

2016

# On the Use of a Parallel Object-Oriented Code for Solving the Heat Transfer in Hermetic Reciprocating Compressors

Lopez Mas  
UPC, Spain, [joanl@cttc.upc.edu](mailto:joanl@cttc.upc.edu)

Rigola Joaquim  
UPC, Spain, [quim@cttc.upc.edu](mailto:quim@cttc.upc.edu)

Lehmkuhl Oriol  
[oriol@cttc.upc.edu](mailto:oriol@cttc.upc.edu)

Oliva Assensi  
UPC, Spain, [oliva@cttc.upc.edu](mailto:oliva@cttc.upc.edu)

Follow this and additional works at: <https://docs.lib.purdue.edu/icec>

---

Mas, Lopez; Joaquim, Rigola; Oriol, Lehmkuhl; and Assensi, Oliva, "On the Use of a Parallel Object-Oriented Code for Solving the Heat Transfer in Hermetic Reciprocating Compressors" (2016). *International Compressor Engineering Conference*. Paper 2496.  
<https://docs.lib.purdue.edu/icec/2496>

This document has been made available through Purdue e-Pubs, a service of the Purdue University Libraries. Please contact [epubs@purdue.edu](mailto: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>

## On the use of Parallel Object-Oriented Code for Solving the Heat Transfer in Hermetic Reciprocating Compressors

López JOAN<sup>1\*</sup>, Rigola QUIM<sup>1</sup>, Lehmkuhl ORIOL<sup>1</sup>, Oliva ASSENSI<sup>1</sup>

<sup>1</sup>CTTC, Polytechnical University of Catalonia,

Terrassa, Spain

[joanl@cttc.upc.edu](mailto:joanl@cttc.upc.edu)

### ABSTRACT

The heat transport phenomenon in a hermetic reciprocating compressor is addressed in this work. This is far from straightforward. It involves several transient physical phenomena interacting with each other. The heat is exchanged between the refrigerant fluid and the solid parts of the compressor (suction muffler, cylinder head, crankcase, etc.). At the same time, the solid parts exchange heat to each other by means of conduction and radiation. Moreover, the phenomenon happens in non-symmetrical complex geometries and the solid parts are made of different materials. This is interesting from both the software engineering and the compressor design viewpoint. A parallel object-oriented software platform for the resolution of multiphysics problems is employed. This platform allows the use of partitioned strategies so that the compressor heat transport problem -a global problem- can be divided into several smaller parts -local problems-. This makes possible the use of multilevel modeling strategies for thermal systems analysis. Furthermore, in order to couple the several sub-problems in an integrated simulation, the platform provides data transfer tools -for matching and non-matching meshes- to exchange sub-domain state information. In particular, the work provides detailed information on the heat distribution and the temperature of the components of a test compressor. By means of comparative studies the thermal properties of some of its components are analyzed. This highlights the importance of choosing proper materials. For example, different suction muffler materials are tested to investigate their influence on the volumetric efficiency. Since the whole compressor is simulated, the consequences of altering specific component properties are also appreciated on the other components. In sum, the work presents illustrative numerical results of the three-dimensional heat transfer in a compressor that show the potential use of computer simulation to support design of components to attain feasibility and energy efficiency.

### 1. INTRODUCTION

The available computational power has increased significantly in the last decade thanks to the rise of computer clusters. This sets a new paradigm in the development and the usage of software tools for physics simulation. This scenario enables the engineers and scientists not only to incorporate the latest modeling techniques to their codes (LES turbulent flow models, fluid-structure interaction, multiphysics coupling, etc.) but also to exploit classical physical models not feasible before such computing growth (multi-phase modeling, three-dimensional compressible flow, etc.). This work, though an illustrative compressor simulation, aims to show that today's software is a reliable tool to understand better and improve industrial devices and processes. Nowadays, hermetic reciprocating compressors are so optimized that it is becoming increasingly difficult to improve their efficiency. For this reason, one must go to the design details in order to remove the small inefficiencies that prevent improvements in modern compressors. Here, the CFD&HT codes have an essential role. So far, powerful software tools based on one-dimensional mathematical models made it possible optimized compressor designs by means of parametric studies. These tools provide essential global information on the compressor performance such as the cooling capacity and the consumption. Other data such as the transient mass flows in the pipes, the transient temperature maps of the refrigerant, the average temperature of the main solid components, the local pressure perturbation, piston leakage, the sliding-crankshaft mechanism strains and much other local information is also provided indirectly in these one-dimensional simulations (Pérez-Segarra 2004, Rigola 2005).

