# Purdue University Purdue e-Pubs

International Compressor Engineering Conference

School of Mechanical Engineering

2010

# Introduction of CFD&HT Analysis into an Object Oriented One Dimensional and Transient Program for Numerically Simulate Hermetic Refrigeration Compressors

Joan López CTTC - UPC

Joaquim Rigola CTTC - UPC

Oriol Lehmkuhl Termo Fluids

Assensi Oliva CTTC - UPC

Follow this and additional works at: http://docs.lib.purdue.edu/icec

López, Joan; Rigola, Joaquim; Lehmkuhl, Oriol; and Oliva, Assensi, "Introduction of CFD&HT Analysis into an Object Oriented One Dimensional and Transient Program for Numerically Simulate Hermetic Refrigeration Compressors" (2010). *International Compressor Engineering Conference*. Paper 2012.

http://docs.lib.purdue.edu/icec/2012

This document has been made available through Purdue e-Pubs, a service of the Purdue University Libraries. Please contact epubs@purdue.edu for additional information.

 $Complete \ proceedings \ may \ be \ acquired \ in \ print \ and \ on \ CD-ROM \ directly \ from \ the \ Ray \ W. \ Herrick \ Laboratories \ at \ https://engineering.purdue.edu/Herrick/Events/orderlit.html$ 

# Introduction of CFD&HT Analysis into an Object Oriented One Dimensional and Transient Program for Numerically Simulate Hermetic Refrigeration Compressors

Joan LÓPEZ<sup>1</sup>, Joaquim RIGOLA<sup>1</sup>, Oriol LEHMKUHL<sup>1/2</sup>, Assensi OLIVA<sup>1</sup>\*

<sup>1</sup>Centre Tecnològic de Transferència de Calor (CTTC), Universitat Politècnica de Catalunya (UPC), 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

<sup>2</sup>Termo Fluids, S.L. Magí Colet 8, 08204 Sabadell (Barcelona), Spain E-mail: termofluids@termofluids.com

### **ABSTRACT**

The present paper is a coupling between a modular object oriented one-dimensional and transient numerical simulation model for the thermal and fluid dynamic analysis of compressor behavior and CFD&HT code over the circulating flow between the compressor and the shell evaluated under Large Eddy Simulation turbulence models. Both computational strategies and their coupling are here presented.

The use of new refrigerants, new compressor circuitry, new designs, some aspects like several parallel paths, more than one compression chamber, etc., have obliged to develop new modeling strategies. Thermal and fluid dynamic behavior of hermetic reciprocating compressors is characterized by complex heat transfer and fluid flow phenomena: three-dimensional, turbulent, fast transient, etc. Thus, CFD&HT codes coupled with the compressors numerical simulation model, under specific critical points are necessary to help on compressor improvement and efficiency optimization. The development and knowledge of both computational methodologies allow better results interpretation and deeper phenomena comprehension.

As result of this research a new code which couples 1D based strategies and CFD&HT analysis has been used to carry out the full numerical simulation of an hermetic reciprocating compressor. The results of this simulation are presented in this paper showing first steps in this area.

### 1. INTRODUCTION

An object oriented approach for the numerical simulation of the thermal and fluid dynamic behavior of hermetic reciprocating compressors has been developed, verified and experimentally validated (Damle *et al.*, 2008). The compressor domain is formed by connecting individual elements such as tubes, chambers, compression chambers, valve plates, etc., which exchange information (pressure, temperature, mass flow etc.) between themselves. The numerical model analyses the fluid flow based on full integration of the one-dimensional transient governing equations (continuity, momentum and energy) through each one of the compressor elements (objects from a C++ point of view). The coupled system is solved until convergence is reached. This procedure allows coupling each type of geometry desired with the possibility to add Computational Fluid Dynamics and Heat Transfer (CDF&HT) code to solve elements of the compressor integrated with the general compressor model.

