2014

Tubulence Modelling Evaluation for Reciprocating Compressor Simulation

Tadeu Tonheiro Rodrigues
Embraco - Research and Development, Brazil, tadeu.t.rodrigues@embraco.com

Turbulence Modelling Evaluation for Reciprocating Compressor Simulation

Tadeu Tonheiro RODRIGUES[1]*, Diogo Lôndero da SILVA[2]

EMBRACO, Research & Development Group
Joinville, Santa Catarina, Brazil

[1]tadeu.t.rodrigues@embraco.com
[2]diogo.l.silva@embraco.com

ABSTRACT

In this work a numerical investigation is performed aiming to account the effects of the turbulence modeling in the performance of the Fluid-Structure Interaction simulation of a reciprocating compressor. An actual compressor geometry was analyzed, particularly considering the suction muffler and valve system. The global performance numerical results were compared with experimental data of the compressor operational characteristics in order to validate the approach. The analysis comprises straightforward RANS models, k-ε, SST, and traditional LES sub-grid models and the Smagorinsky and the Dynamic model of Germano, looking for the best option in terms of results reliability, convergence robustness and computational effort minimization.

1. INTRODUCTION

The design of reciprocating compressors is closely related to fluid-structure interaction due the employment of automatic reed valves, dynamically driven by pressure pulsations and fluid flow patterns. Accurate simulation predictions for the pressure pulsations are fundamental for improvements in the valve system, indicating great importance for the numerical procedure utilized.

Generally, reliable numerical results are obtained mainly by careful meshing process, high order discretization and adequate setup of boundary conditions. However, turbulent flows demands extremely fine meshes for obtain stable convergence, resulting in prohibitive hardware requirements. The answer for the limitations in simulating turbulent flows was the development of several approaches for modelling the turbulence effects, which are necessary to stabilize the numerical solution convergence for relatively coarse meshes.

A significant number of turbulence models have been proposed and in most of the cases, it is known each model fits better for specific patterns. In the area of the refrigeration compressor modelling through CFD a few works can be found about evaluation of the performance of the available turbulence models.

The very first work related to effective turbulence modelling evaluation was performed by Deschamps *et al*. (1996) using RANS methodologies. In this case the RNG k-ε model was applied to predict the flow though a radial diffuser and experimentally validated. The investigation was carried out due poor results obtained with high and low Reynolds number k-ε models in Deschamps *et al*. (1988) and Deschamps *et al*. (1989).

A dedicated investigation regarding turbulence modelling was presented in Colaciti *et al.* (2007), using the radial diffuser geometry and high Reynolds number simulations. Results were obtained for RANS eddy viscosity models k-ε, RNG k-ε and SST, as well for a Reynolds Stress BSL model. The best results were obtained for BSL and SST, the last one presents according the authors the best compromise between accuracy and convergence robustness. The k-ε based models presented worst agreement with experimental data and incorrect behaviour near the walls, although is more stable and requires less computational resources than the others.

Recently, several works in the area of large eddy simulation have gained presence although focusing on solving the simplified radial diffuser geometry for representing the valve. Numerical experimentation with several LES variations were performed by Rigola *et al.* (2006), Rigola *et al.* (2009) and Rigola and *et al.* (2012) evaluating dynamic Smagorinsky, Yoshizawa and Walle sub-grid models.

The advances in moving walls and FSI technics allied to the growing performance of hardware and parallel programing have been providing alternatives for compressor simulation, using the real geometries, instead of simplified ones, for predicting actual valve dynamics.

The most common examples regarding the full simulation of the compressor valves dynamics are related to one-way FSI, where the fluid domain is solved by discretization of the governing equations of motion while the solid domain is modelled by traditional one degree of freedom equations. Pereira *et al.* (2007), Pereira *et al.* (2008), Takemori (2008) and Mistry *et al.* (2012) presented successful works employing this methodology aiming efficiency optimization.

Approaches on the sight of two-way FSI, where both fluid and solid domains are solved by discretizing both fluid and solid governing equations are presented in Kim *et al.* (2006) and Kim *et al.* (2008) towards reliability analysis and in Lang *et al.* (2011) aiming design and efficiency improvements.