However, since these models are one-dimensional, their geometry model is too simplified and consequently the effect of the real geometry details on compressor efficiency is not always captured. For instance, the pipes are usually connected to chambers searching an expansion effect of the refrigerant. This expansion depends on the geometries of both the pipe and chamber elements. Although the shape of the pipe is a well-known and simple

canonical geometry, the shape of the chamber can be really complex. Even so, one-dimensional mathematical models for solving this flow transitions only depend on the pipe cross sectional area, its length, and the chamber volume. The mathematical model will provide very realistic results of the flow into the pipe. Nevertheless, the resolution of the flow expansion depends on empirical data, which is not always easy to obtain. Furthermore, some of the common assumptions in expansion-contraction flows are not always reasonable in the context of compressor simulations. Finally, the shape of the chamber is not considered in the model, at least the real one, since it is modeled as a cylinder whose inputs are a diameter and a height.

On the other hand, it is important to note that compressors are particularly complex thermal systems. Not only the time scales of the fluid motion are really small but also all their components are in contact with each other and they are continuously exchanging energy. Moreover, the energy flows in a closed loop due to the presence of the shell which is, excluding the refrigerant itself, the only element that can remove energy to the ambient. Actually, removing heat from a compressor is a difficult matter.

Therefore, understanding the heat transfer details into the compressor device is very important, since most of its inefficiencies are linked to thermal effects (Shiva 1998, Ribas 2008). Both the geometry and the materials of the compressor components (suction muffler, cylinder head, cylinder body, discharge line etc.) are important variables in heat removal. Parameters such as the volume and the thermal diffusivity or the surface size, its roughness and its optical properties are directly related to this process. For this reason, the visualization of the heat fluxes in the compressor components, with real three-dimensional geometries and materials, may help to identify unnecessary effects that produce thermodynamic inefficiencies. In particular, this information could help to reduce the effects of the superheating on the volumetric efficiency of the compressor. Innovative designs of the cylinder head and the cylinder body structures can be analyzed, searching for smart thermal paths aimed to move heat far from the muffler surrounds. For instance, one could ask itself whether the expansion chambers in the cylinder body structure are well positioned or not. By means of multi-dimensional numerical simulation, one could find better positions where to place the chambers.

There are two ways to well understand the thermal behavior of compressor components and its fluid refrigerant in real devices: experimentation and detailed numerical analysis. Experimental techniques, such as the use of thermocouples or infrared thermography may help to obtain compressor temperature maps. However, these techniques are far from easy. They need specific setup for every new prototype, they are time expensive, they are intrusive and some of them lack of detailed information. Moreover, they do not provide information on the heat exchanges between compressor components (Shiva 1998). Alternatively, numerical analysis is much flexible. It can be applied to different designs, with almost no cost, and most times the effort is portable to other devices and technologies since software developers usually focus their codes to general multiphysics purposes, as in the platform is hereby presented.

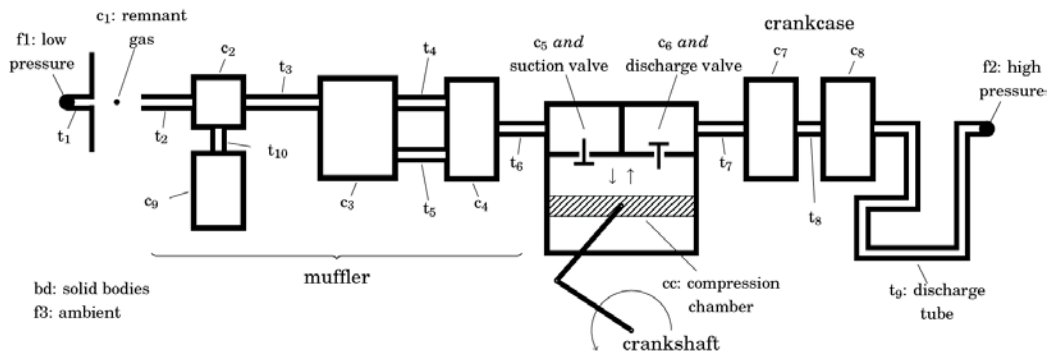
## 2. THE NEST SOFTWARE PLATFORM

### 2.1 Multi-model capability

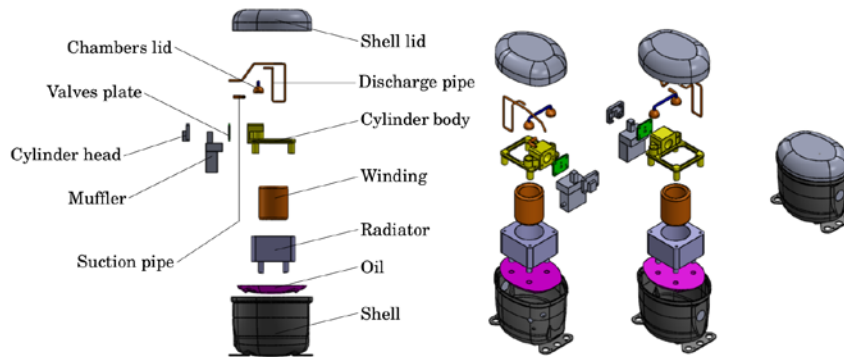
In the simulation of complex systems it is usually interesting to model each component separately by means of a partitioned approach. This brings some advantages from the point of view of software engineering. A big one is the capability to use legacy codes. Another practical advantage is that replacing or removing some component does not affect the remaining components. Hence, you are able to arrange various compressor configurations in a fast and very flexible way since there is no need to modify the program code.