A new unstructured and parallel object-oriented CFD&HT code for accurate and reliable solving of turbulent industrial flows (2), called TermoFluids, has been developed in a spin-off created by different researchers of CTTC Group. A multi-dimensional explicit finite volume fractional-step based algorithm has been used with symmetry preserving discretization scheme. The turbulence modeling is an extension of the Yoshizawa nonequilibrium fixed-parameter subgrid-scale model to non-structured meshes. The pressure equation is solved by means of parallel

Fourier Schur decomposition solver which is an efficient direct solver for loosely coupled PC clusters (Trias et al., 2006).

The coupling of TermoFluids with the one dimensional approach has allowed the implementation of new solver objects which can be used to apply CFD&HT analysis over the compressor elements previously defined by Damle *et al.* (2008). In this paper, the fluid domain between the shell and the internal elements has been established as an own entity object and a CFD&HT solver has been inherited to be included on its core.

Hence, this coupling not only allows the thermal and fluid dynamic study of hermetic reciprocating compressors in an intuitive and flexible way, but also allows the implementation of CFD&HT analysis over its objects to obtain detailed numerical results of the compressor parts. First numerical results from the numerical simulation of the whole domain of a hermetic reciprocating compressor are presented.

## 2. NUMERICAL SIMULATION OF COMPRESSOR BEHAVIOUR

The compressor domain is divided into separate elements. Figure 1 shows the most simplified compressor domain and the possible objects here selected.

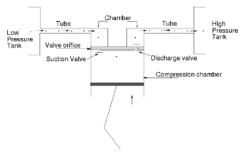


Figure 1: Simplest combination of elements (objects) forming the compressor

Each type of compressor element -object- is divided into strategically distributed control volumes (CVs). The general conservation equations of the fluid flow are semi-discretized as:

$$\frac{\partial m}{\partial t} + \sum \dot{m}_e - \sum \dot{m}_w = 0 \tag{1}$$

$$\frac{\partial m\overline{v}}{\partial t} + \sum \dot{m}_e v_e - \sum \dot{m}_w v_w = F_s \tag{2}$$

$$\frac{\partial m(\overline{h} + \overline{e}_c)}{\partial t} + \sum \dot{m}_e (h_e + e_{c,e}) - \sum \dot{m}_w (h_w + e_{c,w}) = \dot{Q}_{wall} + \overline{V} \frac{\partial \widetilde{p}}{\partial t}$$
(3)

Some elements like the gas in a chamber or a compressor chamber have one CV and are not able to be divided into smaller ones. Other elements like the gas through a tube are able to be divided into an arbitrary number of CVs. For each CV a mesh node is assigned at its center (see Fig. 2a). Pressure p, enthalpy h and density  $\rho$  are obtained from continuity equation (1), energy equation (3) and state equation  $\rho = (p, h)$  respectively, and are evaluated at each

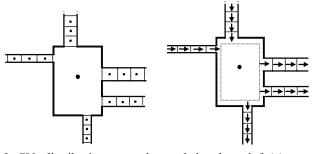


Figure 2: CVs distribution over tubes and chambers: left (a) center grid nodes; right (b) staggered mesh.

node. The compression chamber volume, which changes along time, is the one evaluated by means of the space conservation equation. Staggered arrangement is used to determine velocity field at the faces of the main control volumes through momentum equation (2) (see Fig. 2b).

Tube elements are solved according to pressure-based methods SIMPLEC algorithm (Patankar, 1980) using staggered mesh for velocity map. Upwind criteria are used for convective terms. In case of sudden expansion or contraction between section I and II, together with its vena contracta c, when tubes are connected to other tubes or chambers, the following relation is used to relate inlet i and outlet o pressures.

$$l\frac{\partial m}{\partial t} + \frac{|m|v_P}{2} \left( \frac{(S^S)^2}{(S^I)^2} - \frac{(S^S)^2}{(S^I)^2} \right) + \left( \frac{S^S}{S^0} - \frac{1}{C_C} \right)^2 = (p_i - p_o)S_S$$
 (4)

The tube gives pressure and mass flow rate as output to the chamber/compression chamber which is used for the first pass through the chamber/compression chamber iterative loop at each iteration, until convergence is reached.

