2014

Predicting the Suction Gas Superheating in Reciprocating Compressors

Jonatas Ferreira Lacerda
Tecumseh Products Company, Brazil, jonatas.lacerda@tecumseh.com

Celso Kenzo Takemori
Vibroacústica Development and Research, Brazil, celso@vibroacustica.com.br

Predicting the Suction Gas Superheating in Reciprocating Compressors

Jônatas Ferreira LACERDA[1]*, Celso Kenzo TAKEMORI[2]

[1]Tecumseh Products Company, Research and Development,
São Carlos, São Paulo, Brazil
jonatas.lacerda@tecumseh.com

[2]Vibroacústica Development and Research
Joinville, Santa Catarina, Brazil
celso@vibroacustica.com.br

* Corresponding Author

ABSTRACT

A considerable portion of the cooling capacity losses on a household compressor occurs due to heating of the gas entering through the suction tube, which is called as superheating. A better understanding of the flow and heat transfer phenomena involved is crucial to improve compressor efficiency. In this work, it is presented a numerical simulation study using a commercial code to analyze a transient three-dimensional, compressible and non-isothermal flow, which has been used for modeling the problem. The computational domain considered the suction tube, a portion of the compressor cavity and the suction muffler. Prescribed transient mass flow rates obtained by simpler 0D/1D models have been used as boundary conditions while the cavity gas and wall temperatures were prescribed based on experimental methods. The results showed that the mean temperature of the gas entering into the compressor was 40°C, while the mean temperature of the gas leaving the suction muffler was 63°C, presenting 23°C of superheating.

1. INTRODUCTION

During the operation of reciprocating compressors, part of the incoming refrigerant gas through the suction line flows into the compressor cavity and contacts hot parts like the electric motor and the cylinder head (Figure 1). This cold gas refrigerates the hot parts of the compressor but, on the other hand, increases the gas temperature. In fact, the gas entering into the suction muffler is a mixture of this hot gas coming from the cavity and the cold gas coming from the suction line. This mixture causes superheating of the gas being suctioned, decreasing its density and the compressor capacity. Under standardized conditions, e.g. ASHRAE, Cecomaf, the temperature of the entering the compressor is defined as 32°C. This being the temperature at which the refrigerant reaches the compressor, the fluid still needs to go up to the cylinder inlet, experiencing a heating that can reach 60°C to 70°C (Morriesen and Deschamps, 2012). The superheating is a consequence of the internal heat generation of the compressor due: electric motor losses, bearings losses and the compression process itself. It is important to notice that the cooling capacity reduction implies in a direct reduction of the energy efficiency. Ribas *et al.* (2008) showed that almost 50% of the compressor thermodynamic losses are related to the superheating.

Thus, it is interesting to know how the heat is exchanged inside the compressor, and analyze different ways to decrease superheating effect. In this subject Hass (2013) presented an extensive review regarding studies in the area of thermal models applied to compressors. Based on that, it is possible to classify the researches on this area in two main groups: Transient models supported by integral (0D) and one-dimensional (1D) differential formulations, and steady state three-dimensional (3D) models. With the continuous increase in the computational speed, analysis of 3D transient flow models is becoming a reality in the product development environment across industries.

