

2010

Numerical Analysis of Heat Transfer inside the Cylinder of Reciprocating Compressors in the Presence of Suction and Discharge Processes

Evandro L. L. Pereira
Federal University of Santa Catarina

Cesar J. Deschamps
Federal University of Santa Catarina

Fernando A. Ribas
Embraco

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

Pereira, Evandro L. L.; Deschamps, Cesar J.; and Ribas, Fernando A., "Numerical Analysis of Heat Transfer inside the Cylinder of Reciprocating Compressors in the Presence of Suction and Discharge Processes" (2010). *International Compressor Engineering Conference*. Paper 1995.
<http://docs.lib.purdue.edu/icec/1995>

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>

Numerical Analysis of Heat Transfer inside the Cylinder of Reciprocating Compressors in the Presence of Suction and Discharge Processes

Evandro L. L. PEREIRA¹, Cesar J. DESCHAMPS^{1*}, Fernando A. RIBAS Jr.²

¹POLO Research Laboratories for Emerging Technologies in Cooling and Thermophysics
Federal University of Santa Catarina
Florianopolis, SC, Brazil
deschamps@polo.ufsc.br

²EMBRACO
Rua Rui Barbosa, 1020
Joinville, SC, Brazil

* Corresponding Author

ABSTRACT

This paper presents a numerical analysis of the unsteady in-cylinder heat transfer for a simplified model of a small reciprocating compressor under actual operating conditions, with an assessment of the influence of suction and discharge processes. A two-dimensional finite volume model is used to numerically solve the associated turbulent flow field, whereas the dynamics of valves is described by a one-degree of freedom model. Numerical results are provided for the instantaneous heat transfer at different wall surfaces inside the cylinder. Additionally, predictions for heat transfer are compared with estimates of correlations available in the literature. It is found that heat transfer is strongly affected by suction and discharge processes and this aspect is not well represented by current correlations.

1. INTRODUCTION

It is well known that heat transfer inside the cylinder of reciprocating compressors affects the compression work and, therefore, the isentropic efficiency. Moreover, the temperature and pressure at the end of the discharge process determines the amount of gas that will remain in the clearance volume, which acts on the volumetric efficiency. Numerical simulation has been frequently used to assist the development of reciprocating compressors, allowing their characterization for different operating conditions. However, accuracy of predictions is affected by models chosen to describe physical phenomena, such as the instantaneous heat transfer inside the cylinder. Therefore, the choice of an appropriated correlation to model the gas-to-wall heat transfer is fundamental to predict the overall efficiency of a reciprocating compressor if a lumped modeling approach is to be adopted.

Several correlations have been proposed in the literature to evaluate heat transfer inside the cylinder of reciprocating compressors. Fagotti *et al.* (1994) made an assessment of such correlations with reference to experimental data of heat transfer for a small hermetic reciprocating compressor. According to the authors, the models of Polman (1981) and Liu and Zhou (1984) give rise to inconsistent results for some working conditions. Better agreements were verified with the correlations of Brok *et al.* (1980), Adair *et al.* (1972) and Annand (1963), with the latter being the proposal that best fitted the experimental data.

Following Kornhauser and Smith (1988), Fagotti and Prata (1998) derived a correlation capable of accounting for the phase difference between the instantaneous heat flux and the gas-wall temperature difference that occurs in reciprocating compressors. The authors applied a heat transfer correlation, proposed by Lawton (1987) for internal combustion engines, with new constants derived from numerical solution of two-dimensional flow inside a piston-

cylinder assembly without valves. The adjusted correlation was incorporated into a compressor simulation code and good agreement was observed between prediction and experimental data for temperature. However, because their model was derived for a specific compressor geometry, the model constants have to be adjusted for other designs.

Müllner and Bielmeier (2008) have numerically investigated the heat transfer process inside the cylinder of a small reciprocating compressor, by solving the unsteady turbulent flow with the k- ϵ model combined with wall functions. Effective force and flow areas were adopted to estimate the dynamics and mass flow rate associated with valves. The authors analyzed the influence of location and number of inlet and outlet ports on the heat transfer phenomenon, with results for time dependent heat fluxes and heat transfer coefficients being provided for different wall surfaces (piston, valve plate and cylinder wall).