Pressure correction approach for the compression chamber is also used. Chamber treatment is similar to that of compression chamber with the volume at the current time step V, equal to that at the previous instant  $V^0$  as the volume of chamber remains constant.

The mass flow rate equation for the inlet/outlet of the compression chamber is written as:

$$\dot{\boldsymbol{m}} = \frac{N_k(p_p - p_k) + H_k + (\frac{l_k m_k^0}{\Delta t})}{M_k} \tag{5}$$

where, the variables take different values for valve orifices and tubes. The subscript k indicates the different elements connected to the compression chamber/chamber. From this equation a mass correction is sought as:  $\dot{m}_k' = d_k(p_P' - p_R')$  where  $d_k = N_k/M_k$  and put in the continuity equation (1) to obtain an equation for pressure correction p' for compression chamber/chamber. Here, flow entering the chamber is considered positive (*west*) and leaving as negative (*east*). The compression chamber/chamber gets pressure from the tubes and valve orifices as data. Thus,  $p'_{kF} = 0$  and  $p'_{kW} = 0$ .

The valve orifice object is connected between a chamber and a compression chamber, where a similar equation to the expansion/contraction is extended to compressible flow to evaluate the mass flow rate through the valves between any inlet i and outlet o pressures and it pressure ratio  $\Pi$ .

$$l\frac{\partial \dot{m}}{\partial t} + \frac{|\dot{m}|v_p}{2} \frac{\gamma - 1}{\gamma} + \frac{1 - p_0/p_i}{\Pi^{1/\gamma - \Pi}} = (p_i - p_o)(KS)_s$$
 (6)

The effective flow area  $(KS)_S$  is function of pressure ratios and it is then calculated turning the mass flow rate.

Fixed value objects serve as boundary conditions (fixed pressure and temperature). It only gives pressure and temperature values as output to the tubes connected to it.

These objects are linked together to form the compressor domain. Each object is capable of solving itself for given boundary conditions. At each iteration, inputs are taken from the neighbors, while momentum, pressure correction and energy equations of the object are solved. Their outputs are the set for neighbors' resolution. Iterations continue until convergence is reached at a given time step and then the next time step calculation starts after updating the variables.

This procedure avoids decoupling between continuity and momentum equations, allowing a better way for handling abrupt changes in the cross-section between tubes and permitting the coupling between tubes and chambers resolution, which are solved independently. It is important to highlight that the aim of this systems library program is to connect all the elements between them and solve it as a whole taking into account all connections, in order to obtain it in a modular way. This procedure allows coupling each type of geometry desired (more than one compressor chamber, more than one connection tube between chambers, more than one resonator connected with a chamber) and with the possibility to add CFD&HT codes to solve parts of the compressor integrated with the general compressor model here described.

## 3. CFD&HT CODE (TERMOFLUIDS)

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, LES or hybrid LES/RANS 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, a new unstructured and parallel object-oriented code called TermoFluids has been used to numerically simulate the fluid flow in the space between compressor shell and compressor crankcase with the objective to determine its influence on suction return design and compressor analysis. TermoFluids (Lehmkuhl *et al.*, 2007) uses efficient algorithms, which work adequately both on slow networks of personal computers clusters and supercomputers. Governing partial differential equations are converted into algebraic ones using three-dimensional unstructured collocated meshes with symmetry-preserving discretization (Verstappen and Vedman, 2003). The systems of equations are solved with full parallel direct and iterative sparse linear solvers, using transient time integration with fully explicit fractional step algorithms. Fully conservative second-order schemes are used for spatial discretization.