The aim of this work is to perform the analysis of the superheating effect of cold gas entering the compressor through suction tube until it leaves the suction muffler into the cylinder. Numerical simulations were used to perform this task, joining the benefits of a transient 0D/1D models to predict the mass flow rates involved together with the benefits of a 3D transient flow model to predict the heat transfer phenomena.
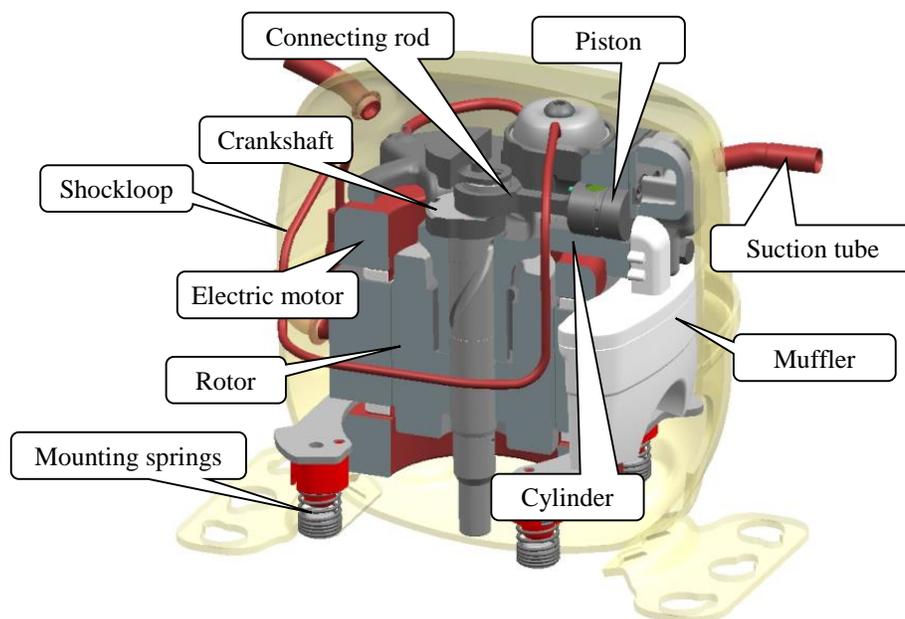

Figure 1: Typical reciprocating compressor for refrigerators. The compressor is sad to be hermetic due to the fact that all components are hermetically sealed inside a shell

2. METHODOLOGY

The methodology comprises two steps, being the first one a model with low computational cost in order to estimate the compressor mass flow rate, and the second one a transient 3D model to predict the superheating, with higher computational cost.

2.1 Overview
Initially a mixed 0D/1D model was solved to estimate the transient mass flow rates through the compressor components. These mass flows were later used as boundary conditions of a transient 3D model and wall and gas temperatures were prescribed based on experimental methods.
The methodology was employed on a compressor that operates at 60 Hz, with R134a as refrigerant gas. The suction pressure was 97 kPa and the discharge pressure was 1,072 kPa.

2.2 Predicting transient mass flow rate (0D/1D)
To predict the transient mass flow rate, a mathematical model was implemented according to the diagram presented on Figure 2. This model consists by integral formulations (0D) to model volumes, cylinder, orifices, 1D formulations (tubes) and reed valve model for flow and dynamics (Ussyk, 1984), since the inlet to the outlet of the compressor.

As boundary conditions (i) suction pressure and standardized superheating temperature were employed on 'suction tube' inlet, and (ii) discharge pressure was defined on 'discharge tube 2' outlet. Heat transfer between the fluid and walls were neglected. The driven-force was given by cylinder expansion and compression, this being modeled as a crankshaft-connecting-rod mechanism (Figure 1).

