Usage and validation of a fluid structure interaction methodology for the study of different suction valve parameters of a hermetic reciprocating compressor

Wolfgang LANG#, Raimund A. ALMBAUER*, Dalibor JAJCEVIC+
#Christian Doppler Laboratory for Thermodynamics of Reciprocating Engines, Graz, AUSTRIA, langw@vkmb.tugraz.at
*Institute for Internal Combustion Engines and Thermodynamics, Graz, AUSTRIA, almbauer@vkmb.tugraz.at
+Christian Doppler Laboratory for Thermodynamics of Reciprocating Engines, Graz, AUSTRIA, jajcevic@vkmc.tugraz.at

ABSTRACT
The dynamics of the flatter valves inside a hermetic reciprocating compressor used in household appliances is the most important factor concerning the gas dynamic behaviour of such a compressor. Hence for a good valve design and for a reliable simulation of the compressor the ability to predict the movement of the valves is indispensable. The present paper describes a methodology which allows the prediction of the valve dynamics. For the validation of the methodology the simulation results of the suction valve dynamics have been compared with experimental data obtained out of measurements with a Laser-Doppler-Vibrometer. Furthermore the method has been used for the variation of some suction valve parameters and their influence onto the overall compressor performance has been shown. The whole methodology has been implemented into a commercial available CFD code (FLUENT) as a user defined function (UDF).

Keywords: Fluid Structure Interaction, Valve dynamics, Reciprocating Compressor, Valve bending

1. INTRODUCTION

Efficiency is an important factor in the field of reciprocating compressors used in household appliances. With an annual production of more than 100 millions, these machines play a relevant role for the global energy consumption. Due to this fact the EU (European Union) enacts an energy labelling [1] which should force the compressor producers to develop more efficient compressors. To be successful and innovative in this fierce compressor competition, new development tools are necessary.

The methods of computational fluid dynamics (CFD) and finite elements (FEM) are two of these tools. Rapid increase in computational power in the last years provides the possibility to get a deep understanding inside the appearing physical effects. The new obtained knowledge can be used for further improvement of the compressor design and for the increase of the compressor efficiency. In the last few years it pointed out that with a separate treatment of the gas dynamic and the structural problem it's not possible to capture all physical interactions. The complex method of fluid structure interaction (FSI) is an eligible methodology to overcome this lack. The type of valves used in hermetic compressors can be described as a typical FSI problem. Flatter valves (actuated by the pressure difference) inside such a compressor are the most important parts affecting the gas dynamic behaviour. Beside the losses caused by the electrical components and thermodynamic losses the losses caused by the gas dynamic are the biggest player concerning the efficiency of the compressor. Because of this the valves are very important concerning the coefficient of performance (COP), which denotes the quotient between cooling capacity (Q_0) and electric power (P_{el}), which is used as indicator for the efficiency of such a compressor.

1.1. FORMER COMPRESSOR VALVE RESEARCH

In the past various researchers have used different mathematical tools for the prediction of the valve dynamic of a reciprocating compressor. Developed methods range from simple approaches which were used in 1D gas dynamic tools to the first trials with FSI methods.

The most popular and widespread approach is the method presented by Costagiola [2]; he uses a single-degree-of-freedom system to abstract the behaviour of the valves. Further work which uses this basic approach have been done, for example by Nieter and Singh [3], Hayashi et al. [4], Böswirth [5], the set of equations which have been derived out of these works is known as basic valve theory (BVT). This approach has been widely used in 1D gas dynamic tools for the prediction of the valve movement. A more modern approach is presented in Matos et al. [6]; 3D simulations have been done to account for the 3D flow effects, while the valve has been abstracted as circular flat plate which moves parallel to the valve plate. To account for the non symmetric flow field which occurs in such flatter valves, developers started to use the complex methods of FSI. In the work presented by Junghyoun Kim et al. [7] the commercial available tool ADINA has been used to predict the valve dynamic in a 2D way, but with this 2D method not every physical effect can be pictured. Methods which enable the 3D prediction of valve dynamics are needed for further improvement of the valves and a further increase of the compressors efficiency. A quite similar FSI coupling method which will be shown in this work has been presented by Baudille et al. [8] where a 3D FSI method was used for the simulation of the reed valves used in two stroke engines. He used his methodology to study the influence of different materials onto the reed valve behaviour. For the modelling of flatter valves used in hermetic reciprocating compressors, some special problems must be considered, for example high geometric compression ratios (1 : 66) or the closing of the valve, these problems hinder the simulation and must be treated in a special manner.

To keep all of these problems into account this work therefore aims to:

- develop a methodology which allows the prediction of the valve dynamics of a hermetic reciprocating compressor
- validate the methodology on a reference valve with measurements
- using the developed method for the study of various valve parameters

2. BACKGROUND

2.1. THE HERMETIC RECIPROCATING COMPRESSOR

The compressor can be divided into several subsystems listed below (schematically shown in Figure 1).

