Purdue University
Purdue e-Pubs

International Compressor Engineering Conference

School of Mechanical Engineering

2012

Use of a Low-Mach Model On a CFD&HT Solver for the Elements of An Object Oriented Program to Numerically Simulate Hermetic Refrigeration Compressors

López Mas Joan
joanl@cttc.upc.edu

Lehmkuhl Oriol

Rigola Joaquim

Carlos D. Pérez-Segarra

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

Use of a Low-Mach Model on a CFD&HT Solver for the Elements of an Object Oriented Program to Numerically Simulate Hermetic Refrigeration Compressors.

Joan LÓPEZ[1], Oriol LEHMKUHL[1/2], Joaquim RIGOLA[1*], C.D. PÉREZ-SEGARRA[1]

Universitat Politècnica de Catalunya (UPC),
[1]Centre Tecnològic de Transferència de Calor (CTTC),
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@termofluids.com

ABSTRACT

A powerful object oriented approach for the simulation of generic thermal systems (Damle et al., 2008) is used as a framework to numerically simulate the thermal and fluid behavior of hermetic reciprocating compressors. A physical abstraction of the compressor system provides a vertex-edge graph, defining the elements and the neighborhood relations of the system to be solved. Each one of these resulting elements is modeled in order to be solved by itself by giving their respective boundary conditions. Since each element provides its own solver tool, the coupled system can be solved in an integrated form.

Into previous works, an unstructured and parallel object oriented Computational Fluid Dynamics and Heat Transfer code (from now on CFD&HT) for accurate and reliable solving of turbulent industrial flow, called TermoFluids (Lehmkuhl et al., 2007), was used to provide with CFD&HT capability the system elements (López et al., 2010). In this work, a Low-Mach based CFD&HT module (Chiva et al., 2011) implemented within the TermoFluids software has been used solve the fluid domain existing inside the shell of a reciprocating compressor, which is identified as one of the compressor elements in the abstraction stage.

This improvement allows us to numerically simulate the recirculation flow inside the shell of a reciprocating compressor, providing detailed information about suction area of the compressor and allowing study of new geometric configurations of such part. Furthermore, in comparison with previously tested CFD&HT modules, the Low-Mach model allows better treatment of the compressibility effects generated at the inner elements of the compressor such as chambers, tubes and undoubtedly the compression chamber.

1. INTRODUCTION

The use of new refrigerants, new compressor circuitry, new designs, some aspects like several parallel paths, more than one compression chamber, etc., calls for improvement and innovation in the development of new modeling strategies for hermetic reciprocating compressor analysis. Thermal and fluid dynamic behavior of compressors is characterized by complex heat transfer and fluid flow phenomena: three-dimensional, turbulent, fast transient, etc. In this respect, the use of CFD&HT codes may help in the improvement of some specific critical points as well as in optimization of compressor efficiency.

A clever programming strategy such as the object oriented approach assures flexibility, reusability, easy maintenance etc. of the code. In order to follow such a strategy an abstraction effort is required. During this process several elements with its own physical and geometrical entity within the compressor-system are identified. Further analysis of each element allows their individual modeling.

In such way the compressor- system may be seen as a set of linked elements and can be represented using a vertex-edge graph in which the vertexes would be the compressor elements and the edges would represent neighborhood or linkage relations (see Fig. 1b). The lasts impose the boundary conditions for each element. Therefore, the resolution of the compressor-system may be seen as the individual resolution of each one of the identified elements with a certain boundary conditions established by their neighbors in the graph.

Some of the resulting elements from the abstraction process do not require CFD&HT analysis at all since it is not necessary such detail of description. Instead, zero and one-dimension models are far enough good to describe what is happening inside (e.g. tubes, chambers, resonating chambers, etc.). On the other hand, one element could require a more detailed simulation but it is still unfeasible at this moment due the complexity of its physics (e.g. the compression chamber). Finally there are some elements on which a CFD&HT analysis could provide interesting and useful detailed information which could help to improve compressor performance (e.g. by improving the geometry of the suction area).