The present paper reports a numerical analysis of in-cylinder heat transfer process, by using a finite volume methodology to solve the flow through valves and inside the cylinder. A two-dimensional formulation is adopted for the flow and, for convenience, suction and discharge processes are independently simulated. Valve dynamics is modeled by a one-degree of freedom mass-spring model. Numerical results for instantaneous heat transfer at the walls are obtained for actual operating conditions and compared with correlations proposed in the literature.

This paper reports a study of the in-cylinder heat transfer process, based on a two-dimensional model to solve the compressible turbulent flow through valves and inside the cylinder. Numerical results for instantaneous heat transfer and convective heat transfer at the walls are obtained for actual operating conditions and compared with correlations proposed in the literature.

2. NUMERICAL MODEL

The concept of ensemble averaging was adopted to solve the compressible turbulent flow in the compressor, with a computed variable representing an ensemble average over many engine cycles at a specified spatial location. The RNG k- ϵ turbulence model was extensively used and validated for flow through simple geometries of compressor valves (Salinas-Casanova *et al.*, 1999) and, hence, was adopted in all simulations. An equation of state for an ideal gas completes the system required to solve the compressible flow.

In this work, wall functions were used to bridge the viscosity-affected region between the wall and the fully-turbulent region. Based on Launder and Spalding (1974), wall functions comprise laws-of-the-wall for the mean velocity and temperature (or other scalars).

The law-of-the-wall leads to following expression for the mean dimensionless velocity U^* :

$$U^* \equiv \frac{U_P C_\mu^{1/4} k_P^{1/2}}{\tau_w / \rho} = \frac{1}{\kappa} \ln E y^* \quad (1)$$

where U_P and k_P are the mean velocity and the turbulence kinetic energy at the near-wall node P , E is an empirical constant (= 9.793) and κ is the von Kármán constant (= 0.4187). For the RNG k- ϵ turbulence model the constant C_μ is equal to 0.0845. The dimensionless distance from the wall, y^* , is given by:

$$y^* \equiv \frac{\rho y_P C_\mu^{1/4} k_P^{1/2}}{\mu} \quad (2)$$

It should be noted that y^* is approximately equivalent to y^+ ($=\rho u_\tau y / \mu$) in equilibrium turbulent boundary layers.

The dimensionless temperature T^* is defined as follows:

$$T^* \equiv \frac{(T_w - T_P) \rho c_P C_\mu^{1/4} k_P^{1/2}}{\dot{q}} \quad (3)$$

The law-of-the-wall adopted for T^* has the following form:

$$T^* = \begin{cases} Pr y^* + \frac{1}{2} \rho Pr \frac{C_\mu^{1/4} k_P^{1/2}}{\dot{q}} U_P^2 & \text{if } y^* < y_T^* \\ Pr_t \left[\frac{1}{\kappa} \ln E y^* + P \right] + \frac{1}{2} \rho \frac{C_\mu^{1/4} k_P^{1/2}}{\dot{q}} \left[Pr_t U_P^2 + (Pr - Pr_t) U_c^2 \right] & \text{otherwise} \end{cases} \quad (4)$$

where U_c is the mean velocity magnitude at $y^* = y_T^*$. The non-dimensional thermal sublayer thickness, y_T^* , corresponds to the position in which the linear and logarithmic laws intersect each other. Moreover, the parameter P is defined by the following expression (Jayatilleke, 1969):

$$P = 9.24 \left[\left(\frac{Pr}{Pr_t} \right)^{3/4} - 1 \right] \left[1 + 0.28 e^{-0.007 Pr / Pr_t} \right] \quad (5)$$

The reed was considered to be parallel to the valve seat, with its dynamics being represented by a one-degree-of-freedom model:

$$m_{eq} \ddot{x} + c \dot{x} + k x = F_p + F_o \quad (6)$$

where m_{eq} , c and k are the reed equivalent mass, damping coefficient and stiffness, respectively. On the other hand, F_p is the force induced by flow on the reed and F_o can represent any other force, such as reed pre-tension. Finally, x , \dot{x} and \ddot{x} are the instantaneous reed lift, velocity and acceleration, respectively. The differential equation for the valve dynamics, Eq. (4), was solved using an explicit Euler method.