- Suction line (consists of suction pipe, suction muffler and suction valve)
- Electric motor (asynchronic motor with an efficiency $\eta \approx 0.75$ to 0.85)
- Discharge line (consist of discharge valve, discharge muffler and serpentine)
- Piston-cylinder system (piston and cylinder)
- Shell (upper and lower part hermetically welded)

The vaporized refrigerant enters the compressor through the suction pipe which interfuse the shell. The shell consists of a lower part and an upper part, these two parts form a hermetical capsule with covers the whole compressor domain. From the shell the refrigerant is sucked into the suction muffler, the main aim of this muffler is to keep the superheating of the gas as low as possible while at the same time keeping the pressure loss during the suction process as low as possible. Furthermore it consists of a combination of various chambers for the damping of selected acoustic frequencies. The suction valve separates the suction line form the cylinder. The valve is actually a flat plate of steel which is actuated by the forces induced by the acting pressures. It is the most important part which affects the pressure loss during the suction phase of the reciprocating compressor. Inside the cylinder the refrigerant is compressed by the piston. After the compression the refrigerant leaves the cylinder, while actuating the discharge valve (quite similar to the suction valve) and run into the discharge muffler. The discharge muffler is a combination of various plenums to dampen the induced pressure pulsations. After the discharge muffler the refrigerant enters the serpentine, a small pipe for further damping of pulsations. Finally the refrigerant leaves the compressor through the discharge pipe.

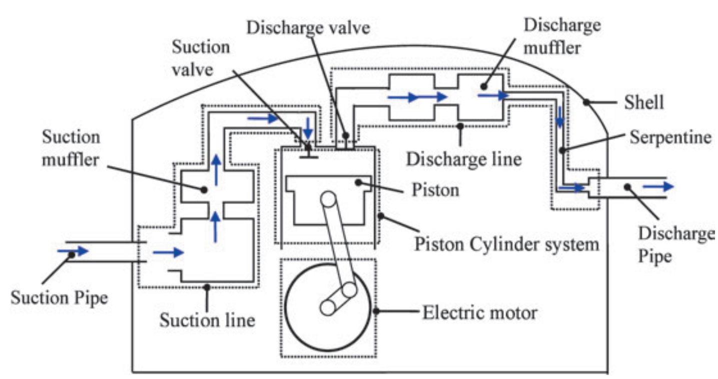

Figure 1 schematic compressor.

2.2. GENERAL COMPRESSOR DATA

The compressor on which all the simulations and measurements have been done can be pictured with some key figures listed below.

- Bore: 21.1 [mm]
- Stroke: 16 [mm]
- Compression ratio: 66 [-]
- Electric power: ~ 58 [W]

The compressor is nearly identically to a HTK 55AA ACC serial production compressor.

2.3. THE COOLING CYCLE

Such compressors described in subsection 2.1 are mainly used in household cooling appliance. To allow a comparison between different compressor designs various reference cycles, which picture the whole physical effects inside a cooling appliance, are available. A general representation of real conditions in a refrigerator and a common known reference cooling cycle is the ASHRAE (American Society of Heating, Refrigerating, and Air-Conditioning Engineers) cycle (defined in ASHRAE [9]). For the used refrigerant R600a (isobutene) this cycle can be seen in Figure 2. The cycle consists of four main parts, the compression (from point 1 to 4), the condensation (from point 4 to 5), the expansion using a capillary tube (from 5 to 6) and the evaporation (from 6 to 1). The vaporized gas is sucked from the evaporator into the compressor, with a nominal pressure of about 0.627 bar and the corresponding temperature of 32° degree Celsius. Inside the compressor the gas is heated by various heat fluxes which causes an increase of the indicated power (1 – 2) compared to the ideal process. In the cylinder the compression (2 – 3) takes place to a pressure of more than 7.78 bar. After the compression the refrigerant leaves the compressor (4) and runs into the condenser where the condensation takes place at the cycle specific temperature of 55.3° degree Celsius. After a sub cooling to 32°C the gas is expanded in a capillary tube and evaporated inside the evaporator.

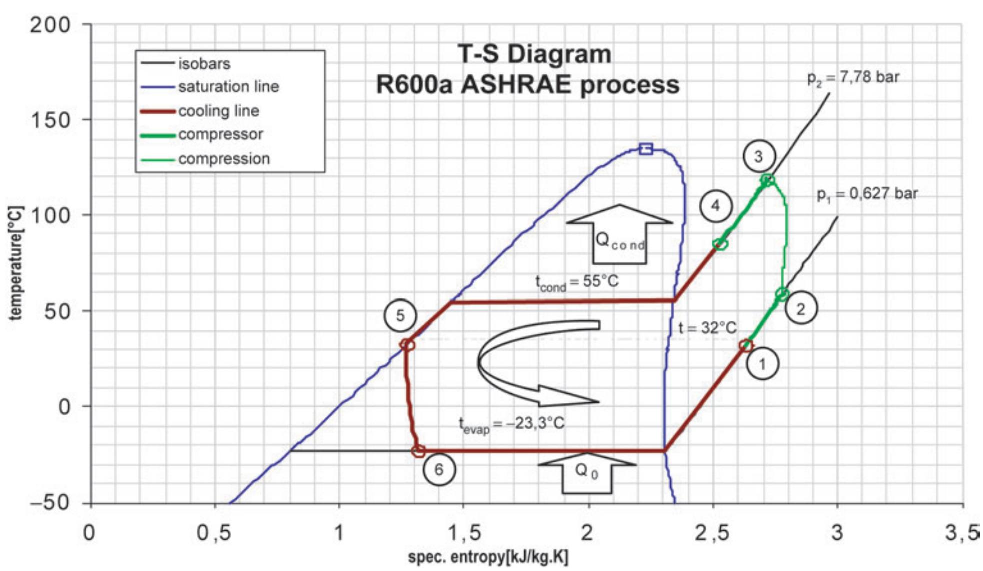

Figure 2 ASHRAE cooling cycle for R600A.

