Influence of Suction Muffler Design on the Suction Effective Flow and Force Areas of a Reciprocating Compressor

Tadeu Tonheiro Rodrigues[1*],

[1]Nidec GA, R&D,
Joinville, Santa Catarina, Brazil
tadeu.t.rodrigues@nidec-ga.com

ABSTRACT

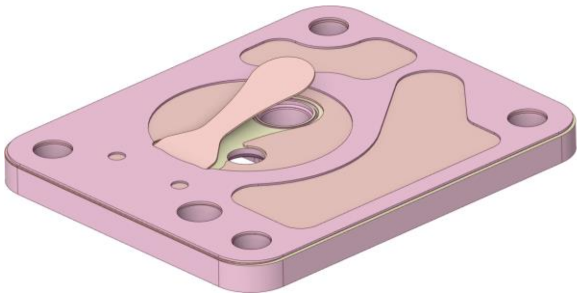
The concept of effective flow and force areas is commonly used to evaluate the performance of the suction and discharge processes in reciprocating compressors, which is strongly dependent on the flow head loss through the orifice-valve pair. The typical approach considers uniform and homogeneous flow at the port inlet. However, for the suction process, the suction muffler disturbs the flow patterns towards the orifice generating, for instance, recirculation structures. This work studies the effect of the suction muffler in the effective flow and force areas during the suction process of a reciprocating compressor. A CFD model is applied for this purpose. The results show that the suction muffler must be taken into consideration when evaluating both effective areas. The analysis also indicates that the optimization of the suction effective areas can be obtained by redesigning the suction muffler rather than the suction port and the valve.

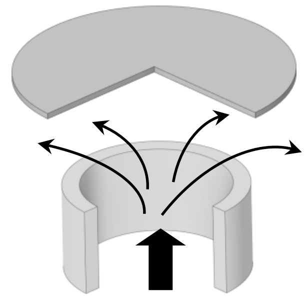
1. INTRODUCTION

Reciprocating compressors performance is strongly dependent on the behavior of the suction and discharge valve system. Based on this the advance on the valve design is one the main pillars for compressor development and competitive advantage, leading to extensive research in the academia and industry alike.

Early research efforts focused on identifying the main flow patterns that influences the valve performance, the subject was studied experimentally by Wark and Foss (1984), Ferreira and Driessen (1986), Tabakabai and Pollard (1987), Ervin et al. (1989) and Gasche et al. (1992). However, it was in the phenomena-modelling field that is observed the majority of the contributions. At first combination of empirical and theoretical approaches was employed by Boswirth (1982), researchers advanced to numerical steady flow simulation in the simplified geometry of the radial diffuser (Ferreira et al. 1989, Gasche et al. 1992, Deschamps et al. 1996, Possamai et al. 2001). Finally with advance of numerical methods, physics integration and powerful hardware availability the complete three-dimensional unsteady turbulent flow simulation of actual valves geometry is solved accounting its fluid-structure interaction nature using simplified single degree of freedom spring-mass system models to account the valve dynamics, works in this subject are found in Pereira et al. (2007), Pereira et al. (2008), Mistry et al. (2012) and Rodrigues (2012). Approaches on the sight of two-way fluid structure interaction, where both fluid and solid domains are solved by discretizing both fluid and solid governing equations, are presented in Kim et al. (2006), Kim et al. (2008) and Silva and Arceno (2014).

One can observe the majority of the studies with focus on numerical simulation or experimental test on the valves in order to obtain the effective flow and force areas to be used in simplified compressors models based on lumped formulation. The fundamental aspects of the fluid flow in valves was developed using the radial diffuser geometry, as depicted in Figure 1, which is a simplification of the actual valve design and the boundary conditions. This approach considerers the physical domain limited to the pair orifice and valve, where the inlet flow in the orifice is taken as uniform.