The platform presented here, called NEST (López 2016), is ready for doing this. Regarding to the simulation of hermetic reciprocating compressors in particular, the heat transport is assessed by splitting the compressor into several parts. The main separation is between the refrigerant fluid and the solid components. However, both the refrigerant as well as the set of solid parts are divided into some more sub-parts at the same time. In order to illustrate this, in the scheme of Figure 1 are depicted the several sub-models used for modeling a compressor. This partitioning is based on the several phases that the fluid is submitted to all along the compression path (suction, muffler, compression and discharge line). On the other hand, the main solid components of the compressor are shown in Figure 2. The several parts have distinct shapes and they are made of different materials. The separate treatment of each part permits a better modeling of the heat transfer, since each sub-model can be customized to the particular mathematical and numerical requirements of each part.

Finally, since the model of a particular element depends on the state of their surrounding elements, NEST provides the necessary tools to exchange information between each other. This is implemented in a transparent way, which allows that the users focus their attention on the physical problem instead of spending time understanding coupling issues and software implementation details.



**Figure 1:** NEST model of a compressor (*c*, *t* and *f* stands for *chamber*, *tube* and *fixed value* respectively).

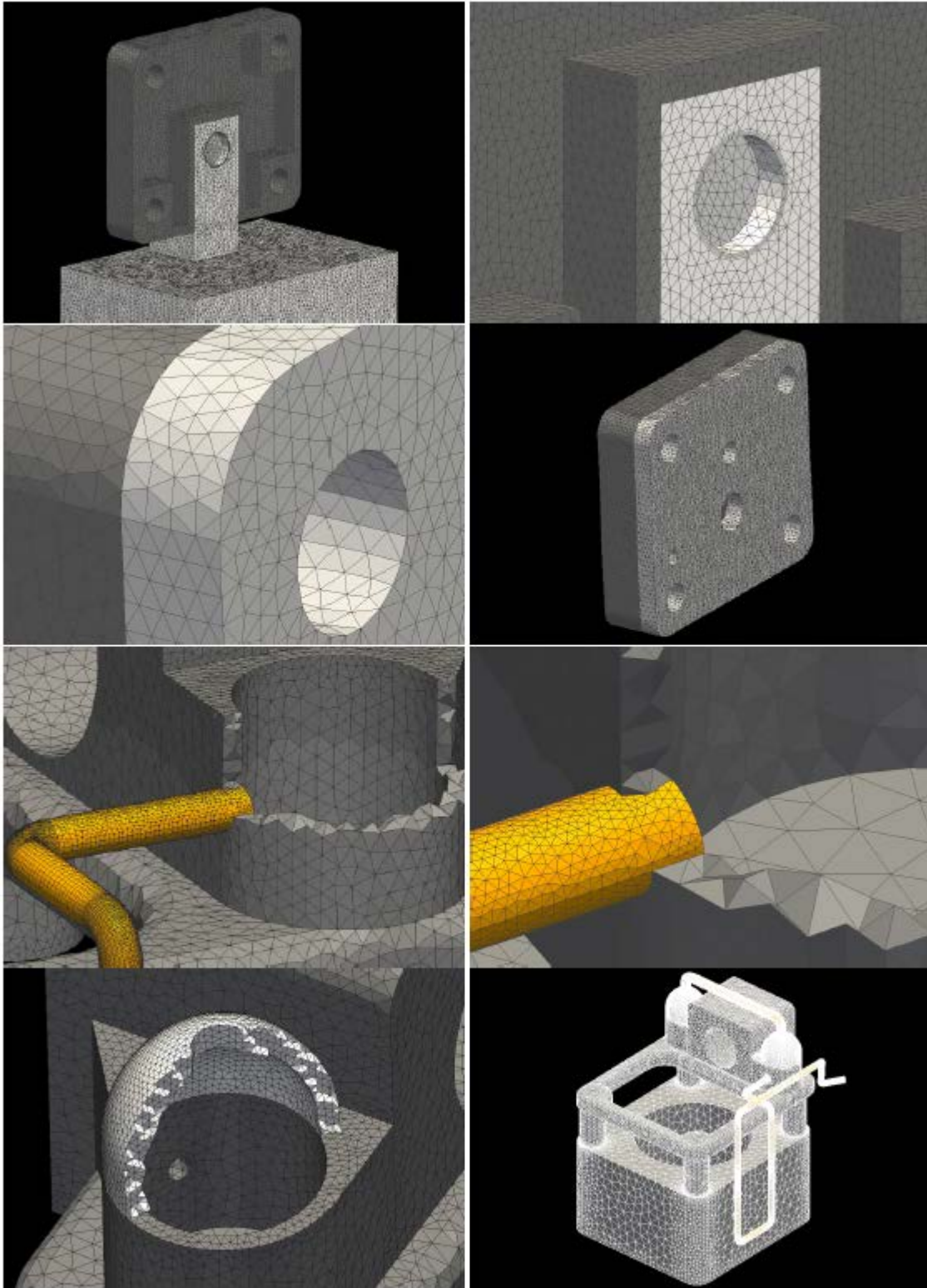


**Figure 2:** Main solid components of the compressor.

## 2.1 Parallel capability and non-matching mesh

It is well-known that the computational expenses are very high when detailed simulation by means of CFD&HT analysis is necessary. However, the last advances in parallel computation make it possible to solve complex industrial problems. Parallel computation is an essential software capability to make use of CFD&HT advances in compressors modeling. NEST software platform is also ready for this. When some model is computational expensive it can be executed in parallel using several processors. Of course this does not affect the resolution of the whole compressor system. Such parts that are solved in parallel are still seen as separate software entities system elements).

One important advantage of the partitioned approach is the possibility to use distinct meshes in each multi-dimensional sub-model. This makes the meshing process much easier since each part is treated apart, which significantly simplifies the geometry of the domain to mesh. For example, you can construct orthogonal meshes in the discharge pipeline, whereas the cylinder body structure or the shell can be meshed using an unstructured meshes. In Figure 3 are shown the meshes for some of the compressor