Besides the mass flow rate required to estimate the superheating temperature, this model is also able to predict other compressor performance parameters like indicated power, suction and discharge power losses, reverse flow on valves and leakages, which were not covered in this work.
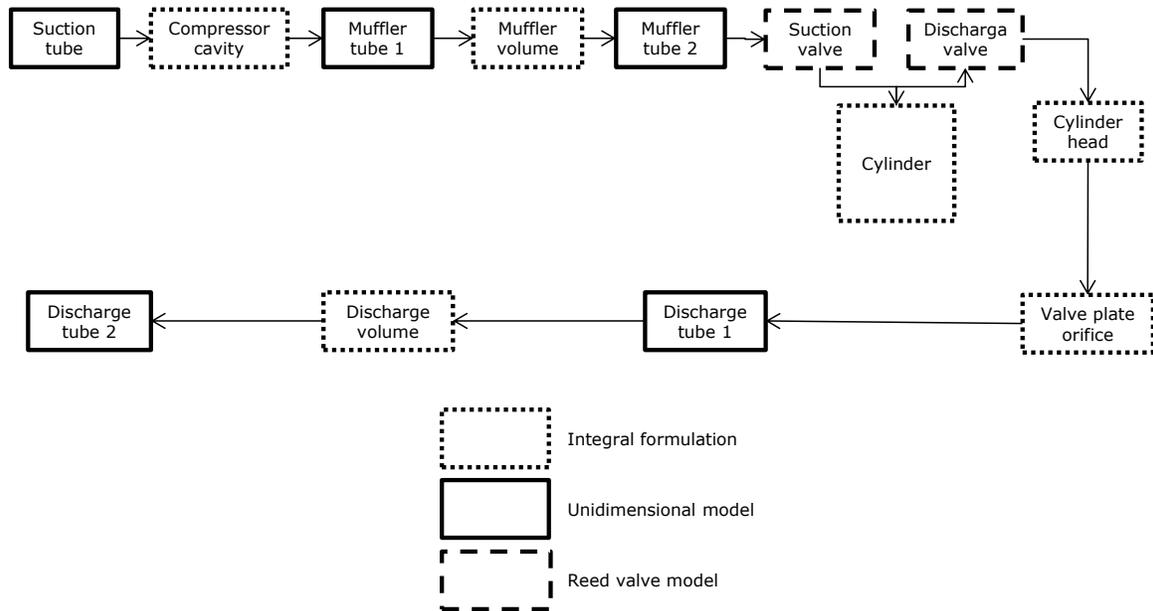
Figure 2: Block diagram of the hybrid model for predicting transient mass flow rate.

2.3 Predicting superheating (3D)

A transient 3D CFD model was used to estimate the superheating of the cold gas entering the compressor through suction tube until and leaving the suction muffler into the cylinder. To perform this task the commercial code Ansys-CFX was used. In this model, the superheating was given by the mixture of the suctioned cold gas coming from the suction tube with the hot gas of the compressor cavity and the heat exchange of the gas with hot walls. The cavity gas temperature and the wall temperatures were prescribed based on previous experimental data. In Figure 3 is presented the fluid portion detached from the compressor and used to perform the simulation of the problem. This strategy was adopted to decrease the computational cost.
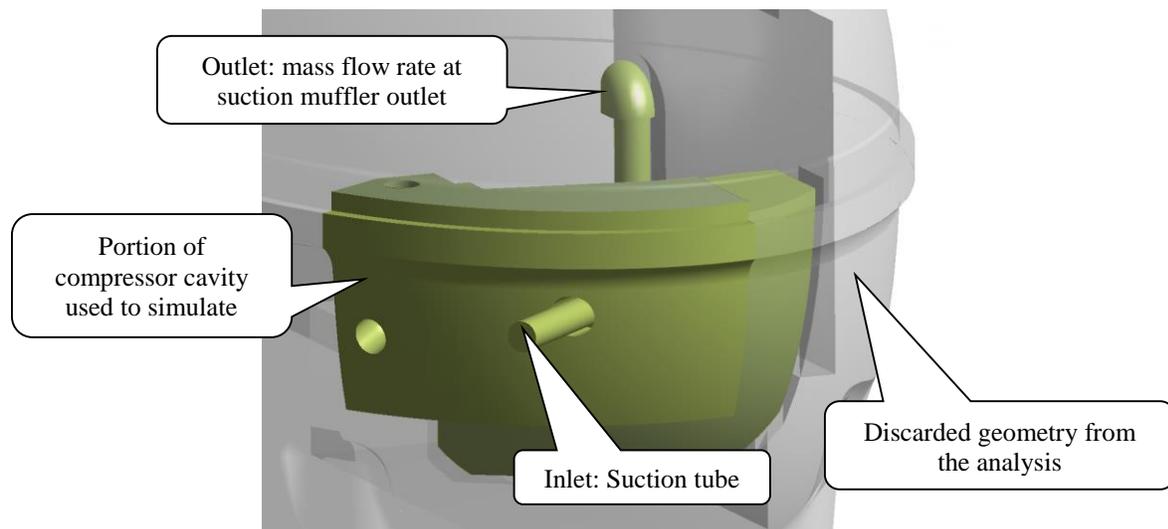

Figure 3: Geometry used for predicting the superheating of the gas leaving the suction muffler