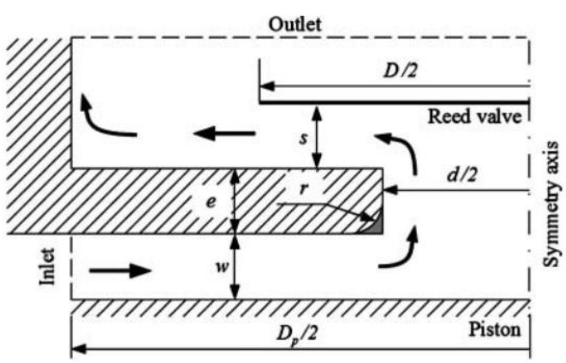


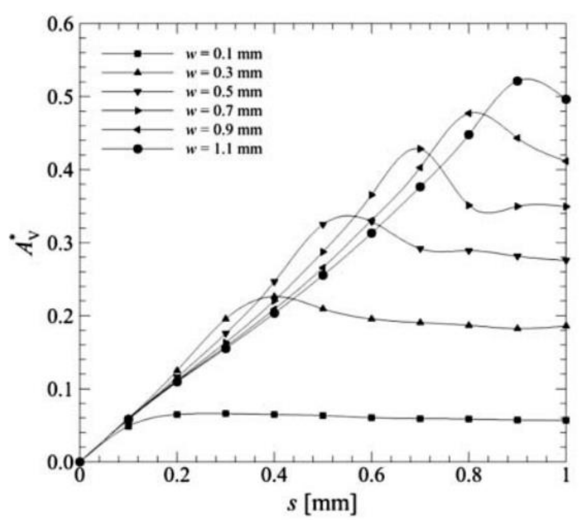
(a) Typical valve system geometry (b) Radial Diffuser

Figure 1: Radial flow diffuser simplification used to study flow through compressor reed valves.

The main advantage of this procedure compared to complete unsteady three-dimensional flow simulation is the reduced computational time, for this reason it still found examples of applied research on lumped model formulation due its good agreement with experimental data obtained with fast computation. Also, is where detail and rich data about the valve flow subtleties is documented, useful as design guidelines regardless the engineering method. In this sense, further advances in valves research can be obtained by removing the idealizations firstly established with radial diffuser airflow approach, as interactions with other compressor components which are potential sources of disturbances for the inlet flow in the valve system influencing the effective flow and force areas of the valves.

As example Lange and Deschamps (2011) through numerical simulation studied the influence of the piston position on the effective flow and force areas of discharge valves (Figure 2.a). They found the proximity between the piston and the valve plate introduces significant viscous friction loss associated with the radial flow through the very narrow clearance left in the cylinder when the piston is close to the top dead center (Figure 2.b), reducing the effective flow area. Additionally it was observed the flow exiting the clearance influences the recirculation structures in the discharge port and on the valve seat.





(a) Discharge valve flow domain with piston. (b) Effective flow area curves.

Figure 2: Piston position influence on the discharge valve effective flow areas (Lange and Deschamps, 2011)

The suction valve system inlet mass flow rate is typically supplied by a piping system known as suction muffler, designed to attenuate the noise generated by the suction valve. For that, the advance in understanding the flow patterns of the suction valve system as well the prediction of effective flow and forces areas requires customize the engineering methodologies to the particularities of the suction system.

The main aim of the present work is to extend the investigation of effective flow and force areas of suction valve system by accounting the effects of the suction muffler outlet which is source of disturbances for the inlet flow on the suction orifice.

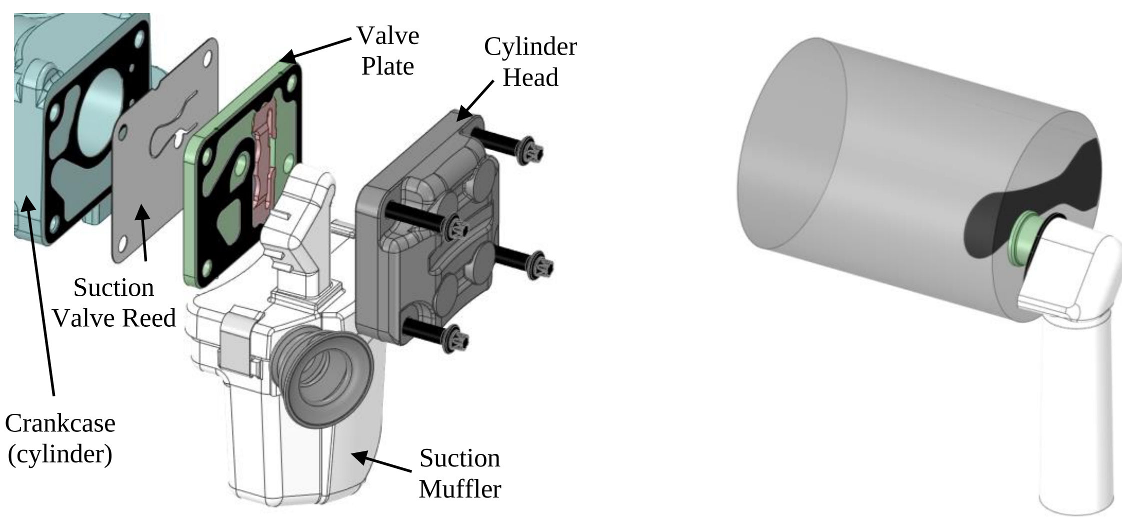
The study was carried numerically using four state of art designs and comparing the effective areas behavior with radial diffuser results, correlations with the main flow patterns are also described. The numerical results evidences the suction muffler must take into consideration to predict precisely the effective flow and force areas, furthermore explore the suction muffler design to reducing the flow disturbances transmitted to the suction orifice and valve should be accounted to improve the flow through the valve decreasing head losses which is beneficial to compressor performance

2. NUMERICAL PROCEDURE

The numerical procedure is well known in the reciprocating compressor industry and academy, for this reason will be briefly described.