Local refinement of the grid is used. For the solution of the pressure equation, it is used a Direct Fourier Schur Decomposition (Trias *et al.*, 2006) using sparse Cholesky for the local variables with an iterative or direct solver for the interface system.

### 3.1 Discrete Navier-Stokes Equation

The Navier-Stokes and continuity equations can be written as:

$$\rho \frac{\partial u}{\partial t} + C(u)u + Du + Gp = 0$$
 (7)

$$Mu = 0 (8)$$

where  $u \in R^m$  and  $p \in R^q$  are the velocity vector and pressure, respectively. The matrices C(u),  $D \in R^{m \times m}$  are the convective and diffusive operators, respectively. Note the u-dependence of the convective operator (non-linear operator). Finally,  $G \in R^{m \times q}$  represents the gradient operator, and the matrix  $M \in R^{q \times m}$  is the divergence operator.

For the discretization of the momentum equation (7), a second order backward difference scheme for the time derivative term, a fully explicit second-order one-leg scheme (Verstappen and Vedman, 2003) for the convective and diffusive terms, and a first-order backward Euler scheme for the pressure gradient are used. Our spatial discretization schemes are conservative, i.e., they preserve the kinetic energy equation, which allow good stability properties even at high Reynolds numbers and with coarse meshes. These conservation properties are held if and only if the discrete convective operator is skew-symmetric ( $C_c(u_s) = -C_c^*(u_c)$ ), if the negative conjugate transpose of the discrete gradient operator is exactly equal to the divergence operator  $-(\Omega_c G_c)^* = M_c$ , and if the diffusive operator  $D_c$  is symmetric and positive-definite. To solve the velocity-pressure coupling, a classical fractional step projection method (Verstappen and Vedman, 2003) is used,

$$\boldsymbol{u}_{\boldsymbol{C}}^{\boldsymbol{P}} = \boldsymbol{u}_{\boldsymbol{C}}^{n+1} + \boldsymbol{G}_{\boldsymbol{C}} \widetilde{\boldsymbol{p}}_{\boldsymbol{C}} \tag{9}$$

where the pseudo-pressure is  $\tilde{p}_C = \tilde{p}_C^{n+1} \Delta t / (\beta + 1/2)$  and  $u_C^P$  is the predicted velocity. The discrete Poisson equation for  $\tilde{p}_C$  is obtained by taking the divergence of equation 9 and after applying the incompressibility condition,

$$L_C \widetilde{p}_C = M_C u_C^P \quad (10)$$

where discrete Laplacian operator  $L_C \in R^{qxq}$  is, by construction, a symmetric positive definite matrix  $(L_C \equiv M_C \Omega^{-1} M_C^*)$ . Once the solution is obtained,  $u_C^{n+1}$  results from the correction:  $u_C^{n+1} = u_C^P - G_C \tilde{p}_C$ .

### 3.2 Large Eddy Simulation Model

In the quest for a correct modeling of Navier Stokes equations, they can be filtered spatially like in Large Eddy Simulation (LES). Doing so, the filtered non-linear convective term must be modeled,

$$\rho \frac{\partial \overline{u}_c}{\partial t} + C(\overline{u}_c) \overline{u}_c + D \overline{u}_c + G \overline{p}_c = C(\overline{u}_c) \overline{u}_c - \overline{C(\overline{u}_c)} \overline{u}_c \approx -\frac{\partial \tau_{ij}}{\partial x_j}$$
(11)

where the filtered velocity is denoted by  $\overline{u}_c$  and the SGS stress  $(\tau_{ij})$  is defined as,  $-2\nu_s\overline{S}_{ij} + \frac{1}{3}\tau_{ij}\delta_{ij}$ . Now, we only need to define a suitable expression for the SGS viscosity. Smagorinsky has proposed de following model (Sagaut, 2001),

$$\nu_s = (C_x \Delta)^2 \left( 2 |\bar{S}_{ij}|^2 \right)^{\frac{1}{2}} \tag{12}$$