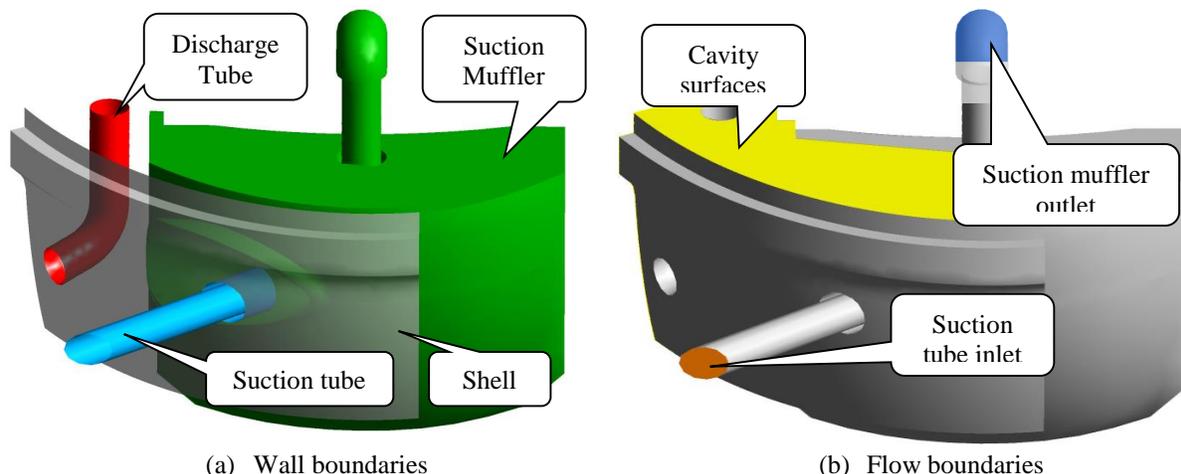(a) Wall boundaries   (b) Flow boundaries

Figure 4: Boundary conditions of the 3D analysis: (a) shell, suction tube, discharge tube and suction muffler walls and (b) suction tube inlet, suction muffler outlet and surfaces that truncates the compressor cavity

Compressible transient mass conservation and Navier-Stokes equations were numerically solved. To close the problem were also solved the equation of energy balance without simplifications and a Peng Robinson real gas model (Ansys CFX, 2012).

From the numeric point of view, 'High Resolution' and 'Second Order Backward Euler' methods were employed, respectively, for spatial and transient scheme. These methods are well suited for the oscillating nature of the flow through the compressor (Takemori, 2009). To model the turbulence was used the SST turbulence model, that falls within the category Unsteady Reynolds Averaged Navier-Stokes (URANS) equations and combines κ-ε and κ-ω models (Cezario, 2007), being the first used on turbulent free stream and the second on near-wall regions.

The boundary conditions, detailed in Figure 4, were defined as:
 (i) a constant mass flow rate and temperature, 1.03 g/s and 32°C, respectively, at the 'suction tube inlet';
 (ii) a varying mass flow rate according to Figure 6 at the 'suction muffler outlet'
 (iii) an absolute pressure of 97 kPa and a temperature of 80°C on the boundary that truncate the compressor cavity;
 (iv) a constant wall temperature at suction tube, suction muffler, discharge tube and shell walls, being 40°C, 65°C, 100°C and 90°C, respectively.

Two meshes were used with around 60 and 120 thousand nodes, respectively, as showed in Figure 5. The refinement was mainly done in the prisms layers close to the walls to enhance the heat transfer calculation precision in these regions. 10 cycles were simulated taking approximately 20 hours for the coarse mesh and 10 hours for the refined one (although in the later case results from the coarse were used as initial condition for the simulation and only 3 cycles were simulated). Parallel simulation on a 12 Gb of RAM with four 3.33GHz computer processing unites computer was used.
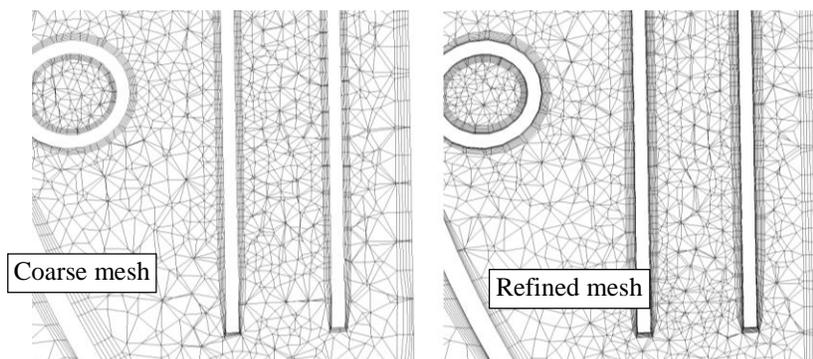