2.1 Geometry Simplification

The exploded view of a typical valve system with suction and discharge chambers is available in Figure 3.a. The physical domain for the simulation of the fluid flow through the suction valve is obtained selecting the muffler outlet tube, suction orifice, suction valve and the cylinder followed by volume of fluid extraction, as presented in Figure 3.b.



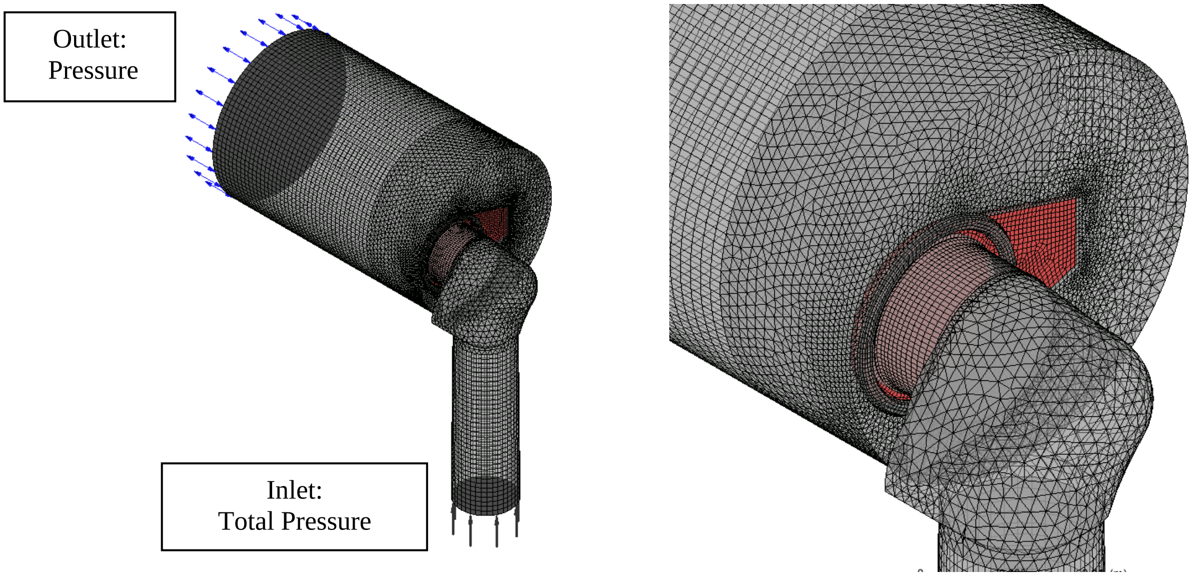
(a) Solid domain. (b) Extracted fluid domain.

Figure 3: Ilustration of the fluid flow phsyical domain construction.

2.2 Numerical Model

The steady, compressible, turbulent and isothermal fluid flow through suction valve system was solved using the commercial code ANSYS CFX® release 2022, which employs the element-based finite volume method to discretize the partial differential equations of the mass conservation, momentum, and turbulence. Advection terms were discretized using a High Resolution Scheme. The Shear Stress Transport model was set to bridge the turbulent effects to the mean flow, which has been proven as a suitable model for compressor simulation (Rodrigues, 2014). The resultant system of equations is solved through Incomplete LU (ILU) decomposition, algebraic multigrid method and coupled strategy. The pressure evaluation correlation is obtained from the Redlich-Kwong library, available in the numerical tool.

Boundary conditions and mesh strategy are described using Figure 4. At the inlet, located in the suction tube, is prescribed Total Pressure and at the outlet located in the cylinder bottom dead center is prescribed pressure. The pressure boundary conditions are based on suction pressure for ASHRAE LBP for inlet and a pressure loss at the outlet was prescribed based on experimental data, for this presented study the refrigerant is propane. In order to obtain a high quality mesh the physical domain was divided in geometries to maximize the hexahedron elements on the computation domain.



(a) Boundary Condition. (b) Mesh magnified view.

Figure 4: Mesh and boundary conditions used for simulations.