The last improvements in compressor simulation points for an increasing adoption of FSI method for designing the compressor valve system. However, there is lack of data about suitable turbulence models aiming accuracy in the FSI simulations, despite the efforts listed before for obtain reliable results in the turbulent flow through simplified valve geometries.

Observing the above revision, the present work investigates the influence of the turbulence modelling in predict a compressor suction system dynamics. The goal is compare straightforward models regarding compromise among reliable simulation, convergence and computational cost, disregarding high fidelity results like DNS.

For this purpose an actual suction and valve system geometry is simulated by the commercial package ANSYS CFX® and a one-way FSI approach, where the valve dynamics is solved by an implementation of the Euler Bernoulli beam model.

2. MATHEMATICAL FORMULATION

The solution of the three-dimensional, unsteady and compressible flow of a viscous Newtonian fluid in a suction system of hermetic compressor is governed by the mass conservation, momentum and energy equations. The closure for the equation system is performed by a real gas equation of state for the refrigerant.

Concerning issues about turbulent flow it will be presented a short overview based on practical guidelines, for a deeper involvement the reader is oriented to traditional references as Versteeg and Malalasekera (2007), Wilcox (1994) and the related developers articles cited in the text.

Reynolds Averaged Navier-Stokes Equations (RANS) and Large Eddy Simulation (LES) procedures were employed to perform the investigation. The RANS procedure is based on the Reynolds decomposition of the Navier-Stokes equations, which separates the average flow behaviour from the unsteady contributions. The LES procedure is based on the application of a spatial filter in the Navier-Stokes equations that physically consists in separate the large scales, which are fully solved, from the small scales, which are modelled by a sub-grid model.

The both class of procedures here described are eddy viscosity models based on the assumption of Boussinesq, where the turbulence effects in the mean flow solution are computed as turbulent viscosity and summed to the molecular viscosity instead of solve the turbulent Reynolds Stresses.

The RANS models employed for this study are the Standard k-ε and the Shear Stress Transport (SST). The Standard k-ε (Launder and Spalding, 1974) has proved to be suitable for most of the engineering flows, is robust and provides in the average good results, however, fails to predict swirling flows with adverse pressure gradient and is suitable for fully turbulent flows. The associated problems with the turbulence eddy dissipation ε of turbulent kinetic energy k lead to the development of the k-ω models (Wilcox, 1994). The k-ω models are based on the turbulence eddy frequency ω and are well fitted to deal with low Reynolds numbers problems without special care with the wall treatment, although are strongly dependent of the free stream boundary conditions and in these cases has presented poor resolution. The SST model (Menter, 1994) employs the k-ε and the k-ω in order to take the best of both approaches. The k-ε is applied for most of the domain far from the walls, once this model doesn't present strong sensibility to the free stream boundary conditions, although, in the vicinity of the wall the ε equation is transformed in the ω equation for accurate near wall flow prediction.

The LES sub-grid models exploited are the classical Smagorinsky (Smagorinsky, 1963) and the Dynamic model (Germano, 1991). Smagorinsky proposed an algebraic sub-grid model, which calculates the eddy viscosity through a direct relation among the strain rate, the length scale and the constant of Smagorinsky (Cs). This sub-grid model has presented good performance in a wide range of problems however it is not a general model once the constant Cs must be evaluated for each situation and in some cases cannot be applied constant for the whole domain, additionally fails in solve the flow near wall due to overestimated the eddy viscosity. The dynamic model in practice aims to overcome the Smagorinsky model limitations by evaluated in space and time the constant Cs.

3. NUMERICAL METHOD AND CASE SETUP

The turbulent fluid flow through the suction system was solved by using the commercial code ANSYS CFX® release 13, based on the element based finite volume method to discretize the partial differential equations of the mass conservation, momentum, energy and turbulence.

The suction system domain, presented in Figure 1, was discretized employing predominately tetrahedrons in order to obtain a mesh isotropy recommended for LES, moreover the mesh quality was controlled keeping the skewness factor below recommended values resulting in a mesh size around 110000 nodes. Although each turbulence models demands specific mesh refinement, this strategy was used in order to identify the effect of the turbulence model on coarse model simulations, commonly used in numerical engineering approaches.