Figure 5: Comparison between the two meshes

3. RESULTS AND DISCUSSION

Figure 6 shows the mass flow rate as a function of the compressor crankshaft angle used as mass flow rate boundary condition at 'suction muffler outlet'. The mass flow rate on the 'suction tube' inlet is constant and equals to the average of the mass flow rate showed on Figure 6: 1.03 g/s.
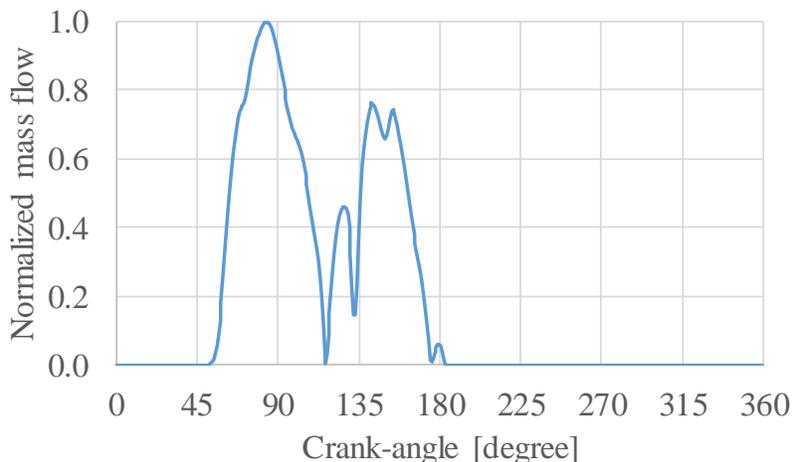

Figure 6: Normalized mass flow rate at suction muffler outlet as predicted by the hybrid 0d/1D model

The temperature behavior at several locations as function of simulated compressor cycles for the coarser mesh is showed in Figure 7. A transient comportment is observed due both the time evolution from an initial state (Figure 7a) and the transient mass flow applied as boundary conditions (Figure 7b). On the other hand, in Figure 8 is showed a time-averaged within the cycle calculation of the temperature at same locations, where is observed that a periodic steady state flow was almost achieved. In this sense, would need to be simulated more cycles to achieve a lower variation.

The results obtained with the coarser mesh were interpolated into the refined mesh to perform the calculations on the later. The comparison of the time-averaged temperature at the suction muffler outlet with both meshes is presented in Figure 9, showing around 0.4% of difference between both. It is possible to see that the gas leaving the suction muffler experiences an increase of 23°C, what is a huge amount of heating and a larger contributor to compressor inefficiency.

With the aid of CFD results, it is possible to improve the understanding of the temperature distribution in the domain. Figure 10 shows the temperature field in a plane sectioning the domain transversally. Using the last cycle results, four different crank-angles were toke: 0°, 90°, 180° and 270°. We can observe that the gas entering the suction muffler is a mixing of cold gas coming from suction tube and hot gas coming from compressor housing (Figs. 10b and 10c). After the muffler is filled, part of the cold gas coming from the suction tube fills the housing cavity (Figs. 10a and 10d).

