Numerical Simulation of Fluid Flow Through Valve Reeds Based on Large Eddy Simulation Models (LES)

Joaquim Rigola
Technical University of Catalonia

Oriol Lekmuhl
Technical University of Catalonia

Carlos D. Perez-Segarra
Technical University of Catalonia

Assensi Oliva
Technical University of Catalonia

Numerical Simulation of Fluid Flow through Valves Reeds Based on Large Eddy Simulation Models (LES)

Joaquim RIGOLA[1], Oriol LEHMKUHL[1,2], Carlos D. PÉREZ-SEGARRA[1], Assensi OLIVA[1]

[1]Centre Tecnològic de Transferència de Calor (CTTC)
Universitat Politècnica de Catalunya (UPC)
ETSEIAT, C. Colom 11, 08222 Terrassa (Barcelona), Spain
Tel. +34-93-739.81.92, Fax: +34-93-739.89.20
E-mail: cttc@cttc.upc.edu http:// www.cttc.upc.edu

[2]Termo Fluids, S.L.
Magí Colet, 8, 08204 Sabadell (Barcelona), Spain
E-mail: termofluids@yahoo.es

ABSTRACT

One of the most critical aspects on hermetic reciprocating compressors design is to know in deep and clearly understand how the fluid flows through their valve reeds. In that sense, the use of CFD&HT codes, together with experimental validation tests are the best way to advance in this purpose. The use of TermoFluids code developed in a spin-off created by different researchers of CTTC Group has the advantage of knowing the whole CFD&HT code, working together with CFD developers, and using a new parallel, unstructured and object oriented CFD&HT code for turbulent industrial problems on low cost PC cluster. The aim of this paper is to study the fluid flow behavior through the valve at different Reynolds numbers and valve displacements, showing the numerical results obtained using only a symmetry-preserving discretization vs. considering large eddy simulation models (LES) for 2D and 2D periodical conditions. The studied cases vary from laminar to turbulent flow, almost to be considered incompressible and lower supersonic conditions and/or chocked flow. The numerical results allow knowing a numerical analysis of the effective force and flow area for the studied cases.

1. INTRODUCTION

The majority of hermetic reciprocating compressors use valves through the suction and discharge ports. The fluid flows from the suction chambers to compression chamber or from compression chamber to cylinder head when valves open or close depending on the pressure difference between them. The reed type valves are the basic configuration for domestic/commercial reciprocating compressors.

It is essential to know or understand the behaviour of the fluid flow across the valves plate through they main parameters in which these phenomena is defined - effective force area, A_F, and effective flow area $(KA)_e$ - in order to feed this information in the hermetic reciprocating compressors numerical models (Pérez-Segarra, et al. 2003) and improve these virtual laboratory prototype tools.

In that sense, these parameters are defined by (Schwerzler and Hamilton, 1972)(Soedel, 1992). The effective flow area, $(KA)_e$ is defined as the ratio between the actual mass flow rate and an ideal mass flow rate per unit flow area (assuming isentropic contraction process):

$$\dot{m} = (KA)_e (\rho v)_{id} = (KA)_e Y \sqrt{2\rho_u (p_u - p_d)} \tag{1}$$

where p_u and p_d are the upstream and downstream pressures of the fluid across the valve, Y is the compressibility factor, and ρ_u is the fluid upstream density. While the effective force area, A_F is defined as the ratio of the actual net force on the valve and the force obtained assuming a constant pressure distribution (and equal to p_u):

$$F = (p_u - p_d) A_F \tag{2}$$

One of the first semi-analytical models for predicting these coefficients were developed by Schwerzler and Hamilton, 1972 or Böswirth, 1982. More accurate predictions need obviously the use of the multidimensional simulation of the flow through the valve. Ferreira et al, 1989 analysed in detail the laminar flow through this kind of valves comparing numerical and experimental results. Later, Deschamps et al. 1996 study the turbulent flow in reed type valves using the well known RNG k-ε model. In the same way, and based on the same low Reynolds number two-equation k-ε turbulence models for general purposes (Pérez-Segarra et al. 1995) a numerical study of the turbulent fluid-flow through valves (Pérez-Segarra, et al. 1999) was also carried out.