In this work it is carried out the simulation of the same compressor taken in previous works (López et al., 2009) and (López et al., 2010). In those works, first numerical results from the coupling of CFD&HT analysis and zero and one-dimension models were presented. Now two aspects are improved: firstly the parallelization of the compressor-system and secondly the mathematical model of the Navier-Stokes equations.

Nowadays, the use of CFD&HT codes easily implies the necessity of modern parallel computational infrastructures due the huge computational effort required to numerically simulate turbulent fluid flows with unstructured meshes of millions of control volumes. This means that the developed codes not only must be parallel but also must be easy to maintain in order to run cases on the latest hardware infrastructures. The new code has been improved in this respect and replaces older non-friendly and complex versions. The new strategy is based on direct parallelization of the graph that results from the abstraction process and allows simulating compressors and thermal systems in general independently of the complexity of the models used by the conforming elements.

Regarding the mathematical model of the Navier-Stokes equations a new Low-Mach based algorithm is used. In previous simulations an incompressible model based on the Boussinesq approximation was used, causing serious stability problems due to the non-negligible oscillations of the pressure and mass flow at the outlet orifice of the suction area (i.e. the motor-side suction tube). The new methodology successfully addresses this problem since it allows small compressibility effects inside the computational domain solving all stability problems.

2. NUMERICAL SIMULATION OF COMPRESSOR BEHAVIOUR SEEN AS A THERMAL SYSTEM

The abstraction process of the compressor-system is the first step in the compressor modeling strategy. This point shows an example of this process, followed by a brief explanation of the mathematical models (zero and one-dimensional models) and some details about the global resolution algorithm. Deeper understanding may be found in (Pérez-Segarra et al., 2003), (Damle et al. 2008) or (López et al., 2010).

2.1 Abstraction process
The abstraction process is done in order to identify all elements conforming the compressor-system. Figure 1 shows a compressor seen as a set of basic elements: tubes, chambers, etc. (see Fig. 1a) and its corresponding vertex-edge graph of the scheme (see Fig. 1b).

The graph shows all the elements of the scheme and their neighborhood relations. It also shows different groups of processors distinguished by color. It is important to note that this is only an illustrative example showing a partitioned graph. Using two groups of processors could increase significantly the latency time between processors (i.e. the processors communicate more data each other) and consequently the computational time would increase as well. Therefore distributing computational load does not necessarily imply the speedup of the simulation. Processors distribution is not trivial, usually system graph partitioning poses a complex problem, especially when vertexes computational load is not homogeneous and these are highly coupled (i.e. many linkage relations).
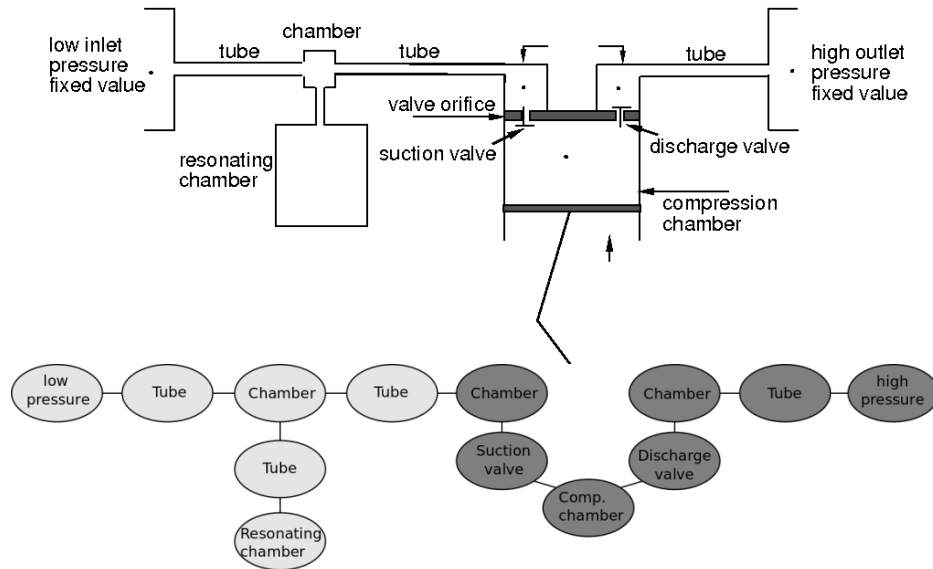
Figure 1: Scheme of a compressor up (a), and its graph representation down (b).