To decrease the suction muffler outlet gas temperature and improve the compressor efficiency, it is necessary to decrease the heating of the cold gas coming from suction tube. Looking at the results it is possible to identify two heating mechanisms: heating due heat exchange by convection with hot walls and heating due mixing of cold and hot flow streams. The first mechanism can be avoided decreasing the heat generated inside the compressor and/or improving the walls thermal resistances inside the compressor, mainly in the suction muffler surroundings. The second one can be decreased by improving the suction muffler inlet shape or suction tube alignment with the muffler entrance to avoid the suction of the hot gas coming from housing. Thus, CFD technique is a good choice to evaluate several configurations due time and cost aspects.
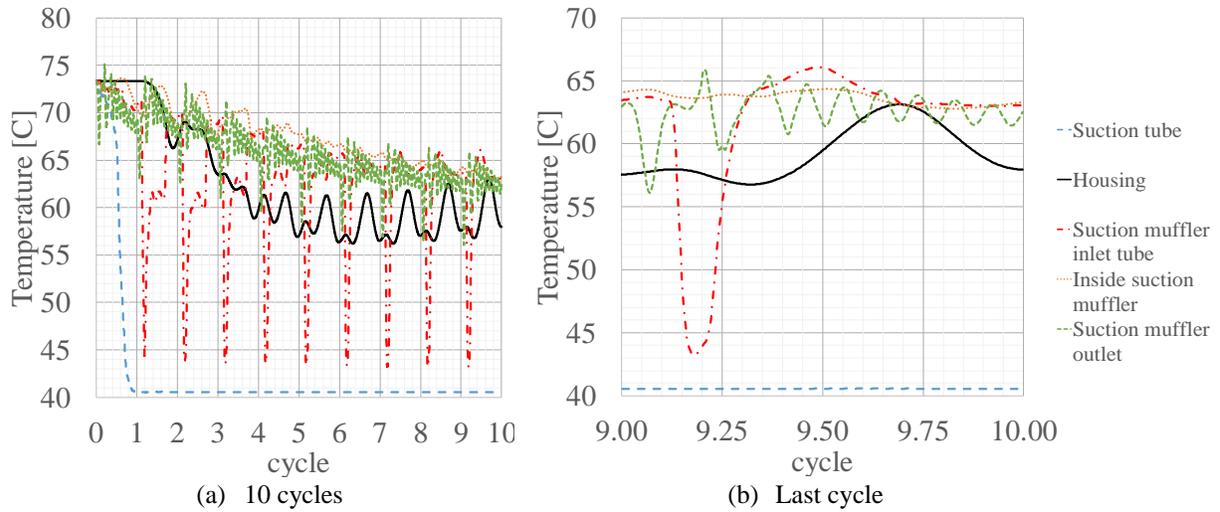
Figure 7: Temperature at several locations for (a) all 10 cycles and (b) last cycle
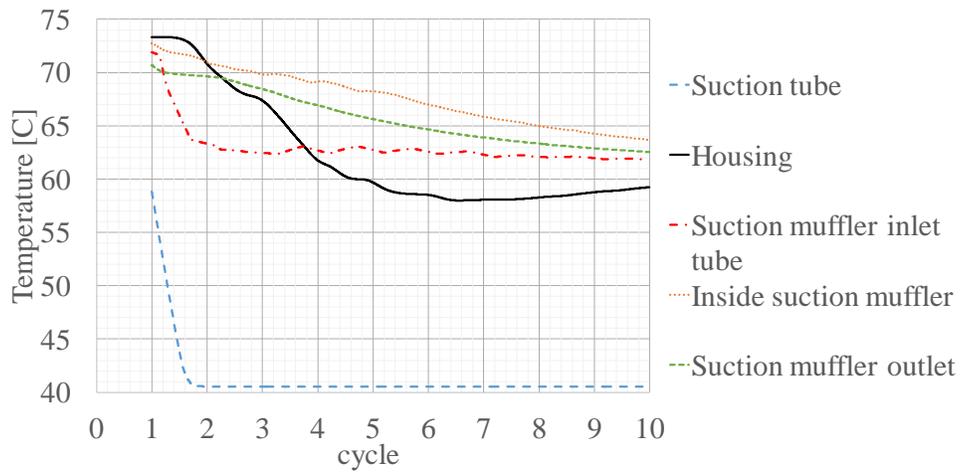