The aim of the present paper is to carry out a group of numerical experiments over the fluid flow through the valve reed, using the CFD&HT code TermoFluids, a new unstructured and parallel object-oriented CFD code for accurate and reliable solving of industrial flows (Lehmkuhl, et al. 2007). In all studied case a multi-dimensional explicit finite volume fractional-step based algorithm has been used with symmetry preserving discretization scheme. When turbulence modelling is needed, an extension of the Dynamic Smagorinsky (Rodi 2006) model to non-structured meshes is used. The pressure equation is solved by means of parallel Fourier Schur decomposition solver which is an efficient direct solver for loosely coupled PC clusters (Borrell, et al. 2007).

In a two dimensional or two-dimensional periodic way the fluid flow is approached by two parallel phenomena (an entrance flow through a channel and a free jet through a surface). In that sense, the present paper is focused on the numerical simulation model of the fluid flow through the valve reeds, considering a simplified geometry of an axial hole plus a radial diffuser. The numerical results presented are based on a specific geometry – valve diameter d is 3 times orifice diameter D -together with a wide range of boundary conditions: entrance Reynolds number from 600 to 6000 and valve displacement s vs. valve diameter d ratio from 0.1 to 0.9. The studied cases vary from laminar to turbulent flow, almost to be considered incompressible and lower supersonic conditions and/or chocked flow.

2. TermoFluids: A GENERAL PURPOSE CFD&HT CODE

The increase in the computational power and the improvement in the numerical methods have been significant over the last decades. This fact, together with the emergence of low-cost parallel computers, has made possible the application of numerical methods to the study of complex phenomena and geometries, such as the simulation of turbulent industrial flows.

Parallel computing of turbulent flows using DNS (Soria, et al., 2004), LES (Saad et al., 1986) or hybrid LES/RANS (Trias, et al., 2006) models are currently being used. However, most of these techniques are commonly applied on structured Cartesian or body-fitted multi-block codes. Taking into account the current state-of-the-art of parallel techniques, and the ability of unstructured meshes to create grids around complex geometries, an unstructured and parallel object-oriented code called TermoFluids has been used to numerically simulate the fluid flow through the valve reeds on hermetic reciprocating design. This code uses efficient algorithms that work well on slow networks of PC clusters. The code implements turbulence models such as LES, hybrid LES/RANS and RANS models. The use of such techniques have made possible the accurate simulation of industrial flows with few simplifications in the geometry and the fluid dynamic and heat and mass transfer phenomena.

The use of state-of-the-art in object oriented based software engineering techniques and the implementation in C++ has allowed a quick development of the code. In this paper, the numerical methods, the linear solvers, algorithm used and the turbulence modelling selected are briefly explained, together with the numerical results presented on fluid flow valves.

In this section the main features of a general purpose CFD code called TermoFluids are presented. The code has been designed for the simulation of complex fluid dynamics and heat and mass transfer problems of industrial interest. Its most relevant technical specifications are:

- Finite volume method on 3D or 2D unstructured collocated meshes (a mixture of tetrahedral, hexahedral, prism and/or pyramid elements).
- Steady state or time integration flows with fully implicit (SIMPLE-like) and explicit (fractional step method) algorithm.
- Direct and iterative sparse linear solvers.

- Laminar and turbulent flow modelling.
- 2nd order space and time discretization.
- 2nd order symmetry preserving discretization.
- Multi-component fluids.
- Conjugate heat transfer with a loosely coupled algorithm (possibility of different time steps for the fluid and the solid).
- Radiative heat transfer using Discrete Ordinates Method for transparent and non-transparent medium.
- Local refinement and Laplacian smoother post-processing tools.

2.1 Governing equations and symmetry preserving discretization

We consider the simulation of turbulent, incompressible flows of Newtonioan fluids. Under these assumptions, the governing equations in primitive variables are:

$$\frac{\partial v_x}{\partial x}+\frac{\partial v_y}{\partial y}+\frac{\partial v_z}{\partial z}=0 \tag{3}$$

$$\rho\left[\frac{\partial v_x}{\partial t}+v_x\frac{\partial v_x}{\partial x}+v_y\frac{\partial v_x}{\partial y}+v_z\frac{\partial v_x}{\partial z}\right]=-\frac{\partial p}{\partial x}+\frac{\partial \tau_{xx}}{\partial x}+\frac{\partial \tau_{yx}}{\partial y}+\frac{\partial \tau_{zx}}{\partial z}+\rho g_x \tag{4}$$