Unfortunately this model is not appropriate in the close vicinity of a solid wall subject to dominant molecular-viscosity effects. To overcame these limitations Yoshsizawa has derived non-equilibrium fixed-parameter SGS model (Yoshizawa et al., 2000),

$$\nu_{s} = C_{vs} \Delta \| \overline{u}_{c} - \overline{\overline{u}}_{c} \| \left( 1 - exp \left( - \left( C_{w} \frac{\| \overline{u}_{c} - \overline{u}_{c} \|}{\overline{s} \Delta} \right)^{2} \right) \right)$$
 (13)

Where  $\bar{u}_c$  is the doubly filtered velocity and  $\bar{s} = \left(2\left|\bar{S}_{ij}\right|^2\right)^{\frac{1}{2}}$ . In this model the equilibrium of SGS fluctuation is not assumed, no use is made of wall-unit distance based on the friction velocity and the near-wall asymptotic behavior of the SGS viscosity is fullfited. None of these properties are included in the standard Smagorinsky model. In fact, in his original paper, Yoshizawa's final conclusion is that this model possesses the features similar to the dynamic SGS modeling without the classical instability of the dynamical models.

### 3.3 CFD&HT code verification

TermoFluids verification was performed in a previous work (López *et al.*, 2009). A well-established verification case with benchmark solution was selected from the literature taking into consideration some important geometrical and phenomenological connections between that and the one studied in this paper.

## 4. COMPUTATIONAL DOMAIN, MESH AND BOUNDARY CONDITIONS

In this section computational domain, mesh and boundary conditions are described. Figure 3 shows a general view of the computational domain, defined by the fluid that fills the volume between the two edge-rounded cylinders. Each element is modeled as a component of the compressor, i.e. the inner cylinder [4] represents a set of components such as the crankcase, the motor, the compression chamber, the suction muffler and so on, the external cylinder [5] represents the compressor shell and the input/output orifices [1] and [2] respectively would be the suction area. Note the presence of the tube [3] which represents the discharge tube of the compressor.

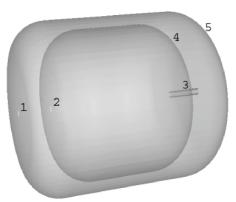


Figure 3: General view of the CFD&HT computational domain.

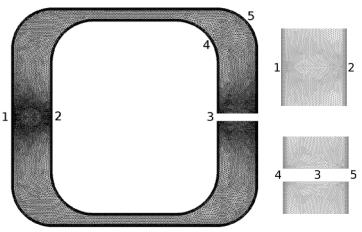


Figure 4: Elevation view of the unstructured mesh (left) and input/output and discharge tube details (right).

This approximation allows studying different suction configurations (e.g. by changing tube length or input/output orifices shape) and therefore the detailed study of the circulating flow influence on compressor behavior. The dimensions of the elements have been fixed from measurements over a compressor prototype.

Figure 4 depicts the computed mesh and zoom views. This mesh has been constructed from the cylindrical extrusion (with 16 planes from 0° to 360°) of the half-domain mesh; this explains the symmetry in Figure 4. This kind of mesh allows the use of Fast Fourier Transform (FFT) solvers and therefore a significant reduction of the computational cost. Note the presence of the prismatic layer as well as the jet area and the tube surrounding refinement.

Regarding to the boundary conditions, velocity and temperature are given at the input orifice [1] while pressure is fixed at the output [2]. This pressure is updated at each time step depending on the evolution of the compressor chamber and the other internal elements. At the beginning the fluid remains at constant temperature. The walls have been set to Dirichlet boundary condition and null velocity. The outer cylinder surface is set up at 40 °C (cold wall), the inner cylinder surface is set up at 50 °C (hot wall) and the tube surface temperature is given by a lineal function between the hot wall and the cold one. Reynolds and Prandtl numbers are set to 10000 and 0.74, respectively.