2.3 Post-processing – Effective Areas

The well know effective flow and force areas formulations are presented in Equations 1 and 2.

$$A_v = \dot{m}_v / \left[p_{up} \cdot \sqrt{\frac{2k}{(k-1)\cdot R \cdot T_{up}}} \cdot \sqrt{\left(\frac{p_v}{p_{up}}\right)^{\frac{2}{k}} - \left(\frac{p_v}{p_{up}}\right)^{\frac{k+1}{k}}} \right] \quad (1)$$

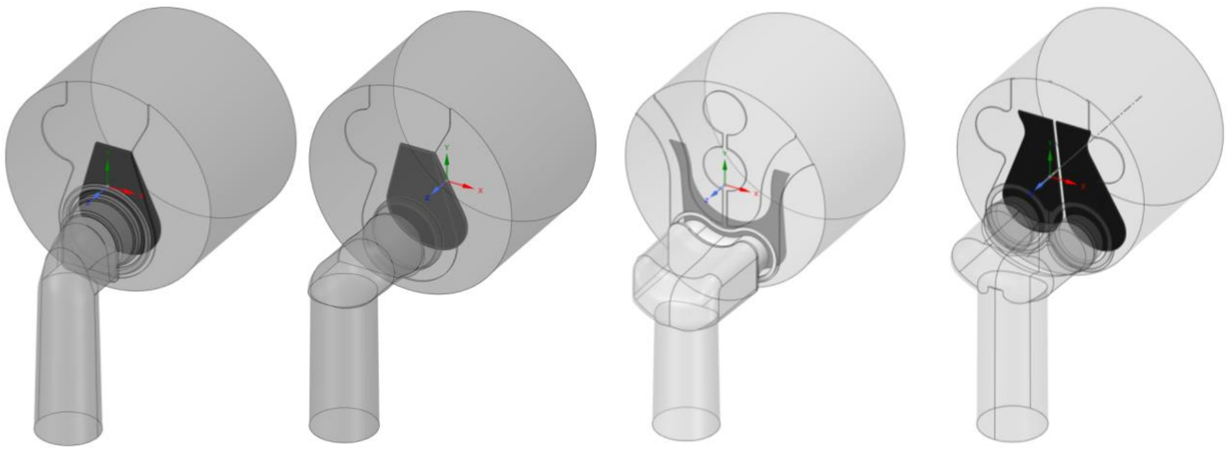
$$A_F = \frac{F_v}{P_{up} - P_v} \quad (2)$$

T_{up} is the upstream flow temperature, while p_{up} and p_v are the upstream and downstream pressure, respectively, imposed as boundary conditions. R is the gas constant and k is the specific heat ratio. $\dot{m}_v$ is the mass flow rate through the valve and F_v is the resulting flow force on the valve, both are obtained from the integration of flow fields resulting from simulation

2.4 Geometries Evaluated

The four suction system designs numerically studied are illustrated in Figure 5. The term suction system refers to assembly of the suction muffler outlet with the pair suction orifice-valve. The pictures of the suction systems are volume of fluid extracted from actual solid designs. The basic dimensions for fluid flow head loss are the same for a suitable comparison, as: suction orifice areas, tubes diameters and lengths, and cylinder diameter.

Design 1is a typical configuration of circular orifice and suction muffler, Design 2 is a variation of Design 1 where the outlet chamber of the suction muffler has an ending curvature in relation to muffler tube main axis. Design 3 is another common found concept, the oblong orifice. Design 4 is also a state of art concept variation with double suction circular orifices.



(a) Design 1 (b) Design 2 (c) Design 3 (d) Design 4

Figure 5: Suction system designs (muffler and pair orifice-valve).

3. RESULTS

For each design, a set of simulations was performed parameterizing the valve opening in 5 steps, simulation results for a radial diffuser of some proportions is shown for comparison purposes. The effective areas curves are normalized by the maximum results obtained in the radial diffuser simulation, while the maximum opening normalizes the valve opening. Effective flow and force areas results are presented in Figures 6 and 7, respectively.

