed that in the exit region of the diffuser, the velocity field is practically constant in the r direction, which justify the use of completely developed flow at the exit of the diffuser as boundary condition.
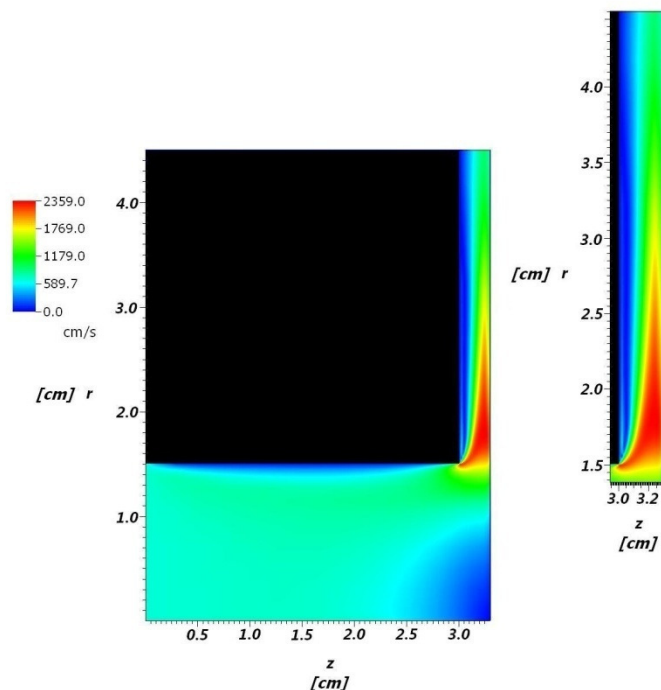
Figure 12. Velocity field in the radial diffuser for Re=14000, s/d=0.1 e D/d=3.

4. CONCLUSIONS

In the present work, a numerical analysis of the incompressible, turbulent and isothermal flow through a radial diffuser was accomplished. The radial diffuser was chosen for representing the compressor valve system used in refrigeration systems. The turbulent flow was solved by using the finite volume methodology coupled with the large eddy simulation methodology and the classical Smagorinsky model. Numerical results were obtained for Reynolds numbers varying for 7000 to 14000, considering dimensionless gaps between the reed and the valve seat equal to 0.05 and 0.10. Besides, two diffuser geometries were used, one with diameter ratio, D/d, equal to 1.66 and another one with D/d=3. Numerical results for pressure distribution on the reed were obtained using several Smagorinsky constants which were adjusted to improve the agreement between the numerical results and experimental data from Souto (2002). In general, the large eddy simulation methodology with the Smagorinsky model produced good results when compared with the experimental data, mainly for the flow stagnation region. RNG k-ε model. The advantage of the present methodology is its computational implementation, which is easier than the RNG k-ε model. In addition, the computational cost of the large eddy simulation is lower than the RNG k-ε model.

5. REFERENCES

Colaciti, A.K, López, L.M.V., Navarro, H.A., Gómez, L.C., (2007), Numerical Simulation of a Radial Diffuser Turbulent Flow, *App. Math. Comp.*, Vol. 189-2, p. 1491-1504.

Deshamps, C. J.; Ferreira, R. T. S.; Prata, A. T., 1987, Analysis of the Orifice Length in Radial Diffuser Flows, *Brazilian Congress of Mechanical Engineering (in Portuguese)*, Florianópolis-SC, p. 335-338.

Deschamps, C. J.; Ferreira, R. T. S.; Prata, A. T., 1988, Application of the k-ε Model to the Flow in Compressor Valves, *Brazilian Congress of Engineering and Thermal Science (in Portuguese)*, p. 259-262.

Deschamps, C. J.; Prata, A. T.; Ferreira, R. T. S., 1996, Turbulent Flow Through Reed Type Valves of Refrigerating Compressors, *ASME, Int. Mech. Eng. Congress and Exhibition*, Atlanta, USA.

Ervin, J.S.; Suryanarayana, N.V.; Chai, NG, H., 1989, Radial Turbulent Flow of a Fluid Between Two Coaxial Disks, *J. Fluids Eng.*, 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, *Int. Comp. Eng. Conf. at Purdue*, West Lafayette, USA, p. 632-646.

Ferreira, R. T. S.; Prata, A. T.; 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, Laminar Flow Through Eccentric Valves of Refrigeration Compressors, *Master Thesis (in Portuguese)*, Federal University of Santa Catarina, Florianópolis, 149 p.

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

Langer, C.A.; Prata, A.T.; Ferreira, R.T.S., 1990, Separation and Bifurcation of Radial Diffuser Flows, *Brazilian Congress of Engineering and Thermal Science (in Portuguese)*, Itapema-SC, p. 357-362.

Piechna, J. R.; Meier, G. E. A., 1986, Numerical Investigation of Steady and Unsteady Flow in Valve Gap, *Int. Comp. Eng. Conf. at Purdue*, p.1-14.

Possamai, F. C.; Ferreira, R. T. S.; Prata, A. T., 1994, Pressure Distribution in Laminar Radial Flow Through Inclined Valve Reeds, *Master Thesis (in Portuguese)*, Federal University of Santa Catarina, Florianópolis, Brazil.

Raal, J.D., 1978, Radial Source Flow Between Parallel Disks, *J. Fluid Mech.*, vol. 85, part 3, p. 401-416.

Souto, L.E.M., 2002, Investigação Experimental Investigation of Turbulent Flow in Compressor Valves, *Master Thesis (in Portuguese)*, Federal University of Santa Catarina, Florianópolis-SC, 105 p.

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

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

6. RESPONSIBILITY NOTICE

José Luiz Gasche and Bruno Ferreira Pussoli are the only responsible for the printed material included in this paper.