The computational model was developed with a commercial CFD code (ANSYS FLUENT, 2010) based on the finite volume method. Due to the presence of moving surfaces such as the piston inside the cylinder and suction and discharge valves, a moving grid strategy had to be applied to simulate the compression process. A simplified axisymmetric geometry was chosen for the analysis and, therefore, valves were centrally positioned in the compression chamber. For convenience, suction and discharge processes were independently simulated. Figure 1 shows a schematic view of the solution domain for simulation of the suction process (a) and discharge process (b).

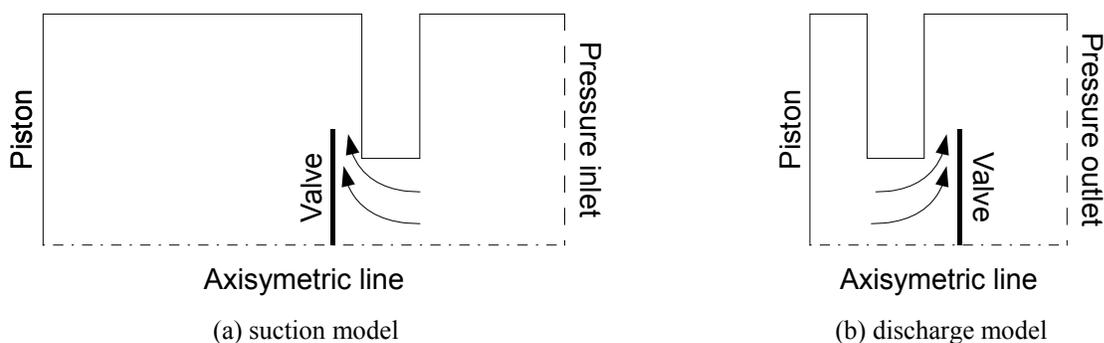


Figure 1: Schematic representation of the solution domains for suction and discharge processes.

A second-order upwind scheme was adopted to interpolate the flow quantities, except for the turbulent quantities which were interpolated with a first-order upwind scheme for numerical stability reasons. Grid and time refinement analyses were performed to verify truncation errors and guarantee adequate solution accuracy. Due to the moving mesh strategy available in the CFD code, the time step was limited to avoid that any moving surface shifted more than its adjacent cell height along a certain time interval. In order to ensure numerical stability of the iterative solution procedure and also because of constraints associated with the deforming mesh, the time step was set to a

value corresponding to a crankshaft angle of 0.2 degree. Because of the wall functions adopted in the simulations, the grid refinement next to solid walls was adjusted to guarantee the adjacent volume was located in the flow logarithmic region ($30 < y^+ < 300$). A structured grid arrangement was used, containing about 10,000 cells, with dimensions between 0.1 mm (discharge) or 0.2 mm (suction) next to the walls and 0.5 mm in the central flow regions. Fig. 2 shows the solution domain and the grid refinement adopted to simulate the discharge process.

In the discharge simulation the simulation starts with the piston at the bottom dead center (0 degrees). The initial pressure and temperature of the gas inside the cylinder are estimated according to the compressor operating condition. On the other hand, when the suction process is considered, the simulation starts with the piston at the top dead center (180 degrees) and the initial temperature and pressure values inside the cylinder are taken from results previously obtained for the discharge process. Evaporation and condensation pressures were imposed as boundary conditions for the suction and discharge chambers, respectively. In both models the cylinder wall temperature is kept constant and uniform throughout the simulation. All velocity components at the walls were set to zero, except for the reed and piston surfaces in which the velocities were obtained from Eq. (4) and the crankshaft mechanism, respectively. A turbulence intensity of 5% and a turbulence length scale proportional to the hydraulic diameter were adopted as the inlet boundary condition for the turbulent flow in the suction and discharge chambers. Averaged quantities of pressure, temperature, mass flow rate and heat flux were monitored during each simulation.