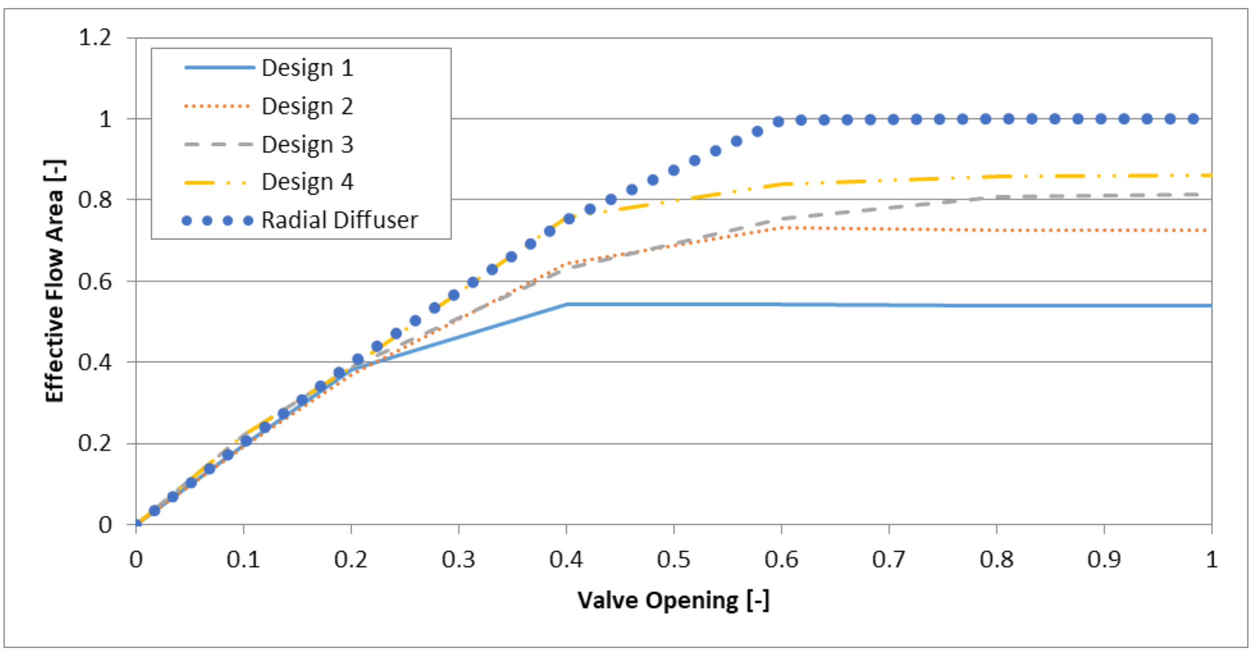


Figure 6: Effective flow area results for the investigated designs.

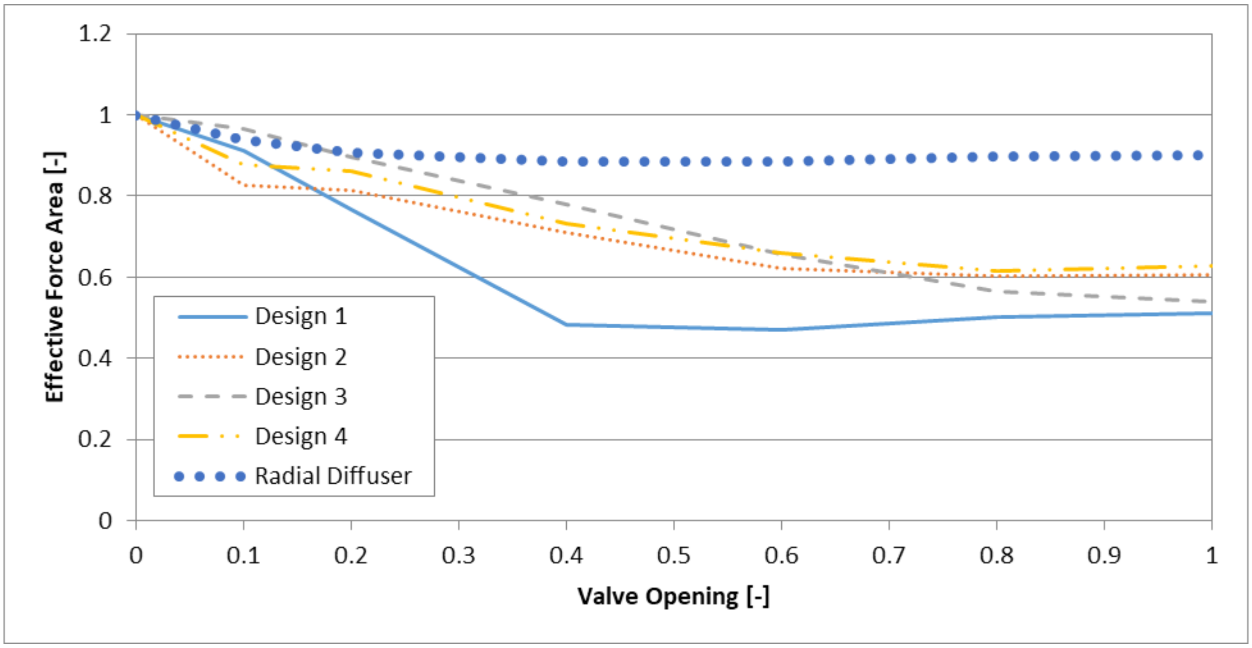


Figure 7: Effective force area results for the investigated designs.