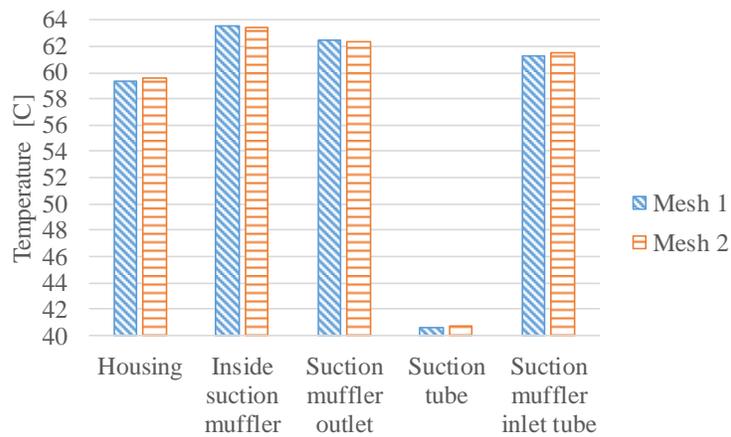**Figure 8:** Time-averaged within the cycle temperature at several locations

Figure 9: Gas temperature at several locations for both meshes
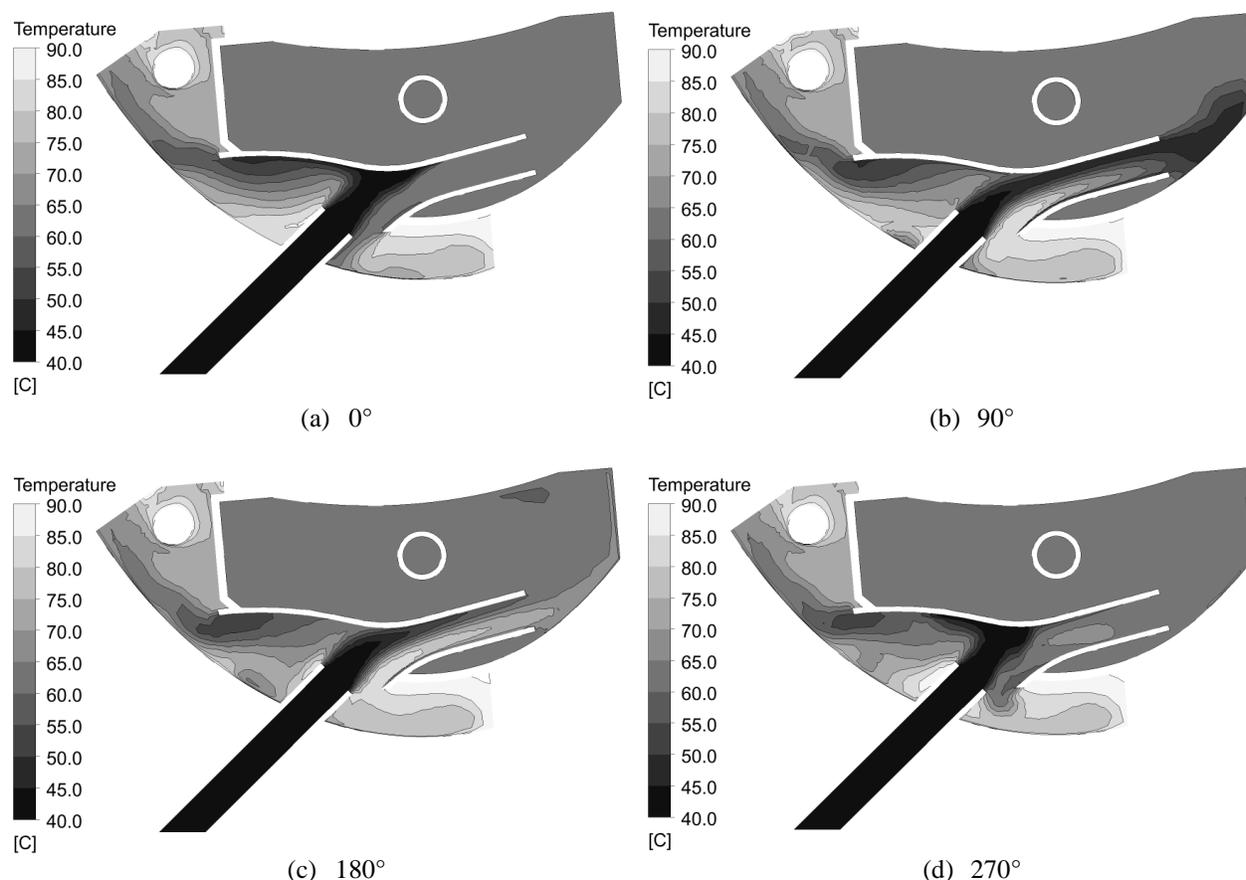
(a) 0°