Advection terms are discretized using a High Resolution Scheme and the temporal discretization obtained by a second order backward euler scheme. The resultant system of equations is solved through ILU decomposition, algebraic multigrid method and coupled strategy. The pressure evaluation correlation is obtained from the Redlich-Kwong library available in the numerical tool.

The simulations were performed during two full compression cycles in order to obtain periodic stabilization. The time step employed was constant and equal to 2x10-5 s, while the compressor frequency set is 50 Hz.

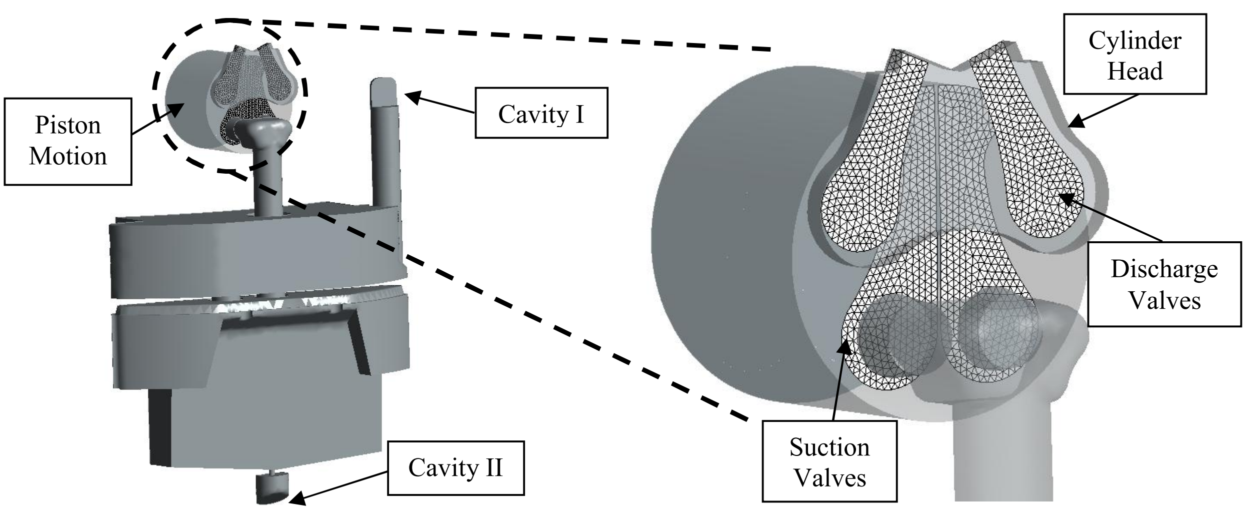

Figure 1: Suction system model and boundary conditions.

The boundary conditions are described observing Figure 1 above. In Cavity I, II and Cylinder Head opening conditions are prescribed for pressure and temperature, both values obtained experimentally. The remainder boundaries are walls with prescribed no slip condition and adiabatic. For the cylinder bottom and valves, specified displacement is imposed by a mechanism motion equation and by the solution of the one-degree freedom Euler Bernoulli beam model, respectively.

Concerning boundary conditions for turbulence closure in RANS models, a medium turbulence intensity of 5% was considered, which is commonly used for tubes. On the sigh of flow near wall default scalable and automatic functions for wall treatment were employed for k-ε and k-ω models, respectively.

4. RESULTS AND DISCUSSION

Dimensionless results for the cylinder pressure volume diagram and pressure pulsations in the suction chamber are presented in Figures 2 and 3, respectively. All numerical results are compared with experimental measurements for validation and normalized by the evaporation pressure (PSL).

