Determination of the oil distribution in a hermetic compressor using numerical simulation

S Posch1, J Hopfgartner1, E Berger1, B Zuber1, R Almbauer1 and P Schöllauf2

1 Graz University of Technology, 8010 Graz, Austria
2 Secop Austria GmbH, 8280 Fürstenfeld, Austria
posch@ivt.tugraz.at

Abstract: In addition to the reduction of friction the oil in a hermetic compressor is very important for the transfer of heat from hot parts to the compressor shell. The simulation of the oil distribution in a hermetic reciprocating compressor for refrigeration application is shown in the present work. Using the commercial Computational Fluid Dynamics (CFD) software ANSYS Fluent, the oil flow inside the compressor shell from the oil pump outlet to the oil sump is calculated. A comprehensive overview of the used models and the boundary conditions is given. After reaching steady-state conditions the oil covered surfaces are analysed concerning heat transfer coefficients. The gained heat transfer coefficients are used as input parameters for a thermal model of a hermetic compressor. An increase in accuracy of the thermal model with the simulated heat transfer coefficients compared to values from literature is shown by model validation with experimental data.

Nomenclature
Roman symbols
- $c_p$: specific heat capacity
- $f_b$: body force vector
- $p$: pressure
- $q_V$: rate of heat source/sink per volume
- $t$: time
- $T$: temperature
- $v$: velocity vector

Greek symbols
- $\lambda$: thermal conductivity
- $\mu$: dynamic viscosity
- $\rho$: density
- $\phi$: viscous dissipation term

1. Introduction
The use of simulation in different fields of engineering has been increased in the last two decades due to the new possibilities of applying numerical methods in combination with raising computational performance. Additionally to the early pioneers of using simulation in the development process like the automotive or aeronautic industry the decreasing costs allow the use of simulation tools in other fields of mechanical engineering. As an example, the traditionally experimentally driven development of hermetic reciprocating compressors for refrigeration appliances can be mentioned. To meet future regulations concerning the energy consumption of refrigeration devices comprehensive research on the level of the overall cooling cycle and on the component level is needed. Especially for the hermetic compressor which serves as the core element of a refrigeration cycle, considerable efforts have to be
made to find energy saving potentials. The development of innovative compressors requires the knowledge of its thermal behaviour and occurring loss mechanisms. Due to the hermetic shell an individual consideration of the electrical, mechanical and thermodynamic losses is not expedient. Electrical and mechanical losses produce heat which influences the thermodynamic behaviour of the entire compressor. Several studies have been published and can be found in the open literature which deal with the thermal modelling of hermetic reciprocating compressors. The following survey of thermal compressor models should give an overview without any claim of completeness.

Generally, thermal compressor models can be characterized due to their complexity and geometrical resolution. Studies presented in [1] or [2] split up the compressor in several lumped volumes representing the appropriate compressor part. Using the first law of thermodynamics the temperature field is calculated. Heat transfer coefficients are determined either by simple correlations or experimental data. These studies represent the early stage of thermal compressor modelling and show the limitations due to low computational performance in the late 80s and early 90s. The models are only able to give a rough estimation of the temperature field.

The Thermal Network approach (TNW) can be considered as more flexible compared to the previous described models. Examples of TNW for thermal compressor modelling can be found in [3] and [4]. Using the Lumped Conductance Method the heat transfer between mass points representing the several compressor parts is modelled. The authors calculate the convective heat transfer with correlations for forced or natural convection based on the Nusselt number which can cause uncertainties especially in regions with transient flow like in the suction or discharge muffler. Low geometric resolution is also critical for the validation of the models using experimental data due to significant temperature differences in parts with a low thermal conductivity.

Modern thermal models integrate Computational Fluid Dynamics (CFD) resulting in so called hybrid models. Hybrid models combine 3d numerical simulation (e.g. solid part conduction) and simple correlation (e.g. convective heat transfer) in one model to gain high geometrical resolution by keeping the calculation time at acceptable limits. Examples for hybrid models using 3d heat conduction of the solid parts in combination with lumped volume formulation for the compressor gas path can be found in [5], [6] and [7]. The modelling of the heat transfer in these models was carried out either by using experimental data or correlations from the literature. In [8] special care was laid on the cylinder of the compressor. The authors combined 1d flow simulation of the compressor gas line, 3d formulation of the compressor cylinder and lumped formulation of the remaining compressor parts.