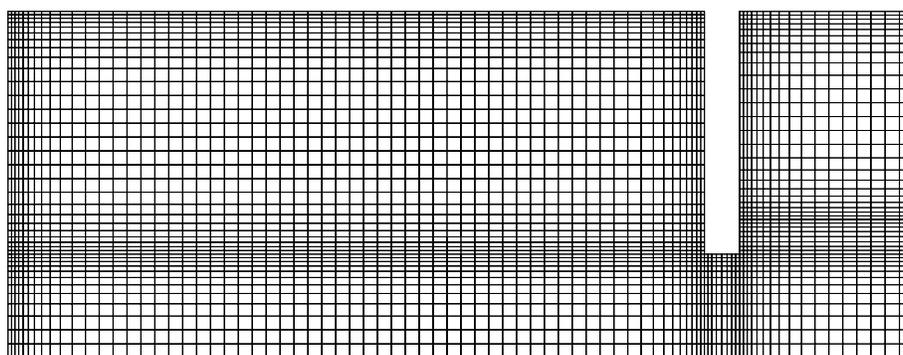


Figure 2: Grid adopted for solution domain of the discharge process.

3. RESULTS AND DISCUSSION

The simulations were carried out for a 50Hz reciprocating compressor, operating with R134a under a LBP condition ($T_{\text{evap}} = -23.3^{\circ}\text{C}$; $T_{\text{cond}} = 54.4^{\circ}\text{C}$). Accordingly, boundary conditions adopted in the simulation can be summarized as follows: inlet pressure and temperature (115kPa, 330K); outlet pressure and temperature (1470kPa, 425K). The geometric dimensions adopted in the model (Table 1) are similar to those found in small reciprocating compressors for domestic refrigeration.

Table 1: Characteristic dimensions of the compressor geometry

Piston bore	20 mm	Discharge port diameter	6 mm
Piston stroke	20 mm	Discharge valve diameter	8 mm
Connecting road length	45 mm	Suction port diameter	7 mm
		Suction valve diameter	9 mm

Results for instantaneous heat flux as a function of the crank angle were obtained for different wall surfaces of the compression chamber (cylinder, piston and valve plate), considering the discharge process (Fig. 3) and suction process (Fig. 4). The discharge and suction processes are situated in the intervals 149° - 190° and 235° - 375° , respectively, as identified by the vertical dashed lines. As one can see, heat fluxes during the discharge process are much higher than those in the suction process. This occurs because the piston is very close to the valve plate in the discharge process and, as a consequence, high levels of velocity prevail in the flow towards the discharge port along the small clearance left. The gas velocity in the clearance can have two orders of magnitude. For the same reason,

heat flux at the cylinder wall is much lower than those at the surfaces of the piston and valve plate (Fig. 3). The continuous lines in Figs. 3 and 4 represent the total heat flux in the compressor chamber, which takes into account contributions of all surfaces involved (cylinder, piston and valve plate). During the discharge process, the area of the cylinder surface is much smaller than the areas of the piston and the valve plate and, hence, its contribution on the total heat flux is negligible.

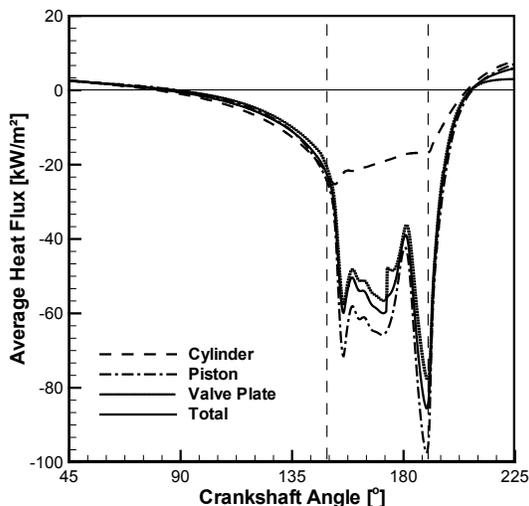


Figure 3: Heat flux at different surfaces inside the cylinder during the discharge process.