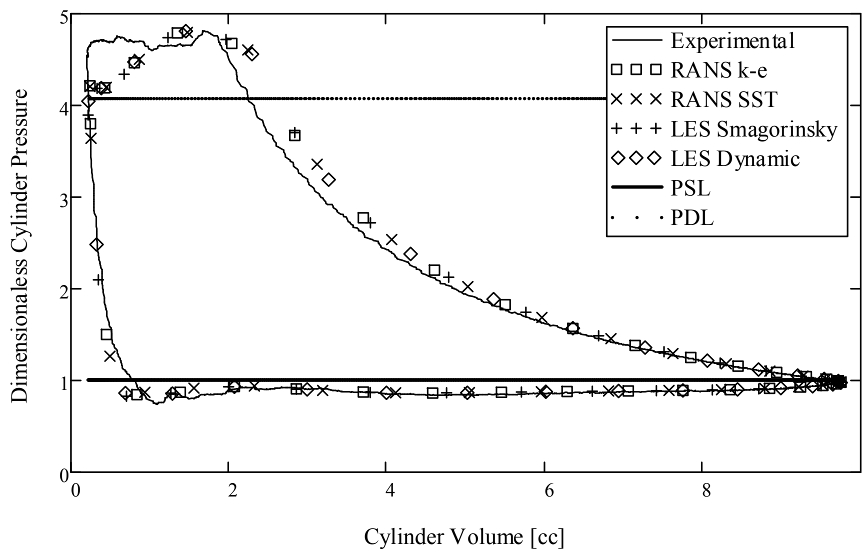

Figure 2: Cylinder pressure volume diagram.

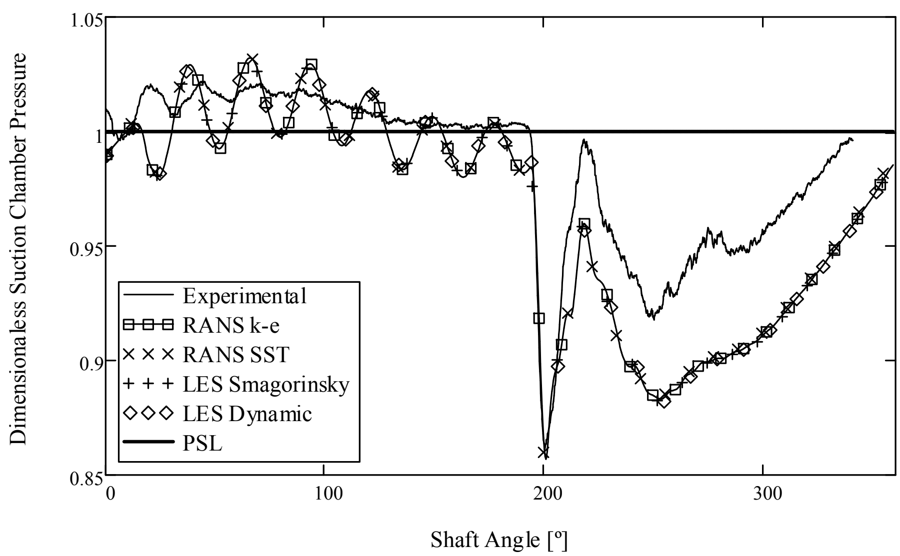

Figure 3: Suction chamber pressure pulsations.

It is observed good agreement for the cylinder pressure volume diagram, disregarding data above the condensation pressure (PDL), once the discharge system was not modelled. Similar performance is observed for the pressure pulsations in the suction chamber, mainly for agreement of the pulsation frequency. The slightly discrepancy between experimental and simulation data for shaft angle above 200º are typical and can be associated to the pressure transducer interference.

All turbulence models are able to predict with physical coherence a full compression process and presenting nearly identical results with small differences detected when signals are treated to obtain mass flow, indicated power and suction losses, presented in Table1. In addition, the difference among the models predictions is smaller than the averaged difference between the experimental and numerical results.

Table 1: Main compressor performance parameters comparison.

	Experimental	k-e	SST	Smagorinsky	Dynamic
Mass Flow Rate [kg/h]	20.34	22.46	22.49	22.38	22.44
Indicated Power [W]	264.52	274.51	274.38	273.84	274.35
Suction Losses [W]	20.9	18.66	18.47	18.71	18.78

The results for suction valve dynamics are presented in Figures 4 and 5, concerning the resulting force on the valve surface and the dimensionless valve displacements, respectively. Observation of the results shows that different wall function approaches and turbulence models predicts same force computed on the valve surface, which leads to reasonable agreement with experimental measurements of the suction valve dynamics. The only remark for valve dynamics is LES models presents maximum displacements subtly higher than experimental data and RANS predictions, but for the remainder steps of the suction valve dynamics was not detected any significant difference among the models predictions.