(b) 90°

(c) 180°

(d) 270°

Figure 10: Temperature field in a sectioning plane at several crank-angles in the last cycle

The main weaknesses of this methodology are related with boundary conditions and simulation time to achieve a periodic steady solution. In this study, it was necessary first an experimental procedure to obtain the wall temperatures to be imposed as boundary conditions. However, the wall temperatures themselves are also function of the flow and heat exchange inside the compressor, which contributes with the walls cooling or heating. This make the assumptions of constant wall temperature a bit far from the actual behavior. Furthermore, the simulation time is still large and can increase even more if a better wall resolution of the mesh is necessary to solve the problem. Thus, the methodology applied in this work can be improved, where 3D simulations would be used to calculate heat transfer coefficients in several parts of the flow domain and then applied into a 0D/1D approach, with lower computational cost, to calculate the transient behavior and reach a periodic steady solution faster than when used 3D simulation. This stays as suggestion for future works.

4. CONCLUSIONS

In this work the superheating in a reciprocating compressor was investigated using a numerical procedure where a transient 0D/1D model was used to predict the mass flow rates involved while a 3D transient flow model was used to predict the heat transfer phenomena. It was observed an increase of 23°C from the gas coming from the suction tube until it leaves the suction muffler in direction to the cylinder. The heating of suctioned gas occurred due convection with hot walls and due the mixing with the hot gas present inside the compressor housing. Increase wall thermal resistances, decrease heat generation and improve suction muffler entrance shape to align better the flow with the suction tube are among the suggestions to decrease the superheating in this case. The method showed suitable to analyze this problem, however, the time consuming simulation and the dependence of experimental results are seen as weaknesses of this model and still open subjects. For future works is suggested the use of 3D simulations to calculate heat transfer coefficients in several parts of the flow domain and apply them into a 0D/1D approach, with lower computational cost, to calculate the transient behavior and reach a periodic steady solution faster.

REFERENCES

Morriesen A., Deschamps, C. J., 2012, Experimental investigation of transient fluid flow and superheating in the suction chamber of a refrigeration reciprocating compressor, *Applied Thermal Engineering*, vol. 41, p. 61-70.

Ribas Jr, F. A., Deschamps, C. J., Fagotti, F., Morriesen, A., Dutra, T., 2008, Thermal analysis of reciprocating compressors – A critical review, *Proceedings of the Int. Compressor Eng. Conf. at Purdue.* 1306.

Haas, D. A., Deschamps, C. J., Diniz, M. C., 2013, A Thermal Network Model to Predict the Temperature Distribution in Household Refrigeration Compressors, *Proc. 2009 IIR/IIF International Congress on Compressors*, p.1-9.

USSYK, M. S., 1984, *Numerical simulation of the performance of reciprocating compressors* (in portuguese), Msc. Thesis, Federal University of Santa Catarina, Brazil.

ANSYS CFX, 2012, CFX-Solver Modeling Guide.

TAKEMORI, C. K., 2009, Numerical simulation of the fluid-structure interaction applied to the reed valves of a miniaturized compressor under high frequency conditions, *Proc. of International Conference on Compressors and their Systems*.

CEZÁRIO, C. A., 2007, *Analysis of the air flow on electric motors* (in portuguese). Msc. Thesis, Federal University of Santa Catarina, Brazil.

ACKNOWLEDGEMENT