3. METHODOLOGY

3.1. INVESTIGATED DOMAIN

The gas dynamic interactions between the different gas guiding parts through the compressor hinder a separate treatment of the different domains. For example the suction valve movement is strongly influenced by the transient pressure fluctuation inside the suction muffler. Furthermore the pressure inside the cylinder depends on the mass inside the cylinder which is related to the opening of the discharge valve, which on the other hand is actuated by the transient pressure difference between discharge line and cylinder. Because of these interactions it is obvious that for a correct prediction of the suction valve dynamic the whole gas dynamic processes inside the compressor have to be considered. To keep the numerical effort in a range which is useable in industrial applications, the use of methods which handle the coupling of a 3D domain with a 1D domain allows the reduction of the 3D domains to a minimum and so to reduce the numerical effort. Due to this above mentioned considerations the investigated calculation domains was divided into 3 sections:

- the 3D fluid domain
- the 1D fluid domain
- the 3D structural domain

In the following subsections these three calculation domains are explained separately.

3.2. THE 3D FLUID DOMAIN

The 3D fluid domain includes the shell, the suction muffler, the suction valve; the valve plate, the piston and the cylinder. These domains are shown in Figure 3 and Figure 4. To simplify the meshing process some small modifications have been done for the top of the piston. The piston nose (which is normally used to reduce the clearance volume in the TDC) and the piston cavity have been removed, booth seen in Figure 4. To account for these modifications, the distance from the valve plate to the top of the piston in the TDC has been modified in that way, that the clearance volume without nose is equal to the clearance volume with nose.

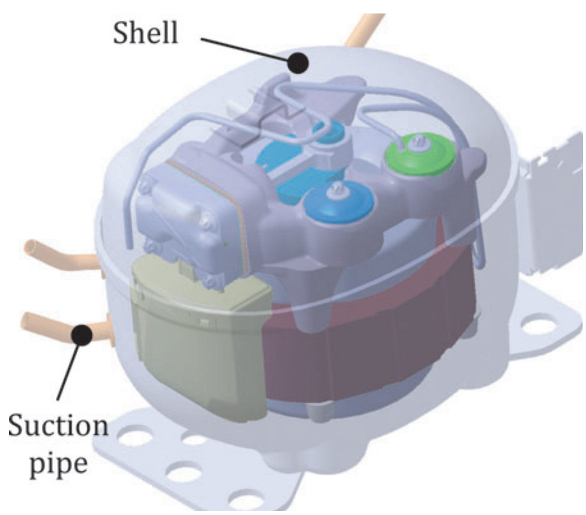

Figure 3 Isometric compressor view.

At the inlet of the suction pipe, a pressure inlet boundary has been set with 0.627 bar and 32 °C as input values. These values represent the values at point (1) in Figure 2. At the outlet of the cylinder a 1D gas dynamic tool (implemented as UDF) was coupled with the 3D domain to reduce the calculation time, further details to the 1D tool can be seen in subsection 3.3. Boundary conditions at the solid walls have been set to constant temperatures, obtained out from measurement data. For the treatment of the turbulence the standard k-ε model has been used. Wall influence was considered by the standard wall function, and a second order discretisation has been used in time and space. For the coupling of pressure and velocity the

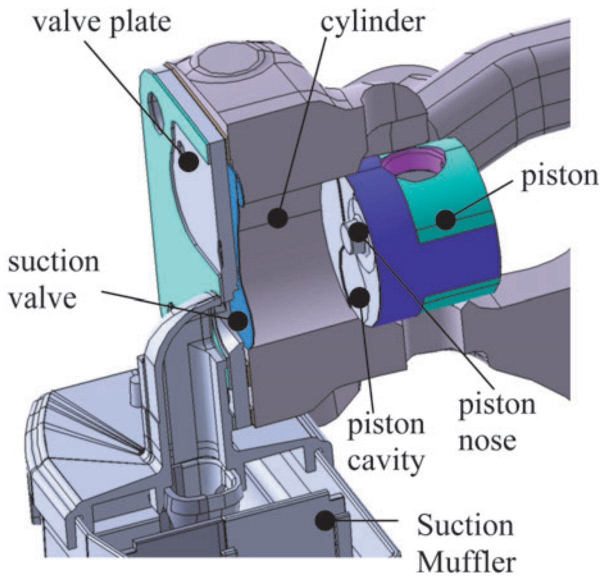

Figure 4 Compressor parts.

SIMPLEC scheme has been chosen. The whole 3D domain consists of a mesh (tetrahedral cells and hexahedral cells) of about 0.4 million cells in the TDC and about 1.8 million cells in the BDC. In Figure 5 the mesh strategy can be seen. The upper region of the cylinder (3 mm from the valve plate), where the valve movement takes place was meshed with tetrahedral cells, this was necessary due to the fact, that the remeshing algorithm provided by the used CFD – tool (Fluent 6.3.26) works only with tetrahedral cells. In the lower region of