$$\rho\left[\frac{\partial v_y}{\partial t}+v_x\frac{\partial v_y}{\partial x}+v_y\frac{\partial v_y}{\partial y}+v_z\frac{\partial v_y}{\partial z}\right]=-\frac{\partial p}{\partial y}+\frac{\partial \tau_{xy}}{\partial x}+\frac{\partial \tau_{yy}}{\partial y}+\frac{\partial \tau_{zy}}{\partial z}+\rho g_y \tag{5}$$

$$\rho\left[\frac{\partial v_z}{\partial t}+v_x\frac{\partial v_z}{\partial x}+v_y\frac{\partial v_z}{\partial y}+v_z\frac{\partial v_z}{\partial z}\right]=-\frac{\partial p}{\partial z}+\frac{\partial \tau_{xz}}{\partial x}+\frac{\partial \tau_{yz}}{\partial y}+\frac{\partial \tau_{zz}}{\partial z}+\rho g_z \tag{5}$$

where: $\tau_{xx}=2\mu\frac{\partial v_x}{\partial x}-\frac{2}{3}\mu\nabla\cdot\vec{v}$; $\tau_{yy}=2\mu\frac{\partial v_y}{\partial y}-\frac{2}{3}\mu\nabla\cdot\vec{v}$; $\tau_{zz}=2\mu\frac{\partial v_z}{\partial z}-\frac{2}{3}\mu\nabla\cdot\vec{v}$

$\tau_{xy}=\tau_{yx}=\mu\left(\frac{\partial v_x}{\partial y}+\frac{\partial v_y}{\partial x}\right)$; $\tau_{xz}=\tau_{zx}=\mu\left(\frac{\partial v_x}{\partial z}+\frac{\partial v_z}{\partial x}\right)$ $\tau_{yz}=\tau_{zy}=\mu\left(\frac{\partial v_y}{\partial z}+\frac{\partial v_z}{\partial y}\right)$

Basic physical properties of Navier-Stokes equations can be deduced from symmetries of the differential operators of Navier Stokes equations. Note, that it is strictly necessary to preserve such operator symmetries, at discrete level, to preserve the analogous (invariant) properties of the continuous equations. It may be argue, especially if the method is to be used on unstructured meshes that accuracy may need to take precedence over properties of the operator. However in the discretitzation schemes used in TermoFluids the philosophy followed by Verstappen (Verstappen et al, 2003) is adopted and extended to non-structured collocated meshes: symmetries of the convective and diffusive operators are critical to the dynamics of turbulence and must be preserved.

2.1 Sparce linear solvers

Once the system of equations is discretized, the resulting system of linear equations (all the dependent variables except the pressure which receive a special treatment), are solved by means of a GMRES (M = 5) (Saad et al., 1986) solver preconditioned with the inverse of the diagonal of the system. This method does not lose performance with the increase of the number of CPU and has low memory requirements.

For incompressible flows, the above solution does not work well with the pressure equation. For this situation, the pressure equation is reduced to the Poisson equation, which has an infinite speed propagation of the information in the spatial domain, i.e. the problem becomes fully elliptic. For the treatment of this particularity, especial solvers have been developed such as: Direct solvers (direct Schur Decomposition (Trias, et al., 2006) using sparse Cholesky for the local variables with iterative or direct solver for the interface system) and Iterative solvers (CG with a PFSAI preconditioner) (Borrell, et al., 2007).

2.2 Parallel algorithms

The parallelisation of the non-linear system of equations in the spatial domain decomposition is one of the main features of TermoFluids code development. It has been specially focused on the implementation of new techniques to balance the load for each CPU used. To solve the non uniform load of the processes that affects the overall efficiency of the code is an important problem in low-cost PC clusters. Partitioning of the computational domain is carried out by means of METIS (Karypis, et al. 1998) software. The algorithm implements the constructive proof of Vizing's theorem (Misra, et al., 1992) in order to minimise the synchronic point-to-point communication between CPUs.For the studied cases the CFD&HT code has run in a one CPU. No parallel algorithms have been needed due to the low CPU time needed and thanks to the fast time of TermoFluids code.

2.3 Turbulence modelling

Traditionally, in most industrial unstructured CFD codes turbulence is treated by means of RANS models. TermoFluids code employs a wide variety of RANS models usually used for industrial applications such as: κ-ω models: Wilcox; Peng, Davidson and Holmberg; Bredberg, Davidson and Peng; κ-ε models: Ince and Lauder with standard Yap correction; Abe, Kondoh and Nagano; Goldberg, Peroomian and Chakravarthy; RNG-Based κ-ε; hybrid κ-ω / κ-ε models: Menter Standard SST Model. To deal with the coupling between the turbulent variables, an efficient locally coupled algorithm has been implemented, combined with BiCGSTAB to solve the spatial couples.