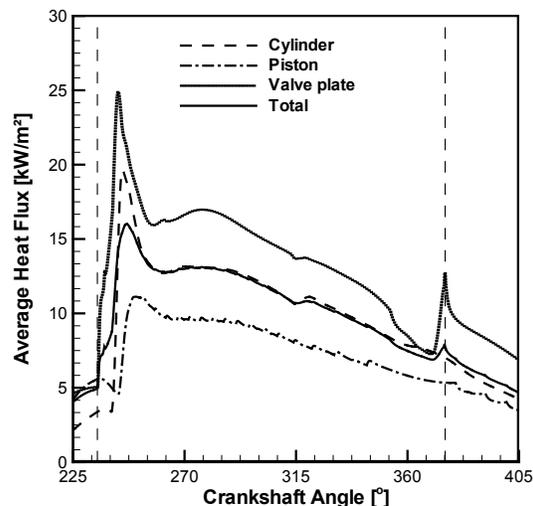


Figure 4: Heat flux at different surfaces inside the cylinder during the suction process.

During the suction process, the gas is initially deflected by the valve reed, flowing along the valve plate and then changing direction again when it reaches the cylinder wall, as shown in Fig. 5. As expected, the maximum heat flux occurs immediately after the opening of the suction valve, because the flow velocity is very high just after entering the cylinder. On the other hand, during the suction process the piston is much farther from the valve plate than it is in the discharge process. Therefore, when the gas eventually reaches the piston surface, its velocity is lower and its temperature higher, which brings about a decrease in the heat transfer intensity (Fig. 4). After the peak of heat flux in the suction process, heat transfer steadily decreases in all surfaces in response to a decrease of velocity levels in the flow through the valve. Again, when the valve is closing the suction port there is a very brief period in which flow accelerates near the valve plate, with a corresponding increase in the heat flux there.

The contribution of the velocity flow field on the heat transfer at each surface of the compression chamber was analyzed for the suction process. To this extent, new simulations were performed with all surfaces thermally insulated, with the exception of the surface of interest. Figures 6 and 7 show results for instantaneous heat flux and convective heat transfer coefficient, respectively. Overall, the results for heat flux shown in Figs. 4 and 6 are very similar, with heat fluxes at the valve plate, cylinder and piston being on average 5%, 15% and 20% higher for the present situation. Therefore, considering the piston surface, a reduction of 20% in the heat transfer happens because the gas is heated at the surfaces of the valve plate and cylinder as it flows towards the piston. Convective heat transfer coefficients shown in Fig. 7 were calculated based on the bulk gas temperature and wall temperature. As expected, such coefficients vary with the crank angle in a similar manner observed for heat flux. Furthermore, the heat transfer coefficients during the suction process are within the range 150-1100W/m²K, which are typical values for forced heat convection in gases.

The results obtained in this study for heat transfer inside the cylinder were compared to those provided by correlations available in the literature. Most of such correlations were developed for internal combustion engines. Annand (1963) proposed a correlation for the Nusselt number solely based on the Reynolds number ($Nu = ARe^b$, where "A" and "b" were experimentally determined as equal to 0.7). Annand (1963) also considered the use of the Peclet number ($Pe = RePr$) instead of the Reynolds number in his correlation and, therefore, this second proposal is also considered here for comparison purposes. Adair *et al.* (1972) proposed a correlation for compressors ($Nu =$

$0.053Re^{0.8}Pr^{0.6}$), whereas Hamilton (1974) made use of the Dittus-Boelter correlation widely used for turbulent heat transfer in ducts ($Nu = 0.0245Re^{0.8}Pr^{0.6}$).