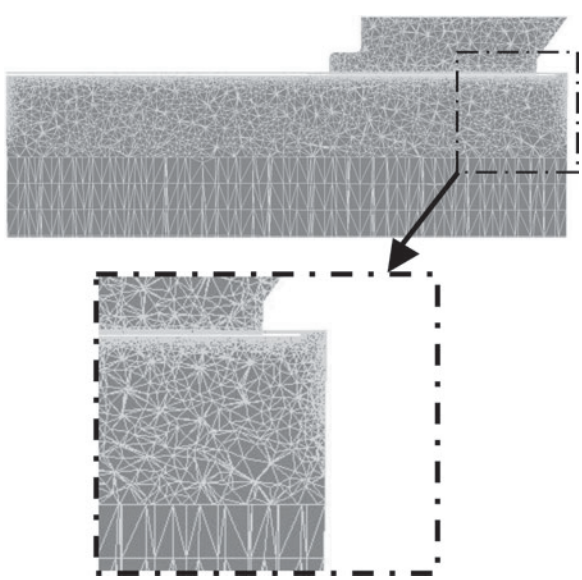

Figure 5 Cross-sectional cut through the cylinder mesh.

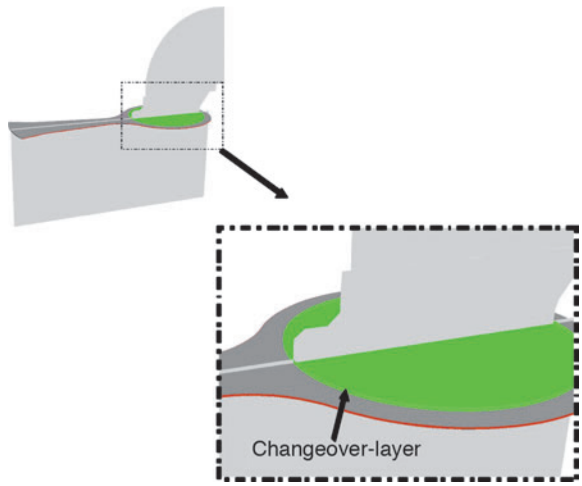

Figure 6 Location of change-over layer.

the cylinder hexahedral cells have been used, to reduce the mesh size and furthermore to reduce the simulation time.

Until now Fluent doesn't allow to change the boundary conditions depending on the physical behaviour inside the calculation domain. In addition it isn't possible to create mesh cells out of a region, where no cells have been before. To overcome these problems, a small gap has been created. To close this gap a layer has been implemented (Changeover-layer) which is set to "interior" if the valve lift is bigger than 0 and which sets the layer to "wall" if the valve lift is less equal 0. The layer is controlled by a UDF (in combination with a scheme file). In Figure 6 the location of this layer can be seen.

3.3. THE 1D FLUID DOMAIN

Figure 7 represents the discharge line of the compressor which consists of the cylinder head (abstracted as Pl 1), discharge valve, two additional discharge plenums (Pl 2 and Pl 3), and the serpentine (P4). The gas dynamic of the 1D domain was connected with the 3D domain described in subsection 3.2, by the use of a coupling algorithm [10]. The principle steps of the coupling algorithm works as following: the pressure of the 3D domain (cylinder outlet) and the 1D domain (pipe 1) have been used to calculate the discharge valve lift, using equation (1):

$$m \cdot \ddot{x} + d \cdot \dot{x} + c \cdot x = \Delta p \cdot A - F_0 \tag{1}$$

In the above written equation, the parameter m denotes the mass of the discharge valve, d denotes the damping coefficient and c stands for the stiffness of the valve. The parameters x, $\dot{x}$ *and* $\ddot{x}$ denote the valve lift, valve velocity and the acceleration of the discharge valve. Further parameters are A which stands for the surface area of the valve and the preload F_0. Δp represents the former mentioned pressure difference between the 3D domain and the 1D domain.

With the calculated valve lift and various flow coefficients (obtained out of 3D stationary CFD simulations) it was possible to obtain the mass flow rate (by the use of the orifice formula), which runs out of the cylinder into the 1D domain or vice versa.

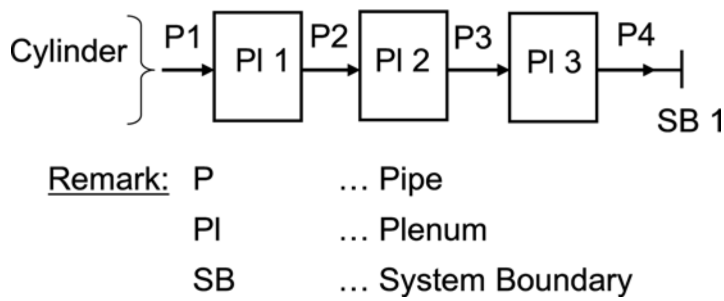

Figure 7 1D discharge line.

The gas dynamic inside the pipes of the 1D fluid domain has been obtained by the solution of the Euler equations (equation (2) to equation (4)).

$$\frac{\partial\rho}{\partial t}+\frac{\partial(\rho u)}{\partial x}+S_1=0 \tag{2}$$

$$\frac{\partial\rho u}{\partial t}+\frac{\partial\left(\rho u^2+p\right)}{\partial x}+S_2=0 \tag{3}$$

$$\frac{\partial E}{\partial t}+\frac{\partial\left[u(E+p)\right]}{\partial x}+S_3=0 \tag{4}$$

with ρ = density, u = velocity, p = pressure, S = source, t = time. Here E is the total energy per unit volume, namely:

$$E=\rho\left(\frac{1}{2}u^2+e\right) \tag{5}$$

The Euler equations have been solved by the usage of a second order TVD (Total variation diminishing) – scheme (Coberan et al. [11]). On the outlet of the 1D domain the values of 7.78 bar and 127°C was set as boundary values. Heat transfers have been taken into account, with fixed values for the plenum wall temperatures has been set, obtained out of measurements.