Wall boundary conditions are treated by means of different approaches. The first one integrates the boundary layer but requires a high mesh density at this zone (low-Reynolds turbulent number version of the RANS model). The second one uses the two-layer approach. That means, wall layer is calculated with a one-equation model (i.e. a differential equation for turbulent kinetic energy and algebraic expressions for viscosity and turbulent dissipation) while the external zone is treated with the selected RANS model. In this case, mesh requirement is lower than the first approach. The third implements the standard wall functions, being the least restrictive of the three. The code implementation allows also the use of the different approaches in the same computational domain, depending on the boundary conditions.

Numerical simulations using traditional RANS models appear to be unable to accurately account for the time-depending and three-dimensional motions governing flows with massive separation or to predict flow transitions. This unsteady three-dimensional behaviour of the flow can be better solved by means of LES modelling (Piomelli, 1999). However, due to the need of solving the small scales motion in the boundary layer, they require a high amount of computational resources.

This drawback can be circumvented if hybrid LES/RANS models are used. The main idea of such modelling is to solve the whole wall layer with a RANS model, while the external zone is treated with LES. This kind of hybrid models takes advantage of the good performance of RANS models in the region near the walls and the capability of LES to resolve the time-depending and three-dimensional behaviour that dominate the separated regions. Among the main LES and hybrid LES/RANS models implemented in the code TermoFluids can be cited: Smagorinsky SGS model; Dynamic Smagorinsky SGS model; one-Equation SGS Model Yoshizawa with possibility of hybrid LES/RANS using all linear RANS models; Spalart Detached Eddy Simulation Model; Menter Detached Eddy Simulation Model.

Another important factor to achieve numerical stability in RANS/LES methods is the transition functions that switch the RANS model (in the boundary layer) and the LES model (in the detached zone). Two different functions can be used in the TermoFluids code. The first one is the traditional transfer function used by Spalart in DES model (Spalart et al., 2006). This function assumes that the model used to solve the smaller turbulent length l in a control volume is the one that dominate. That is, for a generic two-equation turbulence model:

$$\frac{\partial k}{\partial t}+\frac{\partial}{\partial x_i}(\rho u_i k)=\frac{\partial}{\partial x_i}(\mu_{eff}\frac{\partial k}{\partial x_i})+P_k-\rho\beta_k\frac{k^{3/2}}{l} \quad (6)$$

$$\frac{\partial \omega}{\partial t}+\frac{\partial}{\partial x_i}(\rho u_i \omega)=\frac{\partial}{\partial x_i}(\mu_{eff}\frac{\partial \omega}{\partial x_i})+P_\omega-\rho\beta_k\omega^2+C_D \quad (7)$$

where, $l=\min(\sqrt{\kappa/\omega},C_{LES}\Delta)$ depending on both turbulent length of RANS and LES models, respectively.

C is a constant usually considered as 0.65 (Spalart et al., 2006) and $\Delta = \left(\int_{CV} \delta V\right)^{1/3}$

One of the main drawbacks of this formulation is that the RANS and LES zone can be mesh dependent. If the boundary layer is meshed with a regular mesh (i.e. with local refinement methods), the LES model can be switched on inside the boundary layer. On the other hand, (Menter) in his SST model has used a transfer function that can be adapted to this situation. This function is meshed independent (works equal in irregular or regular meshes) but uses normal distance to the wall, so in dynamic mesh methods a huge computational effort is required.

3. COMPUTATIONAL DOMAIN, MESH AND BOUNDARY CONDITIONS

The geometry of compressor valve reed has been idealised as an axial feeding plus a perpendicular diffuser followed by a discharge zone. The following geometrical values have been considered: valve plate thickness *e* is equal to orifice diameter *d*; while valve plate diameter *D* is three times orifice diameter *d*. The ratio between gap *s* and orifice diameter *d* changes from 0.1 to 0.9 over the studied cases. The ratio between the orifice diameter *d* and horizontal domain length l_h is 0.06, while the ratio between the orifice diameter *d* and horizontal domain length l_v is 0.03. Thus, rectangular ratio l_h/l_v is 0.5. Figure 1 shows the whole computational domain and a half zoom of the valve seat, valve reed and fluid gap, respectively.