parts. An important drawback of this approach is the mismatch of the meshes at the contact boundaries of the components. This makes data exchange very hard since the data must be transferred properly, assuring accuracy and conservation of the transmitted variables. Moreover, this operation should be fast as it must be performed many times during the simulation. Data transfer is not a straightforward problem. Actually, this is considered a challenging problem in multiphysics simulation (Jiao 2004). Nevertheless, the data transfer methods implemented in NEST permit the use of non-matching meshes.



**Figure 3:** Some views of the meshes used in the numerical simulation. Every component is meshed apart so that the meshes do not match at the interfaces.

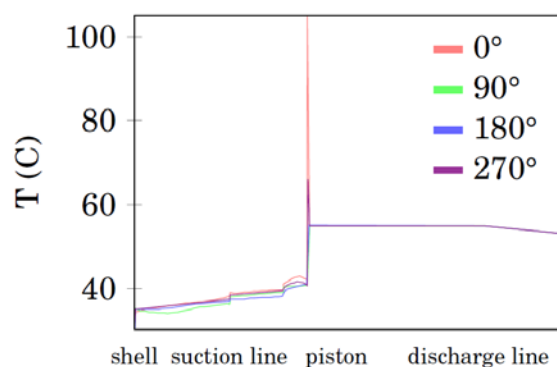
### 3. NUMERICAL EXPERIMENT

In this section the temperature map in a three-dimensional hermetic reciprocating setup is presented. These data are obtained from the numerical resolution of the heat transfer in the compressor components of Figure 2 using NEST software platform. In the background, NEST uses the TermoFluids DiffusionSolver (Lehmkuhl 2009) for resolving the energy equation in the several computational domains. In order to have a complete view of the temperature distribution and the heat flows, the computational domain is composed of all the components where the fluid refrigerant circulates through as well as those are in contact. To get closer to a real-world compressor the parts are made of different materials, for instance the shell is made of steel and the discharge pipeline is made of cooper. Since the overall compressor model is spitted into several sub-models, every component is discretized apart from the others. Hence, there is one discretization mesh per component. In this way, high quality meshes can be obtained easily. Moreover, this makes it possible to modify, replace or remove compressor components without affecting the meshes of the surrounding components. Information about the mesh of each component is presented in Table 1.

**Table 1:** Numerical parameters of the distinct meshes.

Mesh	Cells(x10 <sup>-3</sup> )	Partitions	Component
<i>mpipe<sub>1</sub></i>	5	1	Muffler
<i>mpipe<sub>2</sub></i>	5	1	Muffler
<i>mpipe<sub>3</sub></i>	50	2	Muffler
<i>mcham<sub>1</sub></i>	350	12	Muffler
<i>mcham<sub>2</sub></i>	250	8	Muffler
<i>cylhead</i>	170	6	Cylinder head
<i>valveplate</i>	120	4	Valve plate
<i>cylbody</i>	300	12	Cylinder body
<i>ckchamlid<sub>1</sub></i>	30	2	Crankcase
<i>ckchamlid<sub>2</sub></i>	30	2	Crankcase
<i>ckpipe<sub>2</sub></i>	10	2	Crankcase
<i>radiator</i>	120	4	Motor
<i>winding</i>	25	2	Motor
<i>oil</i>	85	4	Oil
<i>shell</i>	775	20	Shell
<i>dtube<sub>beg</sub></i>	30	2	Discharge tub
<i>dtube<sub>end</sub></i>	20	2	Discharge tub
<i>dtube<sub>bends</sub></i>	170	6	Discharge tub
<i>dtube<sub>extrusions</sub></i>	70	4	Discharge tub
<i>SUM</i>	2615	96	Compressor