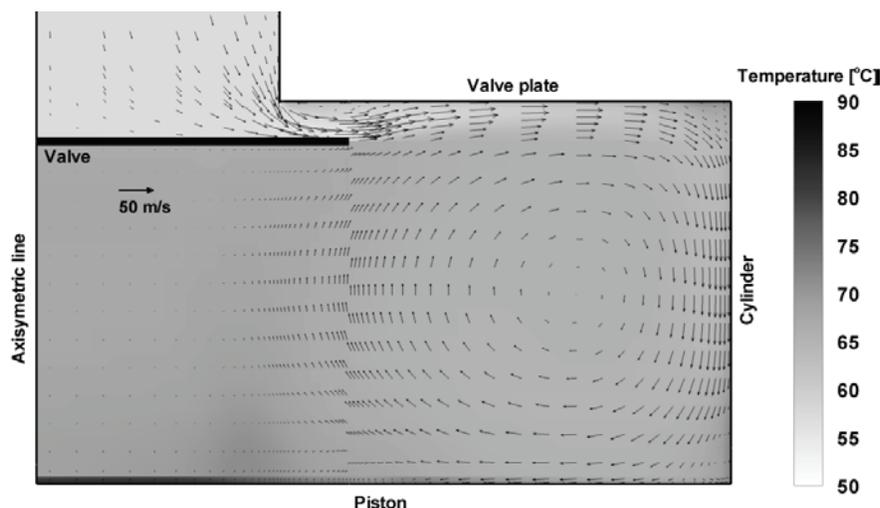


Figure 5: Contours of temperature and velocity vectors during the suction process at $\omega t = 270^\circ$.

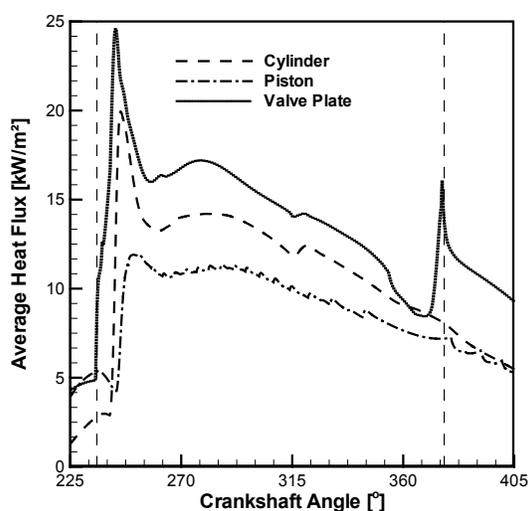


Figure 6: Flow field contribution on heat transfer in each surface during the suction process

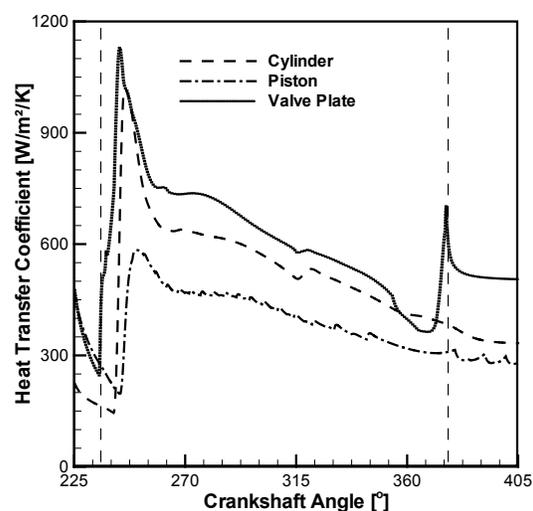


Figure 7: Heat transfer coefficients due to the flow field during the suction process

Figures 8 and 9 show a comparison between values of total heat flux inside the cylinder given by the aforementioned correlations and the present predictions, for the discharge and suction processes, respectively. During the discharge process both correlations proposed by Annand (1963) indicate the highest values of heat flux (Fig. 8), whereas the proposals of Adair *et al.* (1972) and Hamilton (1974) return the lowest estimates. The numerical model predicts intermediate levels. In the suction process, the correlations of Adair *et al.* (1972) and Hamilton (1974) predict again the lowest values of heat flux (Fig. 9). On the other hand, although the level of agreement between the estimates of Annand (1963) and the present model is considerably better when compared with the discharge process, the numerical results show higher heat flux during the period in which the suction valve is open. Part of the differences observed between predictions and estimates of Annand (1963) can be associated with the type of valves in the present model (reed valve) and in the IC engine (poppet valve) adopted by Annand (1963) in his experiments. Naturally, each type of valve gives rise to a distinct flow pattern inside the cylinder, especially in the early stages of valve opening and closing, which affects the heat transfer process.