One can note the effective flow area results improving from Design 1 to Design 4, approaching the radial diffuser data. The effective flow area results can be interpreted splitting in three valve-opening ranges:

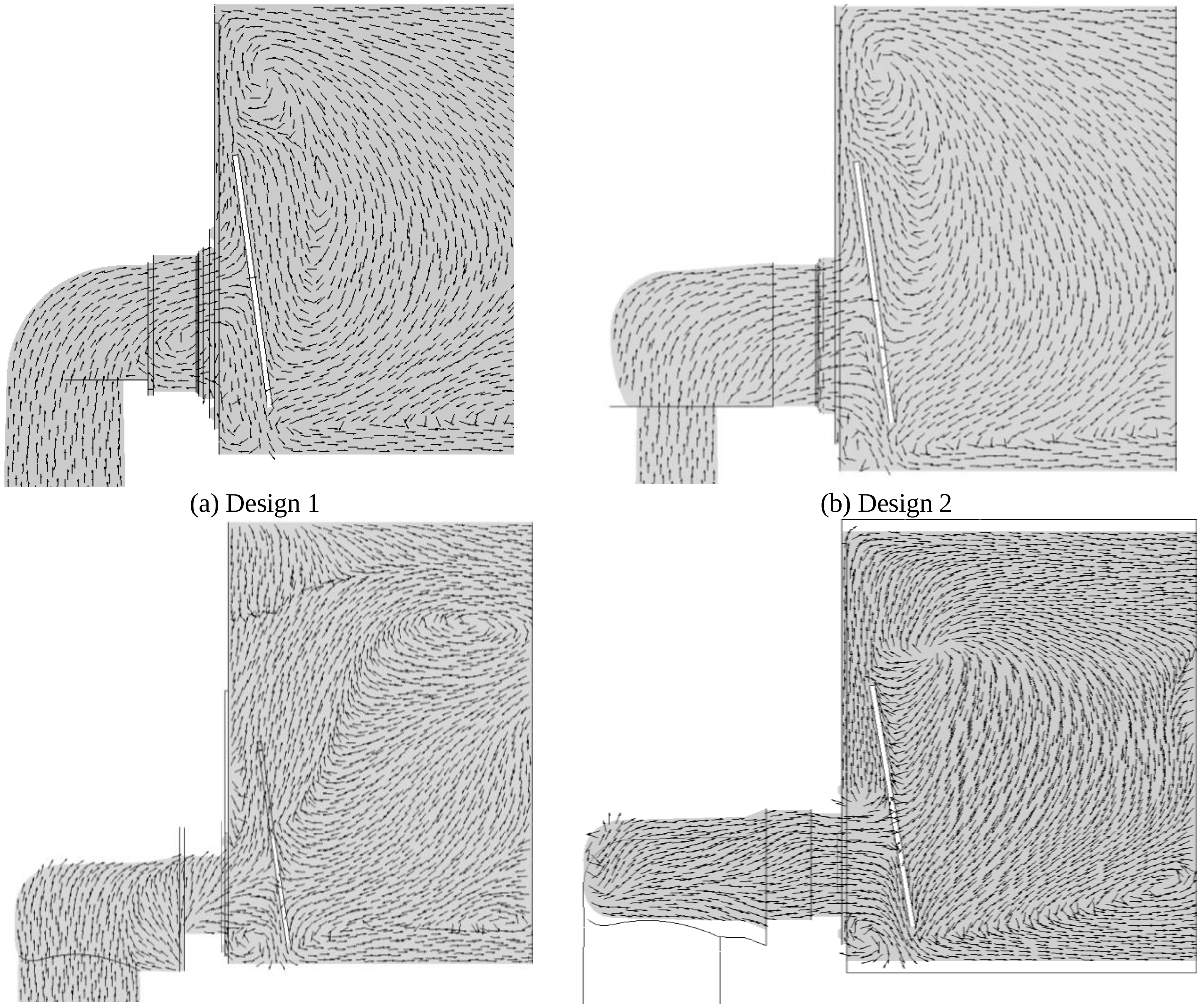
- 0 to 0.2: no difference observed among the designs and radial diffuser, the valve reed proximity dominates the head loss and consequent mass flow rate.
- 0.2 to 0.6: the increasing valve opening reduces its influence to main flow with noticeable difference among the designs. Design 4 is an exception once matches the radial diffuser until 0.4 valve opening.
- 0.6 to 1.0: saturation region characterized as a plateau for each design, where the biggest differences are found with no design achieving the radial diffuser performance. For Design 1 the drop compared to Radial Diffuser is around 55%, while the best results obtained with Design 4 still is 18% below.