3.4. THE 3D STRUCTURAL DOMAIN

The structural domain consists only of the suction valve of the compressor. Figure 8 shows the suction valve for a HTK55 serial production compressor. For the discretization of the valve, beam elements with 6 dof (degrees of freedom) per node have been used. Such an element can be seen in Figure 9. The usage of beam elements has some advantages, like short calculation times and reasonable results with a small number of cells. Small errors concerning the surface area appear because beam elements with constant cross sections are used. A detailed look onto the fixing area shows, that the curved surface at the fixing point couldn't be resolved exactly. (see Figure 8). All cases have been calculated without the influence of damping. Furthermore some sticking effects induced by the oil between the valve plate and the valves aren't considered. This negligence should be kept in mind if comparing the obtained results with measurement data. However for comparative studies and searching for the influence onto the overall performance parameters we expect a minor influence.

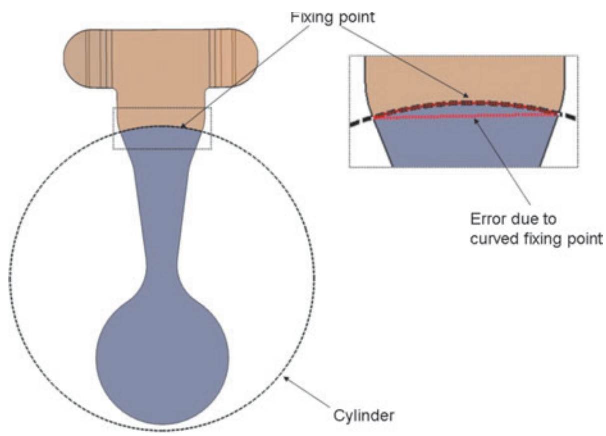

Figure 8 Suction valve boundary & error sources.

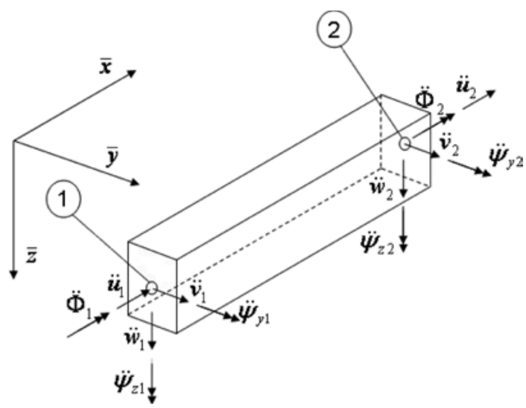

Figure 9 Beam element.

3.4.1. FEM equations & Time integration

The equation of motion for the linear dynamic which have been solved for the structural domain, can be seen in equation (6). M denotes the mass matrix, K the stiffness matrix and D stands for the damping matrix. For implementing the effects of damping a linear dependency of mass and stiffness matrix can be chosen, which is called Rayleigh-damping.

$$M \cdot \ddot{q} + D \cdot \dot{q} + K \cdot q = f \tag{6}$$

Equation (6) can be also rewritten as follows

$$\hat{K} \cdot q^{t+\Delta t} = \hat{f}^{t+\Delta t} \tag{7}$$

with $\hat{K}$ called effective stiffness matrix

$$\hat{K} = K + a_0 \cdot M + a_1 \cdot D \tag{8}$$

and $\hat{f}$ being the effective force vector

$$\hat{f}^{t+\Delta t} = f^{t+\Delta t} + M \cdot (a_0 \cdot q_t + a_2 \cdot \dot{q}_t + a_3 \cdot \ddot{q}_t) + D \cdot (a_1 \cdot q_t + a_4 \cdot \dot{q}_t + a_5 \cdot \ddot{q}_t) \tag{9}$$

with the parameters a_1 to a_7

$$a_0 = \frac{1}{\alpha \cdot \Delta t^2}; \quad a_1 = \frac{\delta}{\alpha \cdot \Delta t}; \quad a_2 = \frac{1}{\alpha \cdot \Delta t} \tag{10}$$

$$\begin{aligned} &a_3 = \frac{1}{2 \cdot \alpha} - 1; \quad a_4 = \frac{\delta}{\alpha} - 1; \quad a_5 = \frac{\Delta t}{2} \cdot \left(\frac{\delta}{\alpha} - 2 \right) \\ &a_6 = \Delta t \cdot (1 - \delta); \quad a_7 = \delta \cdot \Delta t \end{aligned} \tag{11}$$

The time integration has already been implemented into equation ((7) – (9)), for which a Newmark algorithm has been chosen, with the parameters $\delta = 0.5$ and $\alpha = 0.25$. For these parameters the Newmark scheme is implicit, imperative stable, has no numerical damping, and a high accuracy. A disadvantage of these parameters can be seen in the numerical elongation of the oscillation period. To obtain a solution of this system of equations a direct solver with a LU-Decomposition was used. Further details concerning the finite element method or the time integration can be seen in [12] and [13].