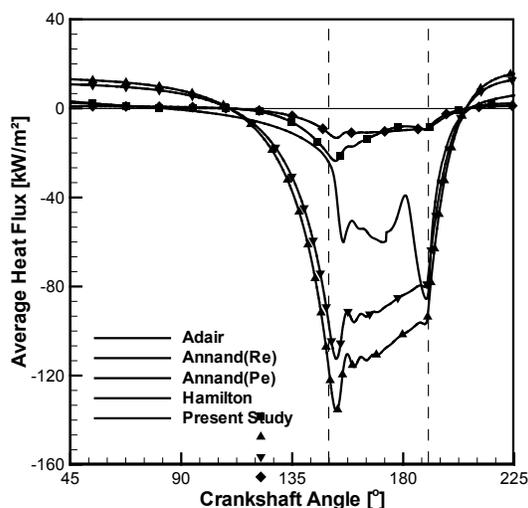


Figure 8: Average heat fluxes predicted during the discharge process

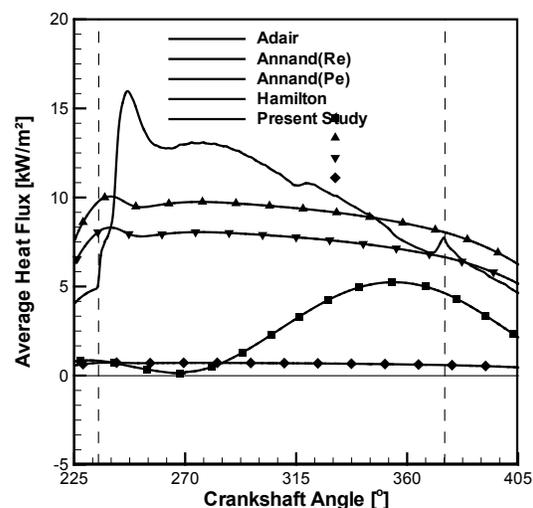


Figure 9: Average heat fluxes predicted during the suction process

It should be noticed that an accurate prediction of heat transfer during the suction process is particularly important for accurate estimates of gas superheating and, as consequence, volumetric and isentropic efficiencies. A number of aspects should be included in future investigations, such the variation of the turbulent Prandtl number along the compression process and in the proximity of walls. Another possible topic could consider the application of a low-Reynolds number turbulence model to investigate the influence of wall-functions, despite the associated higher computational due to the much finer meshes required near the walls to reach the viscous sub-layer.

4. CONCLUSIONS

A numerical analysis has been presented for the heat transfer inside the cylinder of a simplified geometry of reciprocating compressor under actual operating conditions. Special attention was given to suction and discharge valves and it was made clear their influence on the flow pattern inside the cylinder and, consequently, on the heat transfer process. Predictions for heat flux at the cylinder walls were compared to estimates returned by correlations available in the literature and a considerable disagreement was found, mainly for the discharge process. Yet, at least for two of such correlations, some agreement between results for heat flux was observed during the suction process.

NOMENCLATURE

c	reed damping coefficient	(kg/s)	U_c	mean velocity at $y^* = y_T^*$	(-)
C_μ	RNG k- ϵ model constant (=0.0845)	(-)	U_P	mean velocity at node P	(m/s)
E	empirical constant (= 9.793)	(-)	U^*	dimensionless velocity	(-)
F_o	other forces	(N)	u_τ	friction velocity	(m/s)
F_p	pressure force	(N)	x	valve displacement	(m)
k	reed stiffness	(N/m)	\dot{x}	valve velocity	(m/s)
k_P	turbulent kinetic energy at node P	(m ² /s ²)	\ddot{x}	valve acceleration	(m/s ²)
m_{eq}	reed equivalent mass	(kg)	y^*	dimensionless wall distance	(-)
Nu	Nusselt number	(-)	y^+	dimensionless wall distance	(-)
Pe	Peclet number	(-)	y_T^*	thermal sub-layer thickness	(-)
Pr	Prandtl number	(-)	κ	von Kármán constant (= 0.4187)	(-)
Pr_t	Turbulent Prandtl number	(-)	μ	molecular viscosity	(Pa.s)
Re	Reynolds number	(-)	ρ	specific mass	(kg/m ³)
T_c	condensation temperature	(°C)	τ_w	wall-shear stress	(Pa)
T_e	evaporation temperature	(°C)			