## 5. ILUSTRATIVE RESULTS

As was stated before, the circulating gas into the shell (i.e. fluid between the shell and the internal elements) was established as a CFD&HT object while the other elements (i.e. tubes, chambers, valve orifices, etc.) were established as one-dimensional objects. In this section, first detailed numerical results of this new CFD&HT object are presented.

## 5.1 Numerical results on circulating compressor gas

An unstructured mesh with approximately 250000 CVs has been used to run the simulation. The large number of CVs and the complexity of the phenomenon have led to use four nodes (8 cores each) from CTTC-JFF cluster during three weeks. This case mixes both natural and force convection at high Reynolds number. A lot of turbulent scales have been observed from the instantaneous contour maps obtained during the simulation (see Fig. 5), which explains the necessity of so much CVs. Moreover, to solve viscous effects at the boundary layer all boundaries have been refined by means of a prismatic layer.

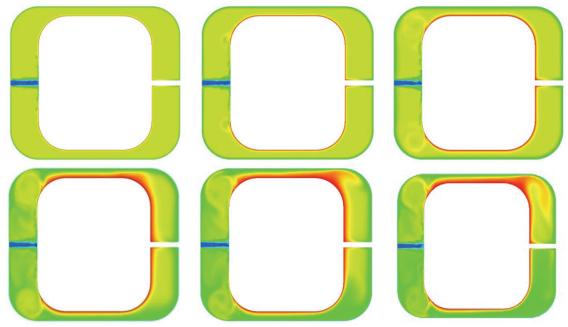


Figure 5: Temperature field at different time steps of the simulation.

Figure 5 depicts the temperature maps of the initial instants of the simulation. Note the jet in the suction area at the left side which evolves quickly generating two main eddies at both sides (up and down) like in a jet impingement.

Regarding the right side of the computational domain, it is mainly characterized by diffusion, probably due to the not too large temperature difference between the hot and the cold walls (inner and outer walls respectively). The thermal boundary layer becomes detached at the right-upper area and it is spread out throughout the right side. This effect, which leads to the heating of the whole domain, was observed in previous simulations (López *et al.*, 2009) although then the transport phenomenon was lead by natural convection due the high Rayleigh number.

## 5.2 Numerical results on 1D compressor elements

The results presented in this section show the influence of the detailed simulation over the one-dimensional objects. Figure 6 shows temperature, mass flow and pressure evolution during one cycle at the output of the CFD&HT object (i.e. at the muffle surround). Note the pressure registered at this point is highly oscillatory; since this value is used as a boundary condition of the CFD&HT domain, a bypass strategy had to be used on the CFD&HT object to avoid compressibility effects, which are not allowed by the implemented solver. Regarding temperature and mass flow profiles, it is interesting to remark that there are slight differences between the beginning and the end of the cycle, probably due to the stationary statistic regime at the CFD&HT object and the accuracy imposed; this is not observed on pressure profile where these values perfectly match.

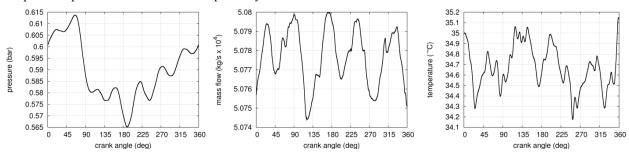


Figure 6: Temperature, mass flow rate and pressure cyclic profiles at the muffler.

Figure 7 shows temperature profile throughout the compressor (left) and a PV diagram at the compressor chamber under cyclic conditions (right). Left Figure depicts the temperature of the refrigerant at each part of the compressor at different piston positions; suction and discharge lines represent internal circuitry such as tube and chamber objects. As it was expected the refrigerant is always heated at the suction area, although this depends on the crank angle. Note also the effect of the discharge tube which is large enough to diminish the temperature at the desired value. Regarding the PV diagram it seems all right, the pressure ratio is in harmony with the one was set up at the simulation parameters. Some pressure picks are observed at the discharge step due to the valve plate characteristics.