The comparison analysis of robustness and stability is performed observing RMS values of mass conversation residual for one compression cycle, presented in Figure 6, where it can be observed similar progression for all models with LES methods presenting values slightly higher.

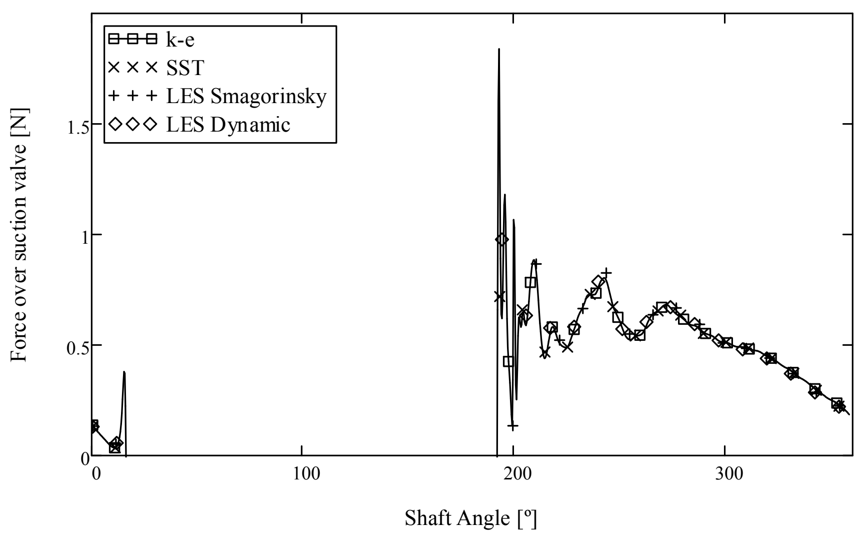

Figure 4: Prediction of resulting force over suction valve.

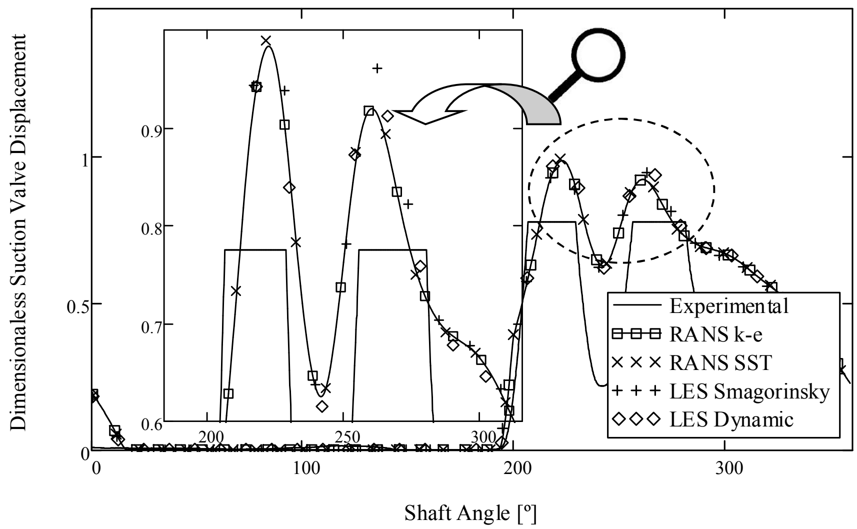

Figure 5: Comparison of suction valve dynamics for numerical predications and experimental measurements.