Both pictures depict that the mesh domain is composed by 4 horizontal zones from bottom to top: i) inlet orifice zone of 15 nodes; ii) down valve zone of 15 nodes; iii) parallel zone around the valve of 6 nodes and iv) up valve zone of 30 nodes.

The same mesh domain is divided by 5 vertical zones from left to right: i) left valve part of 30 nodes; ii) left orifice part over the valve of 15 nodes and under the valve of 35 nodes; iii) middle part of 15 nodes in the orifice, 15 nodes between the orifice and the valve and 30 nodes over the valve; iv) right orifice part over the valve of 15 nodes and under the valve of 35 nodes; and v) right valve part of 30 nodes. In that sense, symmetry mesh is accomplished and exponential gradient is considered when the zone is near the valve.

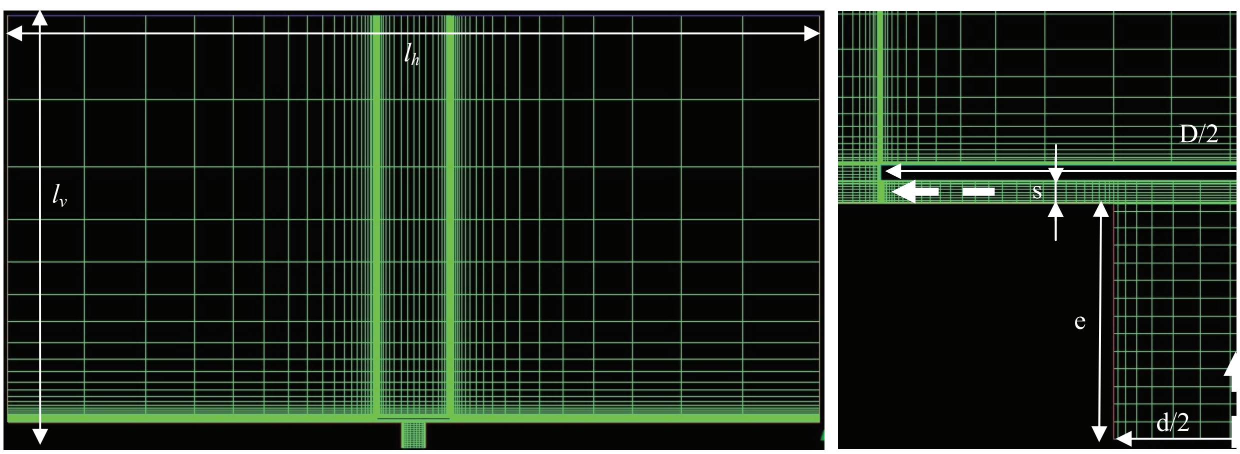

Figure 1. Mesh and computational domain. General view (left); zoom view (right).

The boundary conditions for the simulated cases are:

- Bottom inlet orifice: input constant velocity for Reynolds number from 600 to 6000.
- Bottom part and valve reed: solid wall
- Lateral domain: Neumann conditions.
- Top outlet fluid exit: $p_o - p_b = \gamma_b(\rho_b v_b^2)/2$

The turbulence model used is Dynamic Smagorinsky SGS Model with a mesh of 7000 control volumes. Twelve hours of computation are necessary in order to achieve the statistically stationary motion using 1 CPU.

4. NUMERICAL RESULTS

Based on the numerical simulation code explained above, three different aspects have been numerically evaluated under the specific geometry and the boundary conditions detailed in section 3. Firstly, the pressure distribution above the valve reed is analyzed for 3 different s/d ratios under 3 Reynolds numbers ranging from 600 to 6000. Secondly, a detailed analysis of pressure distribution under reed valve is presented showing the influence to study a two-dimensional case vs. a two-dimensional periodic case. Thus, influence on 3D is taken into account. This second case is also studied considering different turbulence modeling for Reynolds numbers of 3000 and 6000. Finally, global effective force and flow areas are compared for different s/d ratios and Reynolds ranges.

4.1 Reynolds evolution at different s/d ratios

Figure 2 shows the fluid flow pressure distribution under the solid valve reed, considering three s/d ratios of 0.1, 0.4 and 0.9 respectively, through three different Reynolds numbers of 600, 3000 and 6000.