A more detailed hybrid model using 3d formulation of the full gas line and 3d formulation of the solid components can be found in [9]. The authors introduced a simulation algorithm to combine transient flow calculation with steady-state heat conduction calculation. Heat transfer coefficients calculation between the oil and the oil covered walls is carried out with correlations from literature. The authors remark that the results of thermal compressor models should not be trusted without considering the modelling of the heat transfer coefficients especially between oil and oil covered walls. Generally, considerable influence on the thermodynamic compressor behaviour is caused by the lubrication oil. Additionally to the lubrication of the moving parts and reduction of wear losses, the oil acts as heat transfer media between the hot compressor parts, the shell and hence the ambience. Based on the thermal compressor model presented in [9] the present study increases the accuracy of the model by calculating the heat transfer coefficients between oil and compressor shell applying 3d CFD. Therefore, a simulation of the oil distribution inside the compressor shell is performed in the commercial software ANSYS Fluent. The two phase flow (oil/refrigerant) is modelled with the Volume of Fluid (VOF) approach. A comparison between the results of the thermal model with heat transfer gained with correlations from the literature and by CFD, respectively, is carried out and the results are validated with experimental data of a calorimeter test bench.

The compressor used in the present study works with R600a (isobutane) and has a displacement of 5.5 ccm at a rotational speed of 2950 rpm. Its COP at ASHRAE test conditions (-23.3 °C/55 °C) is approximately 1.8.
2. Modelling
An overview of the used CFD methods and solution strategy is given in the following chapter. The fluid flow of the distributed oil is assumed to be a continuous jet which can also be seen in experimental tests using a transparent compressor shell and a stroboscope light, therefore the volume of fluid approach (VOF) is used for fluid flow simulation.

2.1. Theoretical background
The basic idea of CFD is the transformation of governing equations from the partial differential equation form into an algebraic form using finite volume discretization. For the present study the governing equations for mass, momentum and energy are essential [10]:

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{v}) = 0
\]  (1)

\[
\frac{\partial}{\partial t} [\rho \mathbf{v}] + \nabla \cdot (\rho \mathbf{v} \mathbf{v}) = -\nabla p + \mu \nabla^2 \mathbf{v} + \mathbf{f}_b
\]  (2)

\[
\frac{\partial}{\partial t} [\rho c_p T] + \nabla \cdot (\rho c_p \mathbf{v} T) = \nabla \cdot (\lambda \nabla T) + \frac{Dp}{Dt} + \mu \Phi + \dot{q}_v
\]  (3)

Commercial software packages like ANSYS Fluent discretize the governing equations automatically on the regarded flow domain and solves the system of equations by means of appropriate numerical methods. The modelling of the oil flow in the gaseous environment inside the compressor shell, the previous mentioned VOF approach presented in [11] is used thus an additional value, the volume fraction \( \chi \) is introduced. It describes the volumetric amount of oil in a finite volume of the solution domain. The use of \( \chi \) requires the solution of an additional governing equation and the volume fraction is used to determine density and viscosity of a partial filled finite volume by applying the following equations:

\[
\frac{\partial \chi}{\partial t} + \nabla \cdot [\chi \mathbf{v}] = 0
\]  (4)

\[
\rho = \rho_g \chi + \rho_l (1 - \chi)
\]  (5)

\[
\mu = \mu_g \chi + \mu_l (1 - \chi)
\]  (6)

For more details about the VOF approach the interested reader is referred to [11] and [12].

2.2. Solution domain and computational settings
The gas volume inside the compressor shell is meshed for the present simulation with approximately 700,000 mainly tetrahedral cells with hexahedral cells adjacent to the shell wall. To keep the number of cells in reasonable boarders regions which are covered by compressor parts and thus no oil is transported are left out. Moving mesh techniques like layering are taken to model the piston and crankpin movement to consider the induced secondary movement of the gas in the shell which influences the oil jet.

Oil mass flow rate at the oil pump outlet is determined by simulating the lower pump geometry according to [13] assuming constant oil mass flow rate in the rest of the pump and thus at the pump outlet. The properties of the oil are assumed to be constant over the entire simulation. Values for density and viscosity are set to 832 kg m\(^{-3}\) and 6.7 \times 10\(^{-3}\) Pa s, respectively. Surface tension is also set constant at 0.028 N m\(^{-1}\). Additionally, the following assumptions are made for the simulation of the oil distribution:

- Wall temperatures are constant.
- Flow is isothermal.
• Laminar flow.
• Dissolving of gaseous phase in liquid phase is neglected.

The oil distribution simulation is carried out with the following solver settings:
• free surface model: implicit VOF
• time discretization: bounded second order implicit, time step $10^{-4}$ s
• spatial discretization: second order upwind for momentum [10], compressive scheme for volume fraction [12]
• pressure-velocity coupling: PISO [14]
• double precision mode

3. Results
The presentation of the results is divided into the individual work steps until the oil distribution results can be used in the thermal compressor model.

3.1. Oil flow distribution
The simulation of the oil flow distribution is carried out until steady-state conditions are reached. In the present work steady-state means that the difference in the absolute oil mass flow rate into the compressor oil sump between two time steps becomes approximately constant which is obtained after 32 crankshaft revolutions. Illustrations of the simulation results with the starting oil flow can be seen in Figure 1.