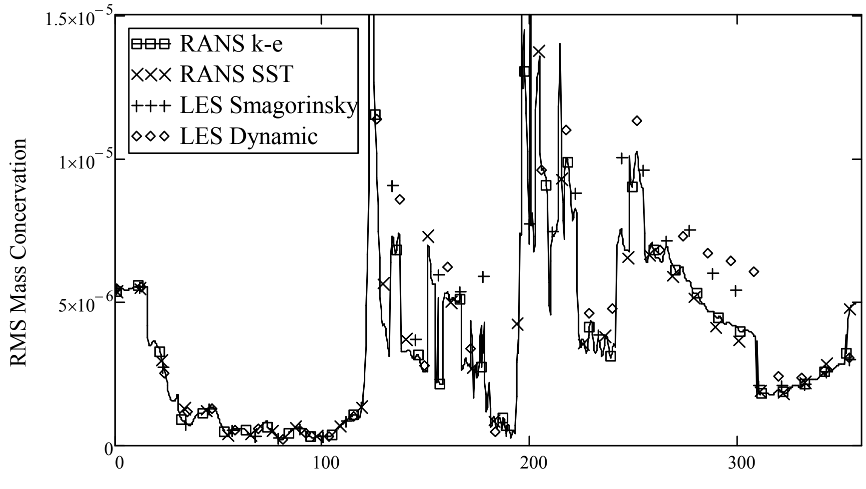

Figure 6: Evolution of RMS mass conservation for different turbulence modelling approaches.

Examination of the models ability to predict the compressor performance and provide robustness for convergence rate drives to conclude the models are even in the overall performance, which demonstrated similar relation between cost and benefit, once no significant difference among the CPU times was observed. It is important to remember this statement is based on the constant mesh refinement and numerical setup applied to all simulations performed in this investigation.

Additionally, due the great similarity among the results it is possible to affirm the unsteady simulation of the presenting compressor model is less dependent of the turbulence model employed than the numerical studies applied for the steady flow in radial diffusers, as discussed in the introduction section, where fully turbulent flow is developed.

Moreover, the results presented do not considerer specific treatment recommend for each turbulence model, mainly for LES methods. Still, additional simulations in that sense, not presented, were performed in order to evaluate the application of Central Difference Scheme (CDS) for the advection terms in LES simulations, which is a standard criteria stated in academic research. It was found problems for convergence stability, where Smagorinsky method developed slow convergence rate obtaining one order higher RMS values for equations residuals and increased CPU time, while the Dynamic model failed to obtain convergence in the second compression cycle.

In this sense once LES methods approaches Direct Numerical Simulation (DNS), if not set properly mesh and time step, additional numerical diffusion from the discretization process is needed to stabilize the solution.

5. CONCLUSIONS

A numerical study of the turbulence modeling effect on the FSI simulation of a small reciprocating compressor was presented, focused on the suction system due the high frequency pulsations, and experimentally validated. The goal was identify the best option for obtain reliable results, convergence robustness and fast CPU time, disregarding the specific mesh refinement and numerical setup demanded for each model, as described in the literature and academic research.

The investigated turbulence models showed small effect on the numerical results when compared to the experimental data. Therefore, the investigated compression process can be considered slightly dependent on the turbulence modeling, considering accuracy, even for coarse mesh size applied to all models. For instance, if computed the standard deviation for the main performance parameters, as the mass flow, the result obtained is below 0.7% of the overall average value.

In the behalf of stable convergence, all models presented similar performance if employed High Order Scheme, but when used CDS approach for LES methods, recommended in the specialized literature, it was observed slow convergence for the Smagorinsky sub-grid model and failed convergence for the Dynamic sub-grid model.

One can conclude, that despite LES be recommended for high fidelity simulations and in theory demands methodic setup criteria for obtain stable solution, it is possible to apply it for practical simulations of daily engineering on compressor simulation if diffusive discretization schemes are applied.

This is a small contribution effort obtained for one compressor model, application condition and refrigerant, and the main outline is the current mainstream turbulence models, also found in commercial CFD packages, provide similar and good results for FSI simulations intended to obtain the main compressor performance parameters.

It is important highlight the necessity of similar evaluations as presented in this report for a wide range of applications and appending model refinement investigations, which shall be addressed in further investigations.

NOMENCLATURE

CDS	Central Difference Scheme	(–)
DNS	Direct Numerical Simulation	(–)
FSI	Fluid-Structure Interaction	(–)
ILU	Incomplete LU factorization	(–)
LES	Large Eddy Simulation	(–)
RANS	Reynolds Averaged Navier Stokes	(–)
SST	Shear Stress Transport	(–)
k	turbulent kinetic energy	($m^2.s^{-2}$)
ε	turbulence eddy dissipation	($m^2.s^{-3}$)
ω	turbulence eddy frequency	(s^{-1})

REFERENCES

Ansys Inc., 2010, *ANSYS CFX-Solver Theory Guide*, v.13.0, 390 p.