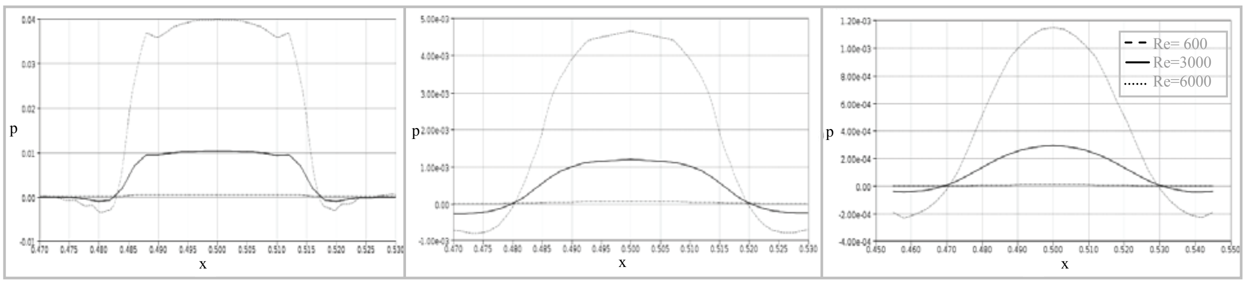

Figure 2. Comparative numerical analysis of Re influence for s/d ratios of 0.1, 0.4 and 0.9, respectively.

The numerical results show that when s/d ratio is 0.1 or 0.4 the pressure distribution indicates a shape with a minimum far from the inlet section under the valve, and a maximum in the center of the inlet section under the valve. The higher pressure values are almost constant for s/d ratio around 0.1 and completely parabolic at s/d ratios of 0.9. The shape increases these effects when Reynolds number increases. The results presented show an agreement with the general pressure distribution referenced in (Ferreira et al., 1986) based on experimental available studies.

4.2 Turbulence modeling influence for a specific geometrical case changing Re number

Figure 3 shows the fluid flow pressure distribution under the solid valve reed, considering the three different Reynolds numbers of 600, 3000 and 6000 for s/d ratio of 0.1, evaluating a 2D case and a 3D case considering periodical conditions between front and back.

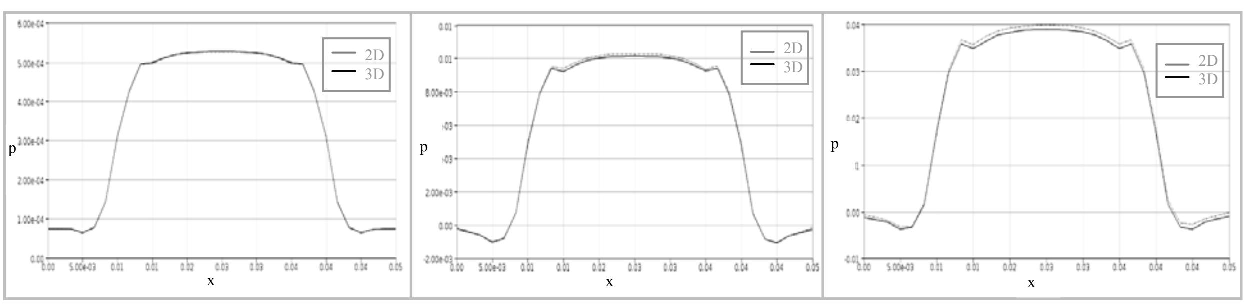

Figure 3. Comparative numerical analysis 2D vs. 2D periodic at Reynolds of 600, 3000 and 6000, respectively.

The comparative numerical results of Figure 3 show that 3D has no influence on Reynolds number of 600. Thus, this case is almost laminar. The other cases shows a differece between the comparative solution, increasing these differences when Reynolds number increases. Although the results show differences on pressure values, the shape of the pressure evolution is very similar. Then, 3D analysis will improve the accuracy of the pressure values maintaining the same pressure evolution.

Figure 4 depicts both 3D periodical cases for turbulence Reynolds numbers of 3000 and 6000 based on the symmetry preserving model like all the cases presented since now, against the dynamic Smagorinsky turbulence modeling.