3.4.2. Contact modeling

Contact modeling is a highly complex topic and usually it is connected with a tremendous need of computational power. Because of the aim to establish a method which is usable for industrial applications, a simplified model to account for the impact of the valve onto the valve plate has been used. The first step for the contact model is to detect, whether there is a contact or not. At every time step it is checked if a node touches the valve plate or not. If it's touches the valve plate a reaction force is calculated including the terms of inertia and an impact term which is a function of the mass corresponding to the node and the velocity of the node. This reaction force is multiplied with a restitution coefficient, which accounts for the dissipation of energy during the impact. The restitution coefficient has been first chosen out of literature with a value of 0.7 for an elastic impact steel on steel. After some trials it has figured out that a value between 0.75 to 0.85 results in much better valve lifts. Obviously this simple model doesn't account for the complex physical effects appearing during the touching of two different parts, but it has been seen that this simple approach produces relatively good results.

3.4.3. Coupling algorithm

Coupling of finite element codes and CFD tools becomes popular in the last years. The reason for this on the one side is that engineers recognize the importance of the interaction between fluid and solid. On the other side the increase of computational power allows a deeper insight into the physics of various effects. The coupling procedures between FEM & CFD can be divided into two different types, Figure 10 gives an overview of these methods according to [14] and [15].

In general there are two different methods, the direct method and the partioned method. The direct method is the most sophisticated method, here fluid and solid domain form one set of equations which is solved together. It's the most exact method, but the high numerical effort makes this method until now unpractical for cases with more than one million computational cells. An easier method is called the partioned method. Here two different variants are available, which differ from the timing of the data exchange. The data exchange between solid and fluid domain can take place every iteration or every time step. These methods are more general and easier to implement and enables the advantage to evaluate

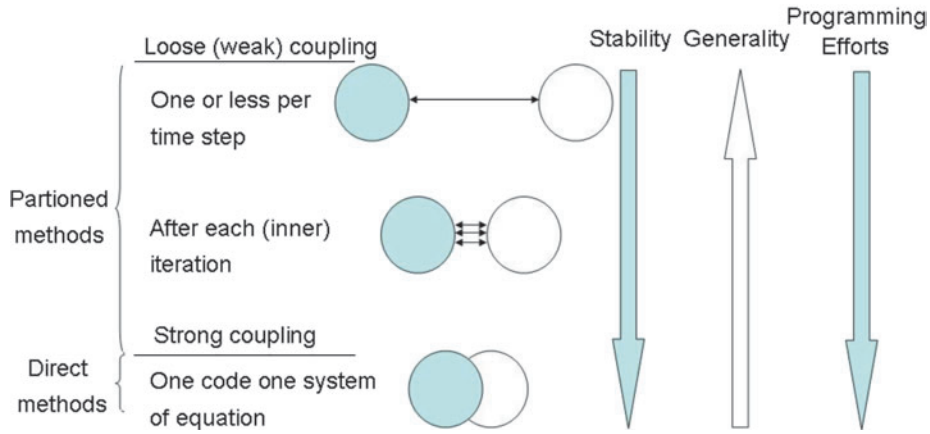

Figure 10 FSI – coupling schemes acc. to [14].

every problem (fluid or solid) using an optimized solver. A Disadvantage of this method is a smaller stability compared to the direct methods. Therefore smaller time steps have to be used. The used CFD-tool allows only a remeshing process between two time steps. Finally a partioned approach with the data exchange after every time step was employed. The detailed coupling process can be seen in Figure 11.

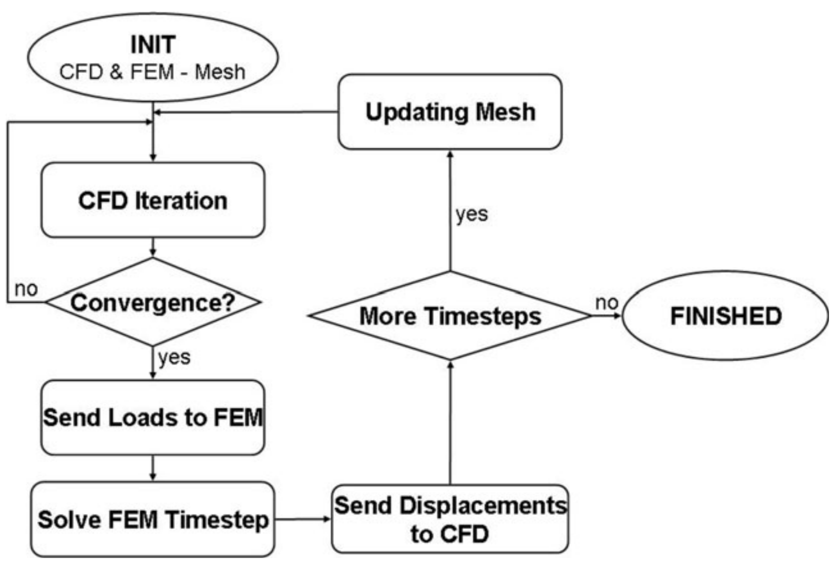

Figure 11 Flow chart of coupling algorithm see [8].

After creating an initial mesh, the CFD simulation takes place. It iterates until a converged solution is obtained. The loads which act on the solid domain are passed to the FEM code (implemented as UDF) and then one FEM time step is calculated. After that the displacements calculated from the FEM code are sent back to the CFD tool. Because of using different meshes for the solid and fluid domain a linear mesh interpolation was carried out. If more time steps are necessary the mesh is updated and the calculation starts from the beginning, otherwise the simulation is finished.