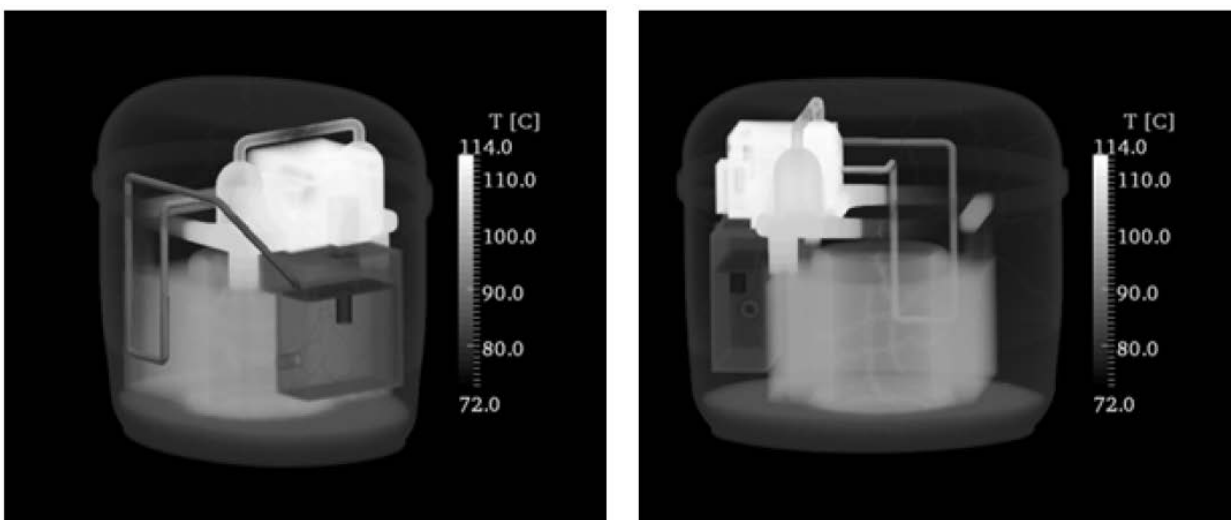
The boundary conditions for each of these components are fixed using the Newton's approach  $q=h(T_{\text{wall}}-T_{\text{gas}})$ . The gas temperature data is based on the refrigerant temperature profile shown in Figure 4. This data is obtained from a simulation of a 9.6cm<sup>3</sup> isobutane compressor using NEST one-dimensional models. The heat transfer coefficients are taken from this simulation as well. They were evaluated using the empirical correlations implemented in the NEST compressors sub-models. The ambient temperature is set at 20C.