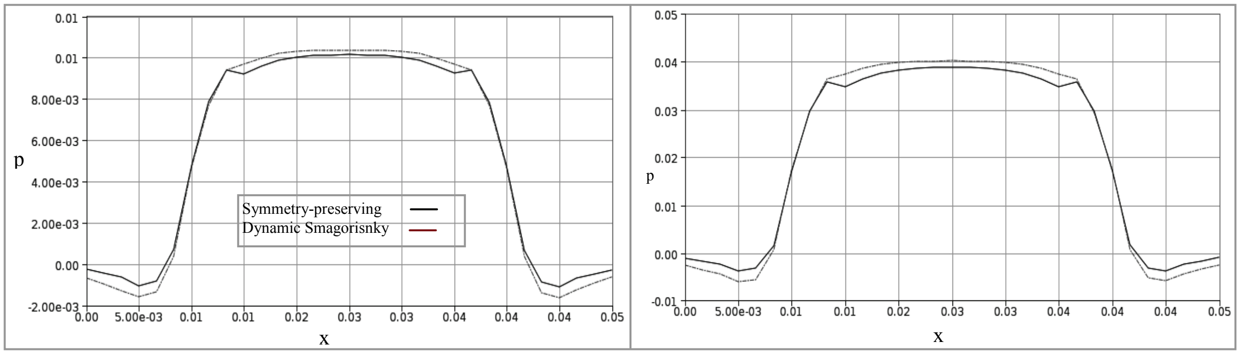

Figure 4. Comparative numerical analysis 3D periodic comparing symmetry-preserving vs. dynamic Smagorinsky turbulence modelling, for Re numbers of 3000 and 6000, respectively.

The numerical results of Figure 4 indicate the importance on turbulence modeling implemented. In both presented cases, the pressure distribution under the valve reed and over the inlet orifice presents a different shape, together with an important decrease on pressure value in the rest of the pressure distribution.

4.3 Global effective force and flow area under the studied cases

Based on the numerical results presented in the above sections, a global values of effective force and flow areas (1)(2) is shown in an non-dimensional way. Figure 5 shows both non-dimensional effective force and flow areas considering the Reynolds numbers of 600, 3000 and 6000, together with the s/d ratios of 0.1, 0.3 and 0.9.

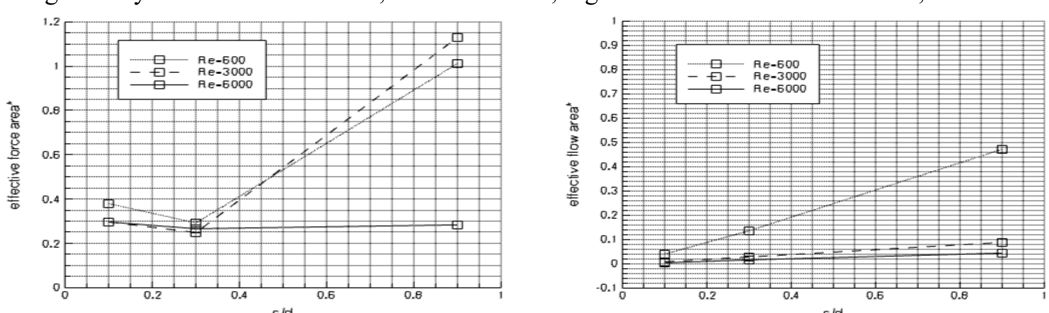

Figure 5. Numerical results globaly evaluated as effective force and flow areas.

Although the numerical results obtained are not points enough to show the global values tendency, the points obtained presents a reasonable agreement with (Ferreira et al., 1986) results. Effective force area decreases until s/d ratio between 0.2 and 0.4 and after that increases. Effective flow area is almost linear in all cases, with different slope increasing when Reynolds number decreases.

5. CONCLUSIONS

A numerical study of the turbulence fluid flow through valve reed has been presented based on a CFD&HT models considering 2D and 2D periodical conditions in order to show the possibilities of these Large Eddy Simulation Models, excluding the use of only RANS models due to their limitations. In that sense a turbulent, incompressible flow of Newtonian fluids are evaluated considering cartesian coordinates on 2D or 2D periodical mesh. Axial symmetric coordinates has not been taken into account due to LES models must be based on 3D, to be developed in axial-symmetric system. The numerical results presented have shown a general reasonable good agreement with the available data. Instead of that, more efforts on 3D resolution, axial symmetric coordinates and compressible effects will be carried out in order to improve the numerical results for this fluid flow problem associated with the compressor valve reeds.

REFERENCES

Pérez-Segarra, C.D., Rigola, J., Oliva, A., 2003, Modelling and Numerical Simulation of the Thermal and Fluid Dynamic Behavior of Hermetic Reciprocating Compressors-Part I: Theoretical Basis, *Int. J. of Heating, Ventilating and Air.-Conditioning Refrigerating Research*, vol. 9, no. 2, 2003.