Effective force areas results have a typical descent behavior with increasing valve opening once the flow disperses and reduces the resulting force on the valve. The main remarks are:

- All results are very close from valve opening 0 to 0.2, reinforcing the dominant effect of reed proximity to the orifice on the flow behavior.
- Radial diffuser detaches from all design evaluated from opening 0.2 to 1.0, with a clear dominant plateau in this range.
- Designs 2 to 4 are very similar, with Design 1 showing the lowest values specially in the valve opening range 0.2 to 0.8, approaching the remainder designs from opening 0.8 on.

In order to explain the effective flow and force area behaviors visualization of the flow patterns by using velocity vectors at the orifice cross section plain is presented in Figure 8 for the normalized valve opening of 0.6. The main highlights from the flow observation are pointed as follows:

- The suction mufflers for Design 1 and 2 creates flow detachments resulting in recirculation zones that dominates the suction orifice, these are significant disturbances and head losses sources, which is quite different from the idealized radial diffuser inlet flow.
- Design 3 minimizes the recirculation zone but still is clear oblique angle of the velocity vector field not aligned with the suction orifices longitudinal axis.
- Design 4 is a further improvement by aligning the flow with the orifice longitudinal axis, which is closer situation to radial diffuser.



(a) Design 1 (b) Design 2

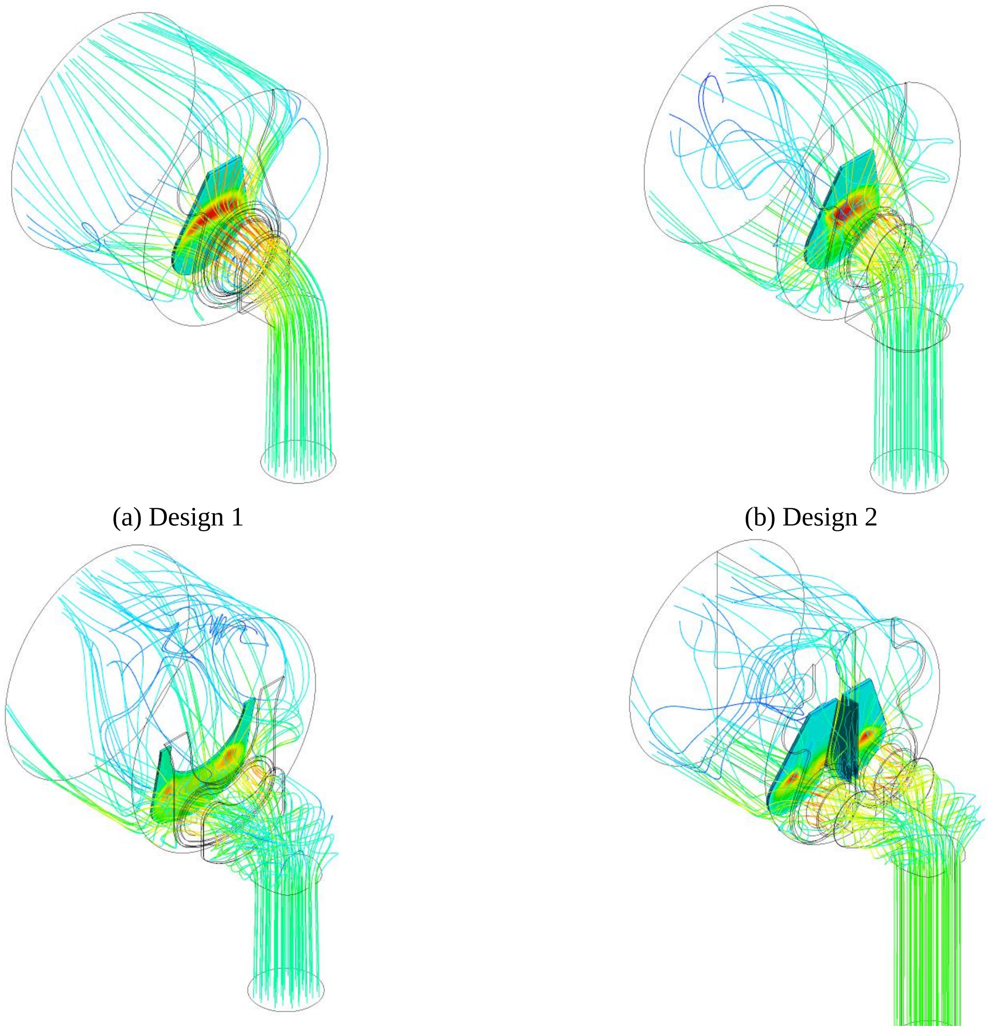
(c) Design 3 (a) Design 4

Figure 8: Velocity vectors on orifice cross sectional plane.