**Figure 4:** Temperature profile of the refrigerant fluid at four crankshaft positions.

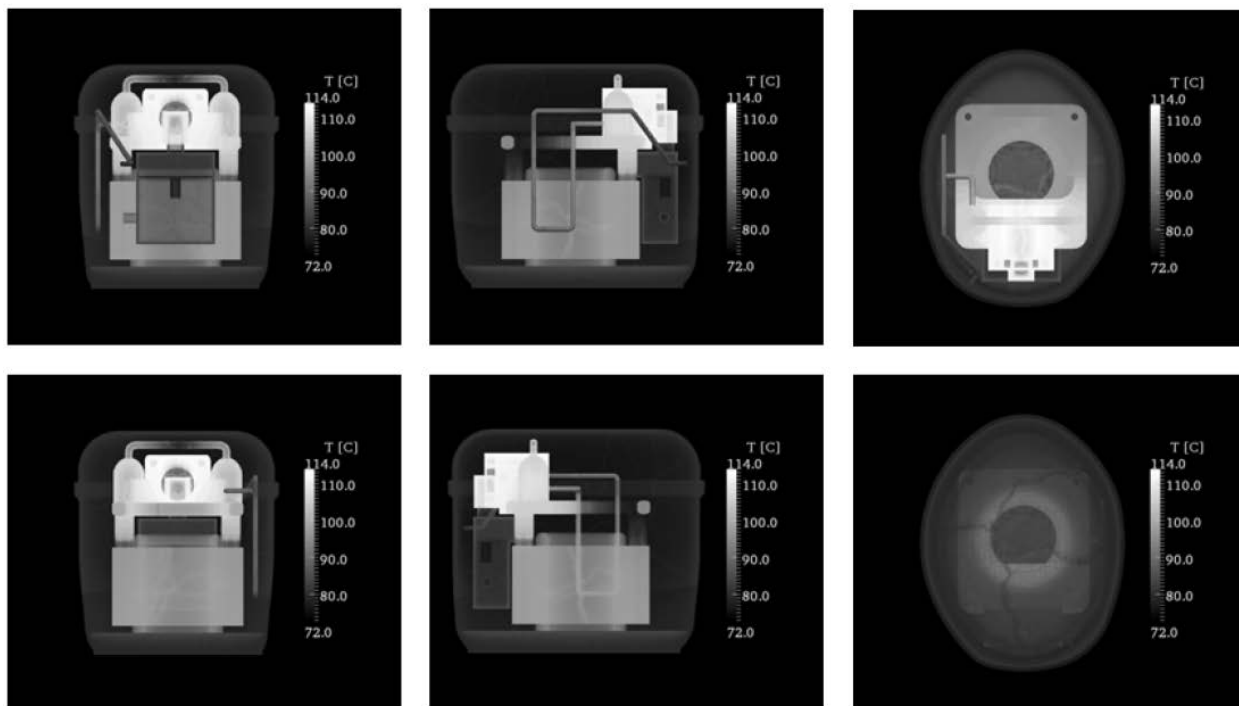
### 3.2 Numerical results

The following figures show several views of the compressor temperature map. One can clearly identify the cold (dark color) and hot (light color) areas in the general perspective views of Figure 5. As expected, the shell is the coldest part while the cylinder surround is the hottest one. Other interesting effects can be observed here. For instance, the chambers in the cylinder body structure are heated by remnant heat flows coming from the cylinder surface. This is a desired effect since the heat is dissipated from the cylinder surrounds to colder parts of the cylinder body structure, which contributes to create a colder environment around the suction muffler. This is very important since the volumetric efficiency can be reduced significantly if the refrigerant is too heated before entering into the cylinder. Furthermore, the backside of the cylinder body structure is about 15C colder than the cylinder surround (see left-right views in Figure 6). This indicates that there is room for intensifying heat removal from the cylinder. On another hand, the images also show how the heat is transferred to the shell first through the radiator and later through the oil pool. Actually the top part of the shell is colder than its bottom. This means that the heat is mainly dissipated thanks to the oil, being the heat convection though the remnant gas not so relevant. In addition, the simulation also shows how the discharge pipe colds down as it is closer to the shell. This temperature gradient indicates that this pipe is also a heat removal path. However, one cannot forget that the refrigerant leaves the compressor through this pipe. Hence, the state of the discharged gas depends on what happens along this pipe.



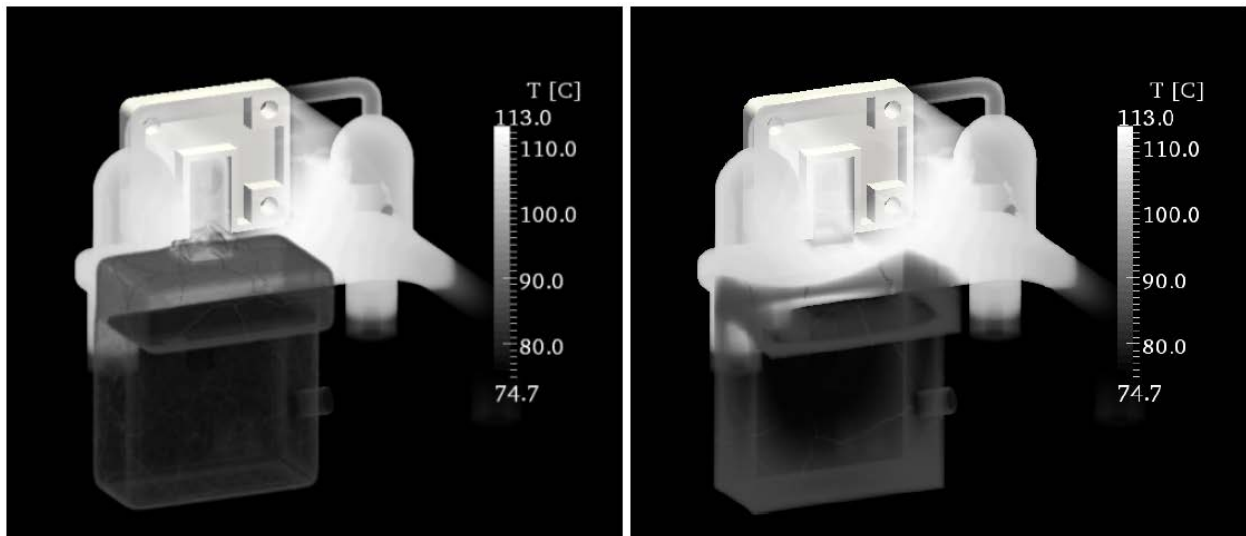
**Figure 5:** Temperature distribution in the compressor setup. The optical properties of these images have been modified in order to emphasize specific temperature levels. Due to this rendering some temperatures are not appreciated.