2.2 Mathematical models

The compressor element implementing zero and one-dimensional models are divided into strategically distributed control volumes (CVs). The general conservation equations of the fluid flow are semi-discretized as:

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

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

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

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 ρ are obtained from continuity equation (1), energy equation (3) and state equation ρ=(p,h) respectively, and are evaluated at each 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).
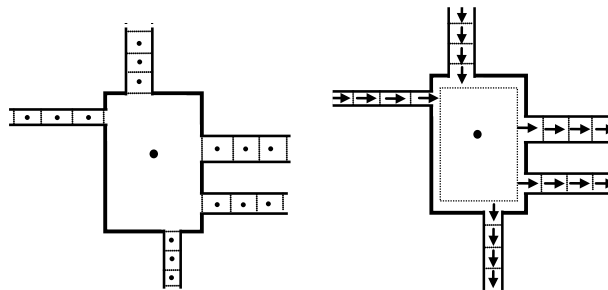
Figure 2: CVs distribution over tubes and chambers: left (a) center grid nodes; right (b) staggered mesh.

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 the same way, pressure correction approach for the compression chamber is used as well. 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 V0 as the volume of chamber remains constant.

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 and outlet pressures and its pressure ratio.

Fixed value objects serve as boundary conditions (e.g. providing pressure, temperature, humidity, etc.).

Regarding those elements modeled by CFD&HT they are solved by using TermoFluids code (Lehmkuhl *et al.*, 2007) which is presented in the following point.

2.3 Global resolution algorithm
The global resolution algorithm is aimed to solve the whole compressor-system, and the procedure applies to all elements, independently of the implemented model. Therefore the introduction of CFD&HT analysis does not alter the resolution algorithm and the elements are still treated in a generic way.

All compressor elements are capable of solving itself for given the necessary boundary conditions. These boundary conditions are taken from the neighbor elements at each system iteration in order to solve momentum, pressure correction and energy equations of the element. The solution of the conservation equations of the element provides the set of boundary conditions for neighbors' resolution at the next iteration. Iterations continue until convergence is reached at a given time step and then the next time step calculation starts after updating the variables.

The above strategy is based on a Jacobi method to solve a system of equations. The introduction of alternative solvers such as based in Newton-Raphson strategies or genetics based solvers will be object of study in the future. For the time being, the current implemented methodology is far enough to solve the compressor-system problem with the prescribed conditions.

3. CFD&HT CODE (TERMOFLUIDS)

An unstructured and parallel object-oriented code called TermoFluids (Lehmkuhl *et al.*, 2007) has been used in several works (López et al., 2009), (López et al., 2010) and in the current work to numerically simulate the fluid flow in the space between compressor shell and compressor crankcase with the aim of determine its influence on suction return design and compressor analysis. TermoFluids 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. TermoFluids provides a large family of CFD&HT solvers. Incompressible models use fully conservative second-order schemes for spatial discretization. On the other hand, Low-Mach based algorithms uses non-conservative form of the scalars transport equations in order to use explicit time integration.

Local refinement of the grid is used. For the solution of the pressure equation, it is used a Direct Fourier Schur Decomposition (Borrell *et al.*, 2011) using sparse Cholesky for the local variables with an iterative or direct solver for the interface system.