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. and Prata, A.T., 1996, Turbulent Flow Through Reed Type Valves of Reciprocating Compressors, ASME, *International Mechanical Engineering Congress*, Atlanta, EUA.

Deschamps, C.J., Ferreira, R.T.S. and Prata, A.T., 1988, Application of the k-ε Model to Turbulente Flow in Compressor Valves, *Proc. 2nd Brazilian Thermal Science Meeting*, São Paulo: p. 259-262, (in Portuguese).

Deschamps, C.J., Prata, A.T. and Ferreira, R.T.S., 1989, Turbulent Flow Modeling in Presence of Stagnation Recirculation, Acceleration and Adverse Pressure Gradient, *Proc. X Brazilian Congress of Mechanical Engineering*, v.I: p. 57-60, (in Portuguese).

Germano,M., Piomelli, U., Moin, P ., Cabot, W.H., 1991, A Dynamic Subgrid-Scale Eddy Viscosity Model, *Phys. Fluids*, A 3 (7): p. 1760-1765.

J. Rigola, O. Lehmkuhl, C.D. Pérez-Segarra, A. Oliva, 2006, Numerical simulation of fluid flow through valves reeds based on Large Eddy Simulation models (LES), *Proc. International Compressor Engineering Conference at Purdue*, Purdue University, West Lafayette: 1405.

J. Rigola, O. Lehmkuhl, A. Oliva, C.D. Pérez-Segarra, 2009, Numerical simulation of the fluid flow through valves based on Large Eddy Simulation models, *Proc*. *International Conference on Compressors and their Systems*, London: p. 137-145.

J. Rigola, O. Lehmkuhl, J. Ventosa, C.D. Pérez-Segarra, A. Oliva, 2012, Numerical simulation of the turbulent fluid flow through valves based on Low Mach models, *Proc. International Compressor Engineering Conference at Purdue* , Purdue University, West Lafayette: 1375.

Kim, J., Wang, S., Park, S., Ryu, K., La, J., 2006, Valve Dynamic Analysis of a Hermetic Reciprocating Compressor, *Proc. International Compressor Engineering Conference at Purdue*, Purdue University, West Lafayette: C107.

Kim, H., Ahn, J., Kim, D., 2008, Fluid Structure Interaction and Impact Analyses of Reciprocating Compressor Discharge Valves, *Proc. International Compressor Engineering Conference at Purdue*, Purdue University, West Lafayette: 1112.

Lang, W., Heimel, M., Almbauer, R.A., Berger, E, 2011, Industrial reed valve development using virtual prototyping approach, *Proc. International Conference on Compressors and their Systems*, London: p.149-157.

Launder, B. E., Spalding, D. B., 1974, The numerical computation of turbulent flows, *Comp Meth Appl Mech Eng*, vol. 3: pp. 269-289.

Menter, F. R., 1994, Two-equation eddy-viscosity models for engineering applications, *AIAA-Journal*, vol. 32, pp. 1598-1605.

Mistry, H., Bhakta, A., Dhar, S., Bahadur, V., Dey, S., 2012, Capturing Valve Dynamics in Reciprocating Compressors through Computational Fluid Dynamics, *Proc. International Compressor Engineering Conference at Purdue*, Purdue University, West Lafayette: 1210.

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, *Proc. International Compressor Engineering Conference at Purdue* , Purdue University, West Lafayette: 1303.

Smagorinsky, J., 1963, General Circulation Experiments with the Primitive Equations, *Month. Weath.Rev*. Vol. 93: p. 99-165.

Takemori, C.K, 2008, Numerical simulation of the fluid-structure interaction applied to the reed valves of a miniaturized compressor under high frequency conditions, *Proc. International Conference on Compressors and their Systems,* London: p. 155-162.

Versteeg, H. K., Malalasekera, W., 2007, *An Introduction to Computational Fluid Dynamics – The Finite Volume Method*, Pearson Education Limited, Harlow, 503 p.

Wilcox, D.C., 1994, *Turbulence Modeling for CFD*, La Canada, California, 314 p.

ACKNOWLEDGEMENT

The authors acknowledge the EMBRACO technical and financial support.