Observations of the flow patterns based on streamlines as presented in Figure 9 is used to complement the explanation about the major differences between actual designs and radial diffuser, pressure contour on the valve surface is also provided.

Noticeable development of three-dimensional swirling flow structures around the main orifice axis reduces the flow efficiency compared to radial diffuser. This explain why Designs 3 and 4 which improved the flow supplied to the orifice (based on Figure 8) did not achieved the radial diffuser results, due to the rotational flow component introduced the by suction muffler outlet.

The instabilities afore described influences the pressure distribution on the valve. Designs 1 and 2 due to the main recirculation zones generated in the suction orifice concentrates the flow with noticeable high pressure zones not aligned with the orifice center, which are displaced in the vertical axis. Design 3 and 4 also presents high-pressure zones misaligned with the orifice center and displaced in the horizontal axis, in this case influenced mostly by the swirling structures generated by the suction muffler.



(a) Design 1 (b) Design 2

(c) Design 3 (a) Design 4

Figure 9: Streamlines with pressure contours on the valve surface.

4. CONCLUSIONS

A numerical study of the influence of the suction muffler on the suction effective flow and force areas was developed using typical compressors designs. The documented approach based on the extensive research in radial diffusers employs a uniform and homogeneous flow in the suction office, disregarding the interface with suction muffler which in the presented work was observed to generate disturbances in the flow supplied to the suction orifice reducing the effective flow and force areas up to 55% compared to a radial diffuser. Account the muffler design reveals as an important component in the simulation of valve system effective flow, by doing that one can avoid misleading compressor simulation results based one-dimensional and lumped models. Finally, by considering the suction muffler design to reduce disturbances is a suitable procedure to improve the flow through the valve system of actual designs and consequently the compressor performance.

REFERENCES

Boswirth, L.(1982). Theoretical and Experimental Study on Fluid Flow in Valve Channels, Parts I and II. *In: Proceedings of International Compressor Engineering Conference*. Purdue University, West Lafayette, IN USA. No. 371.

Deschamps, C.J., Ferreira, R.T.S. & 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. & 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, *Proceedings of International Compressor Engineering Conference*. Purdue University, West Lafayette, IN USA, 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.

Kim, J., Wang, S., Park, S., Ryu, K. & La, J. (2006). Valve Dynamic Analysis of a Hermetic Reciprocating Compressor, *Proceedings of International Compressor Engineering Conference*. Purdue University, West Lafayette, IN USA, No. C107.

Kim, H., Ahn, J. & Kim, D. (2008). Fluid Structure Interaction and Impact Analyses of Reciprocating Compressor Discharge Valves, *Proceedings of International Compressor Engineering Conference*. Purdue University, West Lafayette, IN USA, No. 1112.

Mistry, H., Bhakta, A., Dhar, S., Bahadur, V. & Dey, S. (2012). Capturing Valve Dynamics in Reciprocating Compressors through Computational Fluid Dynamics, *Proceedings of International Compressor Engineering Conference*. Purdue University, West Lafayette, IN USA, No. 1210.

Pereira, E.L.L. & Deschamps, C. J. (2011). Influence of piston on effective areas of reed-type valves of small reciprocating compressors. *HVAC & R Research*, v. 17, p. 218-230.

Pereira, E.L.P, Deschamps, C.J. & Ribas Jr., F.A. (2007). Performance analysis of reciprocating compressor through CFD simulation, *Proc. International Conference on Compressors and their Systems*, London: p. 310-318.

Pereira, E.L.P, Deschamps, C.J., & Ribas Jr., F.A. (2008). A comparative analysis of numerical simulation approaches for reciprocating compressor, Proceedings of International Compressor Engineering Conference. Purdue University, West Lafayette, IN USA, No. 1879.

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.

Rodrigues, T.T. (2014). Turbulence Modeling Evaluation for Reciprocating Compressor Simulation. *Proceedings of International Compressor Engineering Conference*. Purdue University, West Lafayette, IN USA, No 2297.

Silva, J.C. & Arceno, E. (2014). Correlation Between the Fluid Structure Interaction Method and Experimental Analysis of Bending Stress of a Variable Capacity Compressor Suction Valve, *Proceedings of International Compressor Engineering Conference*. Purdue University, West Lafayette, IN USA, No. 2311.

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.

ACKNOWLEDGEMENT

The author acknowledge the Nidec-GA for the technical and financial support.