3.1 Low-Mach Navier-Stokes equations
Incompressible formulations using the Boussinesq approximation with constant fluid properties cannot correctly describe fluxes with high density variations. On the other side, compressible formulations of the Navier-Stokes equations applied to study of such flows are still unfeasible due to stability issues resulting in strong time step limitations.

To address this situation, (Chiva et al., 2011) proposed an extension of an incompressible pressure projection-type algorithm (fractional-step) to simulate Low-Mach fluxes, using Runge-Kutta/Crank-Nicolson time integration

scheme, similar to the one presented by (Najm et al., 1998) and (Nicoud, 2000). In addition, this solver allows unstructured finite volume spatial discretization in order to simulate fluxes through complex geometries.

At this time, this extension has been added into the TermoFluids code as a new CFD&HT solver. Thus, we are currently in the possibility to solve Low-Mach flows on complex geometries such as the space between compressor shell and compressor crankcase which is object of study in this work.

4. COMPUTATIONAL DOMAIN, MESH AND BOUNDARY CONDITIONS

In the current work it is carried out the numerical simulation of the same compressor simulated by (López et al., 2010), with the same operation and boundary conditions. This allows taking advantage of all previously acquired knowledge as well as serves as reference for new numerical results. At the same time allows appreciating whereas the simulation is improved with both the new CFD&HT model and the new parallelization strategies. Therefore, for a detailed description of the computational domain, mesh and the boundary conditions the reader should see the above reference.
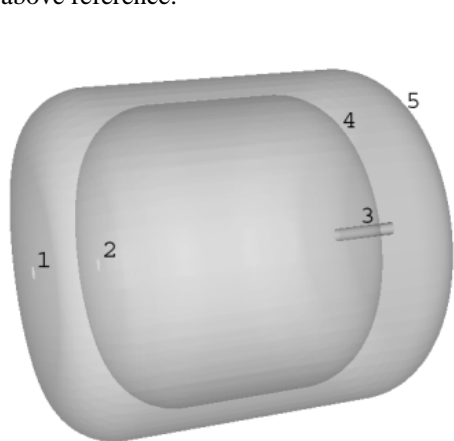
Figure 3: General view of the CFD&HT computational domain.
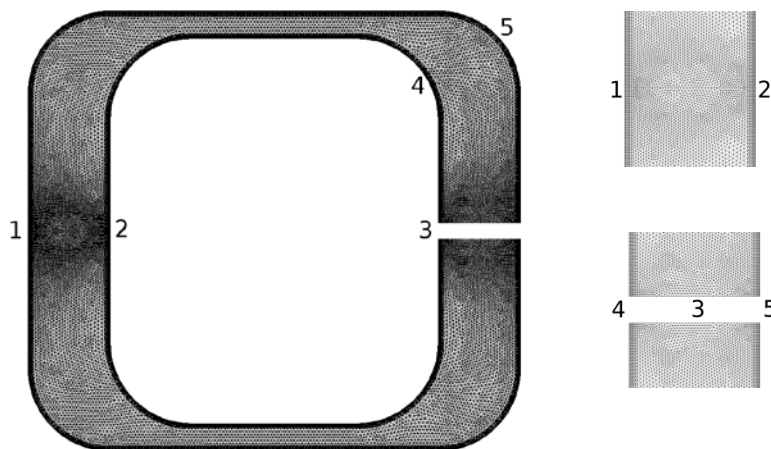
Figure 4: Elevation view of the unstructured mesh (left) and input/output and discharge tube details (right).

Figures 3 and 4 depict the computational domain and its mesh discretization. The CFD&HT domain which is bounded by the compressor shell [5] and the internal elements [4] was modeled as a pair of two edge-rounded cylinders. This approximation allowed studying different suction configurations (e.g. by changing input/output orifices [1/2] distance or shape, discharge tube [3] length, etc.). and consequently clears several unknowns about the influence of the circulating flow on compressor behavior. The dimensions of the elements have been fixed from measurements over a compressor prototype.