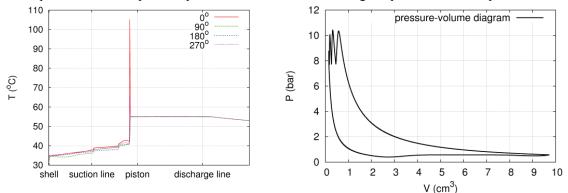


Figure 7: Temperature profile at different objects of the compressor and PV diagram of the compressor chamber.

#### 6. CONCLUSIONS

The present paper is a coupling between the one-dimensional numerical simulation for compressors behavior (Damle 2008) and a CFD&HT code (TermoFluids, Lehmkuhl *et al.*, 2007), numerically verified by López (2009). A

new CFD&HT solver object for the circulating gas into the shell of the compressor has been developed and successfully coupled with the other compressor objects. The whole domain of a hermetic reciprocating compressor has been resolved together and the new CFD&HT object has been tested. Therefore, first detailed numerical results of the circulating gas between the shell and the compressor elements have been obtained.

Future actions may be focused in the experimental validation of the 1D elements coupled with the new defined CFD&HT object. Other actions will be centered in the resolution of the solid elements of the compressor which are currently treated as solids at constant temperature; the coupling between solid domains, CFD&HT objects and the one-dimension approach is the next step. This line of research will allow new high quality simulations, where all parts of the compressor domain will be represented by its own object. Unknowns such as the percentage of circulating flow in the suction area, optimal input/output orifices distance or muffle geometry will be able to be numerically solved soon.

#### NOMENCLATURE

#### REFERENCES

- R. Damle, J. Rigola, C.D. Pérez-Segarra, A. Oliva, 2008, An object oriented program for the numerical simulation for hermetic reciprocating compressors behaviour, 19th International Compressor Engineering Conference, Purdue University, West Lafayette, Indiana, USA, 1402, p.1-8.
- O. Lehmkuhl, R. Borrell, C.D.Pérez-Segarra, M. Soria, A. Oliva, 2007, A new parallel unstructured CFD code for the simulation of turbulent industrial problems on low cost PC cluster, *Parallel Computational Fluid Dynamics*, Ankara, Turkey.
- J. López, J. Rigola, O. Lehmkuhl, A. Oliva, 2009, Numerical study of suction gas flow in the shell of hermetic refrigeration compressors, *International Conference on Compressors and their Systems*, City University, London, UK, C682/037, p. 395-404.
- S.V. Patankar, 1980, Numerical heat transfer, Hemisphere Publishing Corp, USA, 197 p.
- P. Sagaut. Large Eddy Simulation for Incompressible Flows, 2001, Springer-Verlag, Germany, p. 316.
- F.X. Trias, M. Soria, C.D. Pérez-Segarra, A. Oliva, 2006, A direct Schur-Fourier decomposition for the efficient solution of highorder Poisson equations on loosely coupled parallel computers, *Numerical Linear Algebra with Applications*, vol. 13, p. 303-326.
- R.W.C.P. Verstappen and A.E.P. Vedman, 2003, Symmetry-Preserving discretization of turbulent flow, *Journal of Computational Physics*, vol. 187, p. 343-368.
- A. Yoshizawa, K. Kobayashi, T. Kobayashi, N. Taniguchi, 2000, A non-equilibrium fixed parameter subgrid-scale model obeying the near wall asymptotic constraint, *Physics of Fluids*, vol. 12, no. 9, p. 2338.

## **ACKNOWLEDGEMENT**

This work has been financially supported by the Ministerio de Ciencia e Inovación, Secretaría de Estado de Investigación, Spain (ref. ENE 2008-06667).