Moving towards more lightweight geometries of the cylinder body structure could help to reduce the accumulated heat in the solid parts. Indirectly, this could improve volumetric efficiency since it would reduce the temperature of the refrigerant in the suction area. In a similar way, the choice of the employed materials is also critical. In order to show the effect of changing the suction muffler material, a second simulation has been carried out using exactly the same geometry and boundary conditions. In this case, however, the suction muffler is not made of plastic material as usual. Instead, it is done of a hypothetical material, whose thermal diffusivity is ten times the one of the plastic muffler. Figures 7 and 8 show several comparative views of the obtained numerical results. Remember that in these views, the darker is the color the colder is the material. The main difference relies on the temperature, when the thermal diffusivity is low the muffler is colder. Therefore, this numerical experiment shows why suction mufflers are made of plastic.

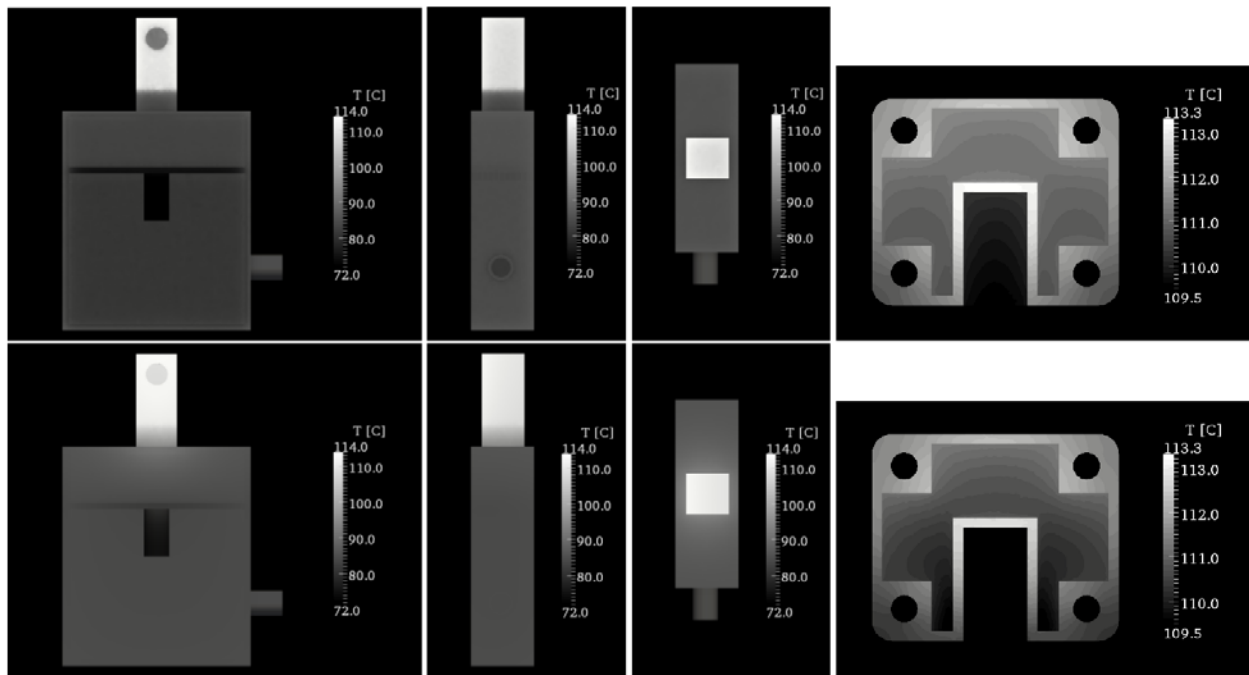


**Figure 6:** Temperature distribution in the planar views of the compressor setup. The presented images correspond to views from: front and back (left column), left and right (middle column) and top and bottom (right column).



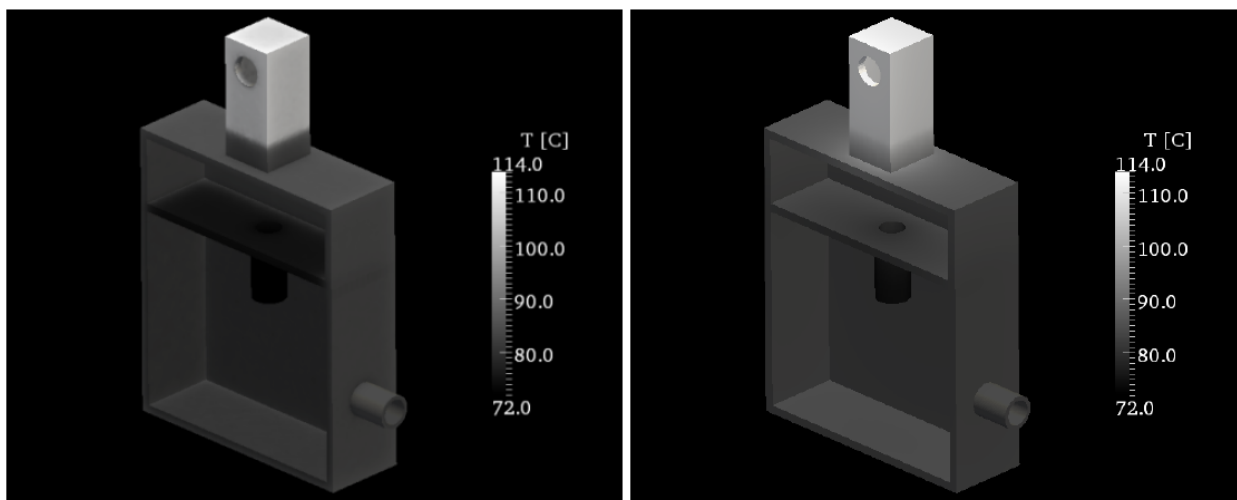


**Figure 7:** Temperature distribution in general views of suction muffler and its surrounds. Low (left) and high (right) thermal diffusivity material.