4. VALIDATION

4.1. TEST BENCH

To keep measuring conditions as close as possible to reality all measurements have been done in a “low power” calorimeter special designed for testing compressor for small cooling capacities. The key figures of the test bench can be seen below.

- Refrigerant: R600A
- Cooling capacity: < 300 [W]
- Compressor electric (input) power: < 200 [W]
- Conditioned room temperature: 25 – 40 [°C] (adjustable)
- Suction pressure: 0.3 – 1.5 [bar] (adjustable)
- Discharge pressure: 5 – 10 [bar] (adjustable)

4.2. EXPERIMENTAL METHOD

For the validation of the simulation approach it is necessary to measure the movement of the valve with a high accuracy and under real operating conditions. The complex structure of the compressor assembly hinders a simple accessibility to the valves. Furthermore the valves are small light parts and so mounting a transducer is not possible. It can be seen that with a standard measuring system is not possible to detect the valve lift in a correct manner. Nagy et al. [16], presents a measuring system which fulfils all requirements for measuring this small and light parts. He uses a Laser Doppler Vibrometer for the measurement. To guide the laser beam to the surface of the suction valve, he uses an acrylic glas rod, which interfuses the shell and the suction muffler. A picture of the measuring setup can be seen in Figure 12. The laser beam is guided through the suction valve and produced repeatable time resolved lift curves for both valves.

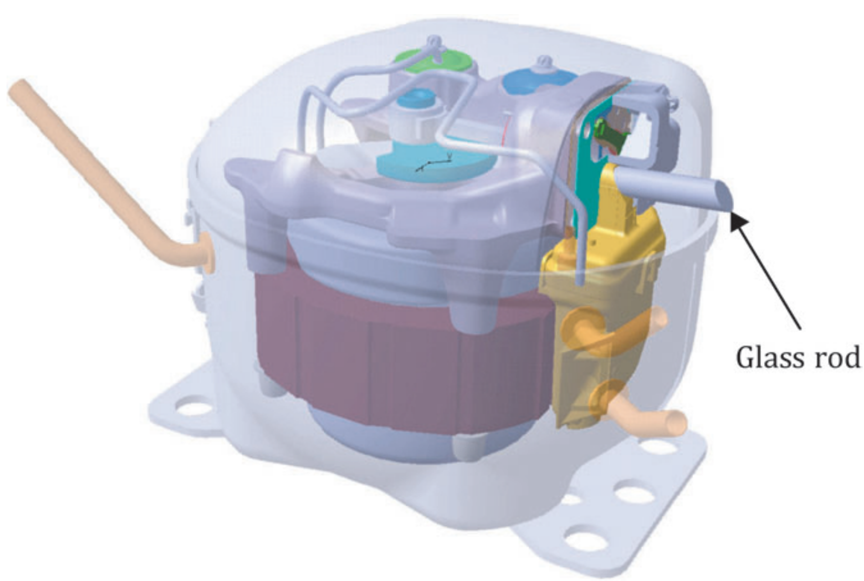

Figure 12 Measuring setup for the valve measurements.

4.3. EXPERIMENT VS. SIMULATION

For the validation of the measuring system the standard configuration of the suction valve has been measured and simulated. The comparison between measurement and simulation can be seen in Figure 13.

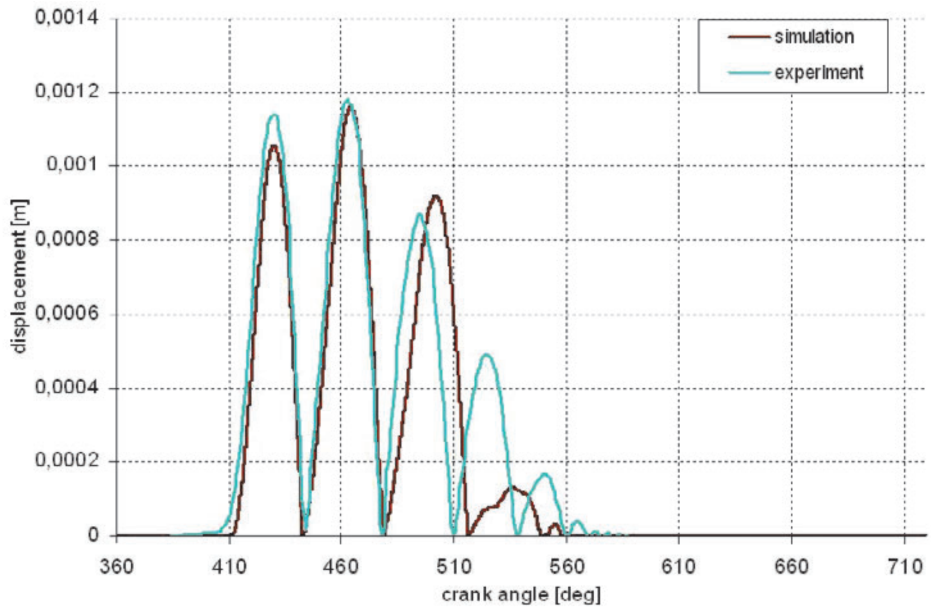

Figure 13 Suction valve lift simulation vs. experiment.