5. NUMERICAL RESULTS

## 5.1 Experimental comparison of the compressor work and COP
Three cases with different operation conditions on the same compressor have been simulated. For all cases the condensation pressure at 55 ºC (ASHRAE standard) for R600a has been set as the discharge pressure of the compressor. In Table 1 numerical and experimental values of the compressor work and COP at the three used operation conditions are shown as well as the resulting relative errors.

It is observed that numerical predictions are in concordance with experimental values. Relative errors between numerical and experimental data are below 10%. The use of more realistic boundary conditions at the CFD&HT element simulation could improve numerical outputs, especially if a better temperature distribution at domain boundaries is imposed.

Table 1: Numerical and experimental data for the simulated compressor.

Case	Tev (ºC)	Π	W	COP
Num.	-15.0	8.70	121.9	2.08
Exp.			111.9	2.27
% relative error			8.9	8.1
Num.	-23.3	12.33	95.7	1.83
Exp.			90.9	1.92
% relative error			5.3	4.8
Num.	-25.0	13.29	90.5	1.70
Exp.			85.3	1.82
% relative error			6.1	6.5

The Figure 4 shows the pressure volume diagrams for all the simulated cases. As it was expected the hotter is the evaporation temperature the bigger is the surface area inside the PC diagram. This can also be appreciated in the work column of Table 1.
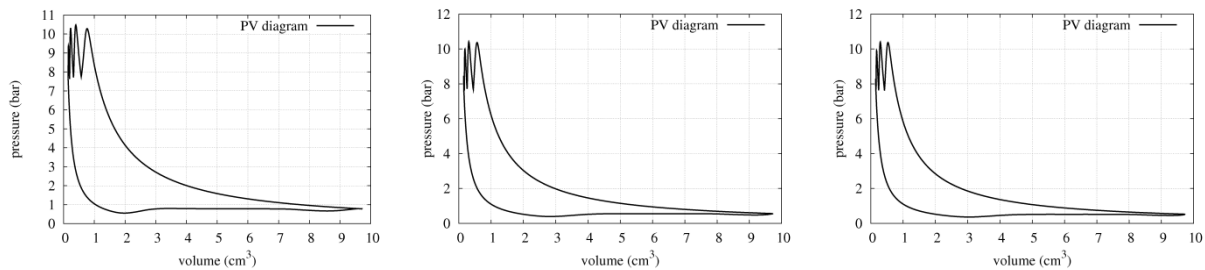

Figure 4: Pressure volume diagrams of the compressor chamber for all three evaporation temperatures. Tev from left to right: -15ºC, -23.3 ºC, -25.0 ºC.

5.2 Duality of phenomenology: Natural and Forced convection

The following figures show the transient evolution of several samples of pressure, temperature and velocity (x component) averages. Compressor cycle is divided in several discrete blocks in order to generate these samples. Samples of a given $\emptyset$ variable are obtained by evaluating the average in time $\overline{\emptyset}$ (Equation 4) at each of these blocks cycle after cycle. In this way piston frequency is removed from $\emptyset$ in order to separate cyclic perturbation from turbulence phenomenon.