**Figure 8:** Temperature distribution in the planar views of suction muffler and the cylinder head. Lower thermal diffusivity material (upper row) and higher thermal diffusivity material (lower row).

Figure 9 shows a better perspective of both suction mufflers. In these figures it is easier to see that the outlet pipe of the muffler is at different temperature. In this zone, the heat transfer between the refrigerant and the muffler walls is intensified and it is much important than the one in the previous chambers. This results show that the discharge of the muffler is an important design consideration.



**Figure 9:** Temperature distribution in perspective views of the suction muffler. Low (left) and high (right) thermal diffusivity material.

#### 4. CONCLUSIONS

The NEST platform, using TermoFluids (Lehmkuhl 2009) software in the background, has been used to solve the heat conduction in a hermetic reciprocating compressor.

Several capabilities of this software have been tested in the simulation. First, the heat transfer has been analyzed in the whole set of compressor components by means of a partitioned approach. This involves the use of several instances of the TermoFluids DiffusionSolver, which exist simultaneously while they are executed on several parallel processes. Using distinct instances has made it possible the use of different meshes and materials. The employed meshes are based on geometric particularities of each component. Consequently, they do not match at the contact boundaries, which permitted to test the NEST data transfer tools also.

Various three-dimensional numerical results of the temperature distribution in the compressor components have been presented and discussed. The obtained numerical results are in good agreement with the expected thermal behavior of a compressor. Although this simulation is far from being a novel contribution to compressors design, it proves that the NEST platform deserves further research. This research can be focus on solving the convection of the remnant fluid refrigerant around the solid components using coupled CFD&HT. In addition, many other compressor details should be considered in this kind of simulations. Some are still far to achieve in numerical simulation, like the oil recirculation or those related with noise generation such as motor vibrations. Some others are already possible, although they make the simulations more computationally expensive. For example, thermal radiation is an important factor in the energy balance and could be introduced by resolving the RTE numerically. Some other details rely on the geometry and the employed materials. Innovative designs pursuing a general cold down of the compressor components could be investigated. This could be focused on many details. For instance, the joining point in the discharge line at the shell side is quite simplified in the simulation model. In real-world compressors this join is usually mechanically fastened using cooper rivets. This is an important consideration because it alters the heat transfer to the shell and the ambient. In sum, the advances in high performance computer and consequently the new possibilities in CFD&HT are not the definitive solution in the improvement of industrial devices. There is still much work to do in this research area and today's simulation tools must be used carefully. Well-thought numerical

experiments are necessary to provide substantial information on designs defects and suggest novel enhancements. While the one-dimensional models lack of detail in the provided results, CFD&HT models are still too computationally expensive for using them in industrial problems. The combination of both one-dimensional models -in a low level- and three-dimensional models -in a high level- is the workaround to this problem. When this is possible, the numerical results such as the ones presented in this work can be complemented with estimations of the COP and hence, with an overall compressor performance analysis.

## REFERENCES

- Jiao, X. and Heath, M.T. (2004). Common-refinement-based data transfer between nonmatching meshes in multiphysics simulations. *Int. J. for Numerical Methods in Engineering*, 61,2402-2427.
- Lehmkuhl, O., Perez-Segarra, C.D., Borrell, R., Soria, M. and A. Oliva (2009). Termofluids: A new parallel unstructured CFD code for the simulation of turbulent industrial problems on low cost PC cluster. *Lecture Notes in Computational Science and Engineering*, 67:275-282.
- López, J. (2016). *Parallel object-oriented algorithms for simulation of multiphysics. Application to thermal systems*. Universitat Politècnica de Catalunya. Thesis.
- Perez-Segarra, C.D. et al. (2004). Detailed thermodynamic characterization of hermetic reciprocating compressors. *Int. J. Refrig.*, 28,579-593.
- Rigola, J. et al. (2005). Parametric studies on hermetic reciprocating compressors. *Int. J. Refrig.*, 28(2),253-266.
- Ribas, F.A. et al. (2008). Thermal analysis of reciprocating compressors - a critical review. In *Int. Compr. Eng. Conf.*, Purdue, IN, USA, Paper 1907.
- Shiva, B. G. (1998). Heat transfer in reciprocating compressors - a review. In *Int. Compr. Eng. Conf.*, Purdue, IN, USA, Paper 1349.