REFERENCES

- Adair, R.P., Qvale, E.B. and Pearson, J.T., 1972, Instantaneous Heat Transfer to the Cylinder Wall on Reciprocating Compressors, Proc. Purdue Compressor Technology Conference, West Lafayette, USA, pp. 521-526.
- Annand, W.J.D., 1963, Heat Transfer in the Cylinders of Reciprocating Internal Combustion Engines, Proc. Instn. Mech. Engrs., 117, pp. 973-996.
- ANSYS Inc., 2010, ANSYS FLUENT V.12.1.4.
- Brok, S.W., Touber, S. and van der Meer, J.S., 1980, Modeling of Cylinder Heat Transfer - Large Effort, Little Effect?, Proc. Purdue Compressor Technology Conference, West Lafayette, USA, pp. 43-50.
- Fagotti, F. and Prata, A.T., 1998, A New Correlation for Instantaneous Heat Transfer between Gas and Cylinder in Reciprocating Compressors, Proc. Purdue Comp. Tech. Conf., West Lafayette, USA, pp. 871-876.
- Fagotti, F., Todescat, M.L., Ferreira, R.T.S., Prata, A.T., 1994, Heat transfer modeling in a reciprocating compressor, Proc. Purdue Compressor Technology Conference, West Lafayette, USA, pp. 605-610.
- Jayatillaka, C., 1969, The Influence of Prandtl Number and Surface Roughness on the Resistance of the Laminar Sublayer to Momentum and Heat Transfer, Prog. Heat Mass Transfer, 1:193-321.
- Kays, W.M., 1994, Turbulent Prandtl Numbers - Where are We? J. Heat Transfer, Trans. ASME, 116, 284.
- Kornhauser, A.A., Smith Jr., J.L., 1988, Application of a Complex Nusselt Number to Heat Transfer during Compression and Expansion, On Flows in Internal Combustion Engines IV, ASME FED Vol. 65, pp. 1-8.
- Lawton, B., 1987, Effect of Compression and Expansion on Instantaneous Heat Transfer in Reciprocating Internal Combustion Engines, Proc. Instn. Mech. Engrs., 201, A3, pp. 175-186.
- Lauder, B. E., Spalding, D. B., 1974, The Numerical Computation of Turbulent Flows, Computer Methods in Applied Mechanics and Engineering, 3:269-289.
- Liu, R. and Zhou, Z., 1984, Heat Transfer Between Gas and Cylinder Wall of Refrigeration Reciprocating Compressor, Proc. Purdue Compressor Technology Conference, West Lafayette, USA, pp. 110-115.
- Müllner, T., Bielmeier, O., 2008, Heat Transfer in Small Piston Compressors, Proc. Purdue Compressor Technology Conference, West Lafayette, USA, 8 p.
- Polman, J., 1981, Heat Transfer in a Piston-Cylinder System, Int. J. Heat Mass Transfer, 24, pp. 184-187.
- Salinas-Casanova, D.A., Deschamps, C.J. and Prata, A.T., 1999, Turbulent Flow through a Valve with Inclined Reeds, Proc. Inter. Conf. on Compressors and Their Systems, London, pp. 443-452.

ACKNOWLEDGEMENT

This study was carried out at the POLO facilities under National Grant No. 573581/2008-8 (National Institute of Science and Technology in Refrigeration and Thermophysics) funded by CNPq (Brazilian Research Council). Support from EMBRACO and FINEP (Federal Agency of Research and Projects Financing) are also gratefully acknowledged.