$$\overline{\emptyset} = \frac{1}{T} \int_t^{t+T} \emptyset(t)dt \qquad (4)$$

Therefore Figure 4 shows the evolution of the discrete average values of three different variables. Each cycle is bounded by vertical lines. In the suction area figures (up), it is observed statistic stationary (or almost) cyclic conditions which is not appreciated in the discharge area figures (down). This observation suggests that there exists a duality between natural and forced convection in the phenomenology produced inside the compressor shell. However, it should be considered again more realistic boundary conditions at the walls of the CFD&HT domain, since temperature gradients and/or temperature oscillations at the walls could introduce important effects in this sense.
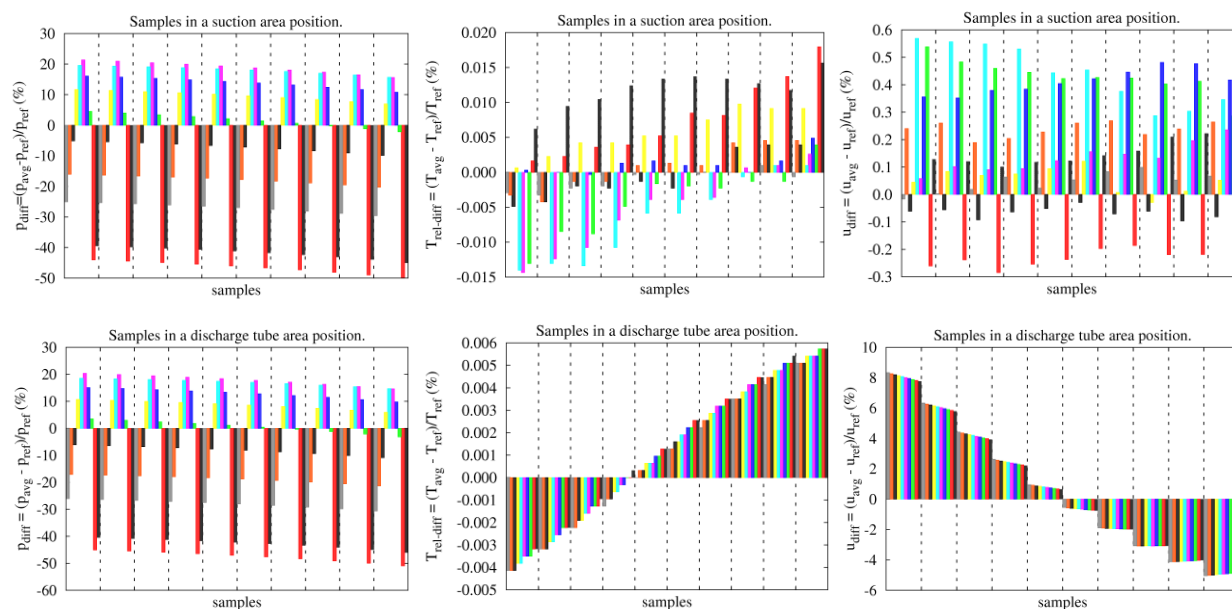
Figure 5: Pressure, velocity (x component) and temperature average samples at two different positions.

5. CONCLUSIONS

This paper is a new contribution to the study of a compressor-system and thermal systems in general continuing the research line of previous works of the group. The coupling between CFD&HT codes and zero and one-dimensional models in order to solve compressor-systems at different levels of modeling was already reached. However it was seen that the full explicit incompressible CFD&HT code did not have a good behavior indicating that the model should be reviewed. Moreover it was also seen the need for a clever parallel programming strategy to develop an easy, intuitive and friendly code in order to be useful in the future as the hardware improves.

In this respect, the current paper shows some illustrative cases simulated with an improved version of an older code which provide new requirements. Older stability problems have been solved by using a new Low-Mach based algorithm provided by TermoFluids. The new model allows simulation of fluxes with high density variations at low Mach numbers which is used in this work to absorb the compressibility effects due pressure and mass flow oscillations at the suction tube of the compressor. On the other hand, the new parallelization strategy does not only allow CFD&HT integration in the element modeling but also allows compressor-system parallelization independently of the nature of its compound elements.

Three cases with different operation conditions on the same compressor have been simulated. Numerical predictions agree with experimental values which verify that the code is able to provide global results as well as detailed information of its elements. Here it is also important to note that the developed code is still providing satisfactory results after the introduction of CFD&HT analysis. Therefore the coupling between CFD&HT codes and zero and one-dimensional codes has been achieved successfully.