Schwerzler, D.D., Hamilton, J.F., 1972, An Analytical Method for determining Effective Flow and Force Areas for Refrigeration Compressor Valving Systems, *International Compressor Engineering Conference*, pp. 30-36, Purdue University, Indiana, 1972.

Soedel, W., Mechanics, Simulation and design of Compressor Valves, Gas Passages and Pulsation Mufflers, Purdue University,1992.

Böswirth, L., 1982 Theoretical and experimental study on flow in valve channels, Parts I and II, *International Compressor Engineering Conference*, pp. 38-53.

Deschamps, C.J., Ferreira, R.T.S., Prata, A.T., 1996, Turbulent Flow Through Valves of Reciprocating Compressors, International Compressor Engineering Conference, pp. 377-382, Purdue University, Indiana.

Pérez-Segarra, C.D., Oliva, A. Costa, M., Escanes, F., 1995, Numerical Experiments in Turbulent Natural and Mixed Convection in Internal Flows, International Journal of Numerical Methods for Heat & Fluid Flow, Vol.5, No. 1, pp.13-32, 1995

Pérez-Segarra, C.D., Cadafalch, J., Rigola, J., Oliva, A. 1999, Numerical study of turbulent fluid flow through valves, Int. Conference on Compressors and Their Systems, C542/021/99.

Lehmkuhl, O. Pérez-Segarra, C.D., Borrell, R., Soria M., Oliva A., 2007, TermoFluids: A new parallel unsructured CFD code for the simulation of turbulent industrial problems on low cost PC Cluster, *Parallel CFD Congress,* Antalya, Turkey.

Rodi, W.,2006, DNS and LES of some engineering flows, *Fluid Dynamic Research*, vol. 38, pp. 145-173.

Verstappen, R., 2004, A synthesis of similarity and eddy-viscosity models, *Direct and Large Eddy-Simulation v.* pp.89-96.

Borrell, R., Lehmkuhl, O., Soria M., Oliva; A. 2007, Direct Schur method for the solution of Poisson equation with unstructured meshes; Parallel CFD Congress, Antalya, Turkey.

Soria, M., Trias, F.X., Perez-Segarra C.D., Oliva; A., 2004, Direct numerical simulation of a three-dimensional natural convection flow in a differentially heated cavity of aspect ratio 4; *Numerical Heat Transfer*, Part A, 45 , pp. 649-673.

Saad, Y.Schultz; M.H., 1986, GMRES: A generalized minimal residual algorithm for solving nonsymmetric linear systems; *SIAM J.Sci.Stat.Comput.*

Trias, F.X., Soria, M., Pérez-Segarra C.D., Oliva; A, 2006, Direct Schur-Fourier Decomposition for the Efficient Solution of High-Order Poisson Equations on Loosely Coupled Parallel Computers; *Numerical Linear Algebra* with Applications 13, pp. 303-326.

Karypis, G., Kumar V., 1998, MeTIS: A software package for partitioning unstructured graphs, partitioning meshes and computing fill-reducing ordering of sparse matrixes, version 4.

Misra, J., Gries, D., 1992, A constructive proof of Vizing's theorem; Inform. Process. Lett. 41, 131-133.

Piomelli, U., Large-eddy simulation: achievements and challenges; Progress in Aerospace Sciences 35 (1999); 335-362.

Mahesh, K., Constantinescu, G., Moin, P. 2004, A numerical method for large-eddy simulation in complex geometries, *Journal of Computational Physics* vol. 197, pp. 215-240.

Spalart, P.R., Deck, S., Shur, M.L., Squires, K.D., Strelets, M.Kh., Travin; A., 2006, A new version of detached-eddy simulation, resistant to ambiguous grid densities; *Theor. Comput. Fluid Dyn.* Vol. 20.

Menter, F.R., Improved Two-Equation k-w Turbulence Models for Aerodynamic Flows; NASA TM-103975.

Verstappen, R., Veldman A.E.P., 2003, Symmetry-preserving discretization of turbulent flow, *J. Computational* Physics. Vol 187,pp.343-368.

Ferreira, R.T.S. and Driessen, J.L., 1986, Analysis of the influence of valve geometric parameters on the effective flow and force areas, International Compressor Engineering Conference, pp. 632-646.

ACKNOWLEDGEMENT

The authors gratefully acknowledge the financial support provided by ACC Compressors Spain, S.A. – Unidad Hermética division (ref. no. C05652) and by the Ministerio de Educación y Ciencia, Secretaría de Estado de Universidades e Investigación, Spain, (ref. no. PTR1995-0886-OP).