![Simulation results](image)
0.0244 s

**Figure 1.** Oil distribution inside the compressor shell at several time steps.

The oil film thickness as a result of the oil distribution simulation can be seen in Figure 2 (only a characteristic area is shown to enable a proper overview). Figure 2a shows the iso-surface of the oil volume fraction and Figure 2b the oil film thickness value. The oil film thickness is not constant along the fall film flow direction which can be explained by the periodic impingement of the oil. By the use of an user defined function (UDF) the mean oil film thickness of 1.4 mm was calculated.

![Image](attachment://oil_distribution.png)

**Figure 2.** Oil film (a) and oil film thickness (b).

### 3.2. Heat transfer coefficient

The simulation of the oil distribution inside the compressor and furthermore on the compressor shell enables the determination of heat transfer coefficients on the oil covered shell walls. Additionally to the main task of lubricating the moving compressor parts the oil acts as the main heat transfer media between the hot compressor parts and the compressor shell and thus to the ambience. High heat transfer coefficients result in increased heat transfer and improved compressor performance by decreasing the overall compressor temperature.

Results of the determination of the heat transfer coefficient on the oil covered shell walls can be seen in Figure 3a. Areas on which the oil jet hits the walls show higher heat transfer coefficients compared to areas where the heat transfer is affected by the falling oil film.

To enable the use of the oil distribution simulation results in the thermal model, the oil covered wall of the compressor shell is divided into 24 elements. The division of the area in circumferential direction is carried out with eight equidistant fields; in vertical direction the area is divided in three fields whereby the middle field represents the shell area where the oil jet mainly hits the shell wall. Figure 3b shows the distribution of the heat transfer coefficient on the shell after the interpolation process.
3.3. Thermal model

The determination of the temperature field in the hermetic reciprocating compressor using the thermal modelling approach published in [9] is carried out with the heat transfer values gained either by oil distribution CFD results and correlations from the literature. To value the prediction accuracy of the two methods the simulation results are compared with temperature measurements on twelve significant measuring points. The position of the measuring points in the compressor can be seen in [9].

For the validation of the simulation the experimental data are gained in a calorimeter test bench with -23.3 °C evaporating temperature, 45 °C condensing temperature and 32 °C ambient temperature, respectively.

The results of the comparison between the simulation results and the experiment can be seen in the diagram in Figure 4 (the temperatures are referred to the temperature of the shell gas in the experiment). As the diagram shows, the temperatures of solid parts inside the compressor, shell gas and oil sump can be predicted accurately. Due to the generally lower heat transfer coefficients gained by literature correlations compared to those determined by CFD, the heat transfer between the hot compressor parts to the ambience is smaller. So the literature correlation case results in higher temperatures of the inner solid parts compared to the case with CFD heat transfer values. The highest deviation between experiment and simulation with CFD oil distribution regarding the inner compressor parts occurs at the discharge muffler surface with a value of 4.2 K. Using similar heat transfer coefficients between compressor shell and ambient air, the temperature deviation of the shell between the two simulation cases is very small which shows the significant influence of the heat transfer coefficient on the shell outer surface. Generally, the deviation between shell temperatures determined by simulation and experiment is higher compared to the deviation of the inner compressor parts. The maximum temperature deviation occurs at the top of the shell with 4.6 K. As the diagram in Figure 3 illustrates, the prediction accuracy is improved using heat transfer coefficients of the compressor oil gained by CFD compared to correlations of the literature.

Figure 3. Heat transfer coefficient distribution on the inner surface of the compressor shell before (a) and after (b) the interpolation.
Figure 4. Comparison between simulation results and experiment (related to shell gas experiment temperature).

Figure 5 shows the results of the simulation with CFD heat transfer correlations for the compressor oil in terms of solid parts temperature field. The highest temperature values occur at the discharge muffler and not as often expected at the valve plate which can be explained by the relatively long duration of the hot refrigerant gas in the discharge muffler and the cooling of the valve plate by the suction gas. As the shell temperature field shows, the temperature values at the oil covered walls are higher than in the vicinity of the mufflers where the walls are not covered with oil.
The accuracy of the temperature determination of the inner compressor parts could be increased by introducing a higher number of zones with different heat flux values at parts with significant temperature gradients which is the case for the suction and discharge muffler. Furthermore, the use of temperature dependent heat transfer coefficients in combination with several zones at the shell outer surface could increase the accuracy of the thermal model.

An important aspect which should be mentioned at this point, is the application of the present model to different operating conditions. If the compressor load is increased and thus the temperature field inside the compressor rises also the oil and shell temperature increase. Assuming similar temperature difference between oil and shell, the determined heat transfer coefficients could be used for other operating points. More accurate results at other operating points can be gained by carrying out the simulation procedure again.