On the other side, both natural and forced convection phenomena have been observed together though numerical simulations. Note that one could figure out the duality inside the compressor but from now on, CFD&HT simulations allow these kinds of observations which were not possible before with other modeling strategies. That is, natural and forced convection duality is now being observed in detail thanks to numerical analysis.

The temperature of the compressor shell and the internal elements (i.e. motor, crankshaft, chambers and tubes) has been assumed constant during the simulation. Take into account the transient evolution of such temperatures as well as the axial heat flows through solid elements due to possible temperature gradients are interesting problems to be solved and will be faced soon. Code infrastructure for the simulation of thermal systems by following an object

oriented strategy is already available. Moreover this allows system resolution at different levels of modeling (e.g. SIMPLE based strategies, CFD&HT models, etc.). However it is still pending the coupling between domains with different physics which poses the problem of solving data transfer between non-matching meshes.

NOMENCLATURE

e_c	kinetic energy	(J/kg)	t	time	(s)	**Subscripts**	
F_s	forces in flow	(N)	V	volume	(m^3)	e	east
h	enthalpy	(J/kg)	T	block length	(s)	w	west
m	mass	(kg)					
$\dot{m}$	mass flow rate	(kg/s)	**Greek letters**			**Superscripts**	
p	pressure	(Pa)	Π	pressure ratio		0	previous
$\dot{Q}_{wall}$	heat rate at the wall	(W)	ρ	density		n+1	next instant
						-	mean value

REFERENCES

S.V. Patankar, 1980, *Numerical heat transfer*, Hemisphere Publishing Corp, USA, 197 p.

Damle, R. and Rigola, J. and Pérez-Segarra, C. D. and Oliva, A., 2008, An Object Oriented code for the numerical simulation of hermetic reciprocating behavior. *Int. Compressor Engineering Conference at Purdue*, USA: p.1-8.

Lehmkuhl, O. and Perez-Segarra, C. D. and Borrell, R. and Soria, M. and Oliva, A., 2007, TERMOFLUIDS: A new Parallel unstructured CFD code for the simulation of turbulent industrial problems on low cost PC Cluster, *Parallel Computational Fluid Dynamics*, Ankara, Turkey: p. 1-8.

López, J. and Rigola, J. and Lehmkuhl, O. and Oliva, A., 2010, Introduction of CFD&HT Analysis into an Object Oriented One Dimensional and Transient Program for Numerically Simulate Hermetic Refrigeration Compressors, *Int. Compressor Engineering Conference at Purdue*, USA: p. 1382-1390.

López, J. and Rigola, J. and Lehmkuhl, O. and Oliva, A. 2009, Numerical study of suction gas flow in the shell of hermetic refrigeration compressors, *Int. Conference on Compressors and their Systems*, City University, London, UK: C682/037 p. 395-404.

Chiva, J. and Ventosa, J. and Lehmkuhl, O. and Pérez-Segarra, C.D., 2011, Modelization of the Low-Mach Navier Stokes equations on unstructured meshes, *7th Int. Conference on Computational Heat and Mass Transfer*: p. 403-411.

Habib, N. Najm and Peter, S. W. and Omar, M. K., 1998, A semi-implicit numerical scheme for reacting flow, *J. of Computational Physics*, 143: p. 381-402.

Nicoud, F., 2000, Modelization of the Low-Mach Navier Stokes equations on unstructured meshes. 7th *J. of Computational Physics*, 158: p. 71-97.

Pérez-Segarra C.D., Rigola, J., Oliva, A., 2003, Modelling and numerical simulation of the thermal and fluid dynamic behaviour of hermetic reciprocating compressors. Part 1. Theoretical basis, *Int. J. Heating, Ventilating, Air-Conditioning Refrigerating*, 9: p. 215-236.

Borrell, R. and . Lehmkuhl, O. Trias, F. X. and Oliva, A., 2011, Parallel direct Poisson solver for discretisations with one Fourier diagonalisable direction, Journal of computational physics, 230, 12: p. 4723-4741.

ACKNOWLEDGEMENT