Due to the investigation of a constant speed compressor the oil distribution at several speeds is not part of the present study. Since the oil transport is highly influenced by the rotational speed of the compressor, the oil distribution would be different and thus the heat transfer.

4. Conclusion

In the presented study the simulation of the oil distribution in a hermetic reciprocating compressor for refrigeration application using commercial CFD software ANSYS Fluent is shown. The simulation was carried out by applying the VOF method to calculate the oil jet flow in the refrigerant environment inside the compressor shell. Furthermore, the CFD results were analysed in terms of heat transfer coefficient between the compressor oil and the oil covered shell walls. The calculated heat transfer coefficients were then used in a thermal model of the entire compressor according to [9]. A comparison of the determined temperature fields with the results of a simulation with heat transfer values gained by literature correlations was carried out and furthermore, the prediction accuracy of the simulations was analysed by a validation with experimental results of a calorimeter test bench. The maximum temperature deviation between simulation results with CFD heat transfer coefficients and experiment was 4.6 K which shows generally good agreement. Due to the lower heat transfer coefficients of
literature correlations, the temperature level inside the compressor is predicted too high compared to the CFD method. Although the temperature prediction of the case using CFD heat transfer coefficients for oil covered shell walls is better compared to the case using literature correlations, the application of this approach has to be reviewed closely in terms of benefit and time. Due to the high computational effort for the CFD simulation this method can only be applied in research and not in the development phase of a compressor. Additionally, the fact that the heat transfer coefficients between solid parts and shell gas were still determined with literature correlations gives scope for further improvements.

Acknowledgements
This work has been carried out within the framework of ECO-COOL, a research project initiated and funded by the FFG (Austrian Research Promotion Agency). Furthermore the authors particularly acknowledge the technical support by Secop Austria GmbH, formerly ACC Austria GmbH and Liebherr-Hausgeräte Lienz GmbH.

References
[1] Todescat M L, Fagotti F, Prata A T and Ferreira R T S 1992 Thermal Energy Analysis in Reciprocating Hermetic Compressors Proc. Int. Compr. Eng. Conf. at Purdue 936
[2] Meyer W A and Thompson E 1998 An Analytical Model of Heat Transfer to the Suction Gas in a Low-Side Hermetic Refrigeration Compressor Proc. Int. Compr. Eng. Conf. at Purdue 662
[3] Ooi K T 2003 Heat Transfer Study of a Hermetic Refrigeration Compressor Appl. Thermal Eng. 23 1931-1945
[4] Sim Y H, Youn Y and Min M K 2000 A Study on Heat Transfer and Performance Analysis of Hermetic Reciprocating Compressor for Refrigeration Proc. Int. Compr. Eng. Conf. at Purdue 1390
[5] Sanvezzo J Jr and Deschamps C J 2012 A Heat Transfer Model Combining Differential and Integral Formulations for Thermal Analysis of Reciprocating Compressors Proc. Int. Compr. Eng. Conf. at Purdue 2104
[6] Lohn S K, Diniz M C and Deschamps C J 2015 A Thermal Model for Analysis of Hermetic Reciprocating Compressors und the On-Off Cycling Operation Condition Proc. Int. Conf. Compr. and their Sys.(London) 012068
[7] Ribas Jr F A 2007 Thermal Analysis of Reciprocating Compressors Proc. Int. Conf. Compr. and their Sys.(London) C65804207
[8] Almbauer R, Burgstaller A, Abisid Z and Nagy D 2006 3-Dimensional Simulation for Obtaining the Heat Transfer Correlations pf a Thermal Network Calculation for a Hermetic Reciprocating Compressor Proc. Int. Compr. Eng. Conf. at Purdue 1794
[9] Posch S, Hopfgartner J, Heimel M, Berger E, Almbauer R and Stangl S 2016 Thermal Analysis of a Hermetic Reciprocating Compressor using Numerical Methods Proc. Int. Compr. Eng. Conf. at Purdue 1215
[10] Moukalled F, Mangani L and Darwish M 2015 The Finite Volume Method in Computational Fluid Dynamics (Springer)
[11] Scardovelli R and Zaleski S 1999 Direct numerical simulation of free-surface and interfacial flow Annual Review Fluid Dynamics 31 567-603
[12] Fluent 14 2011 User Guide (Fluent Inc.)
[13] Posch S, Hopfgartner J, Heimel M, Berger E, Almbauer R and Schöllauf S 2016 A Numerical Investigation of the Oil Pump Suction Behaviour in a Hermetic Reciprocating Compressor Proc. Int. Compr. Eng. Conf. at Purdue 1215
[14] Isaa R I 1986 Solution of the Implicitly Discretised Fluid Flow Equations by Operator-Splitting J. of Comp. Physics 62 40-65