It can be seen that the first three valve openings are predicted in a quite good manner. Discrepancies in the last two valve openings can be seen. These discrepancies could be caused, because of the negligence of the oil sticking forces and there are also problems predicting the valve dynamics in the late opening phases. This problem in the prediction appears, because of the difference in the valve opening and valve closing forces become much smaller in this late phase (30 – 40 degree before BDC). These differences in the force distribution can be seen in Figure 14 and Figure 15.

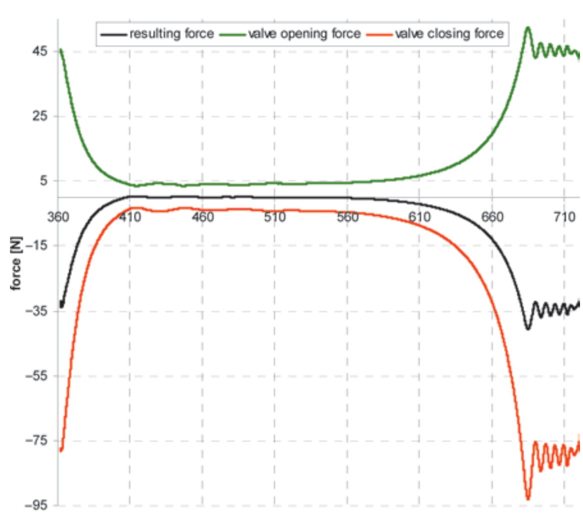

Figure 14 Integrated forces over a suction valve.

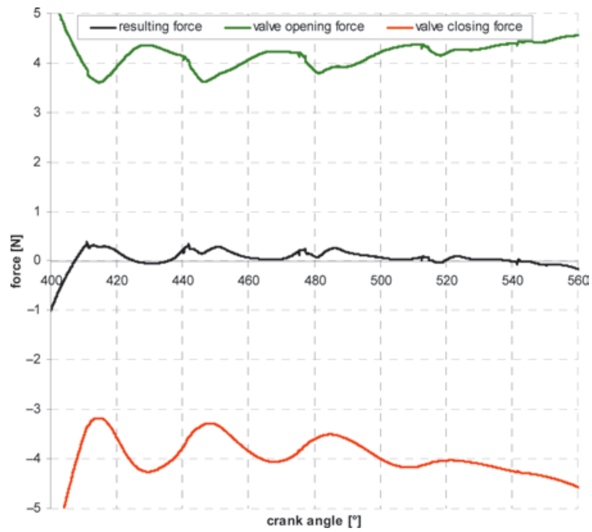

Figure 15 Integrated forces in over the suction valve from 40 to 200 degree crank angle (after TDC).

In Figure 14 the integrated forces over the entire suction valve area are shown. The forces which where pictured are the forces which try to open the valve (pressure forces) and forces which try to close the valve like (pressure force and preload forces). The valve opening forces are denoted in a positive direction. Figure 15 gives a detailed look to the forces, when the valve is in its opening phase. For the validation it can be said that the comparison with the experimental results shows sufficient accuracy, especial in the important points like the opening time span and the phase of the suction valve opening can be predicted in an adequate way.

5. RESULTS

For the variation of the suction valve, 3 different cases have been set up. The values which are varied are the most important parameters for the valve design:

- natural frequency
- pre load force
- bore diameter

The natural frequency (natural frequency of standard valve ~ 300 Hz) was changed by increasing the valve thickness. The preload force have been varied with a higher value compared to the standard valve. For this force no smaller value than the original value (–0.15 N) can be used, because otherwise the valve won't close anymore and no measuring data can be achieved. For the bore diameter only a reduction of the inlet bore diameter was possible, because the construction of the compressor, doesn't allow to use bigger valves than the standard one (bore diameter = 7.8 mm). Table 1 summarizes the test setup.

Table 1 overview of the test cases

Natural frequency		**Preload force**		**Bore diameter**	
Standard	Lowered	Standard	Increased	Standard	Reduced
~ 300 Hz	~ 150 Hz	–0.15 N	–0.25 N	7.8 mm	5.8 mm

The influence of the parameter variations results in a change of the main performance parameter of the compressor. These parameters are as already mentioned in a former section COP, Q_0 and P_{el}. The effects of the changes are compared with measured data. The measured data for the electric power has been obtained out of the indicated power multiplied with the known values for the mechanical and electrical efficiency of the used test compressor. The values for the standard valve, for simulation and measurement have been have been set as reference values (100 percent), in Table 2 the differences in the change are drawn.

Table 2 Effect of parameter variation

	Natural frequency			Preload force			Bore diameter		
	PEL	Q0	COP	PEL	Q0	COP	PEL	Q0	COP
Simulation	–0.63	–1.06	–0.43	–3.77	–7.03	–3.38	–1.94	–4.78	–2.87
Measurement	0.58	–1.24	–1.81	–3.93	–6.58	–3.77	–1.58	–3.78	–3.23

It can be seen that the effects caused by the changed parameters can be predicted in an adequate way, the difference between simulation and measurements are rather small.

The effects of the parameter variation can be directly seen in the mass flow history (Figure 16) and in a pressure change during the suction process of the reciprocating compressor (Figure 17). An advantage in the use of this relatively simple procedure lies in a short simulation times. It takes approximately 2.5 days to calculate one revolution of the compressor. All simulations have been done on 1 CPU at an Intel Xeon@3.0Ghz computer with 32 GB memory. In order to accomplish a stationary result 3 to 5 revolutions have to be simulated. A further decrease of simulation time can be achieved by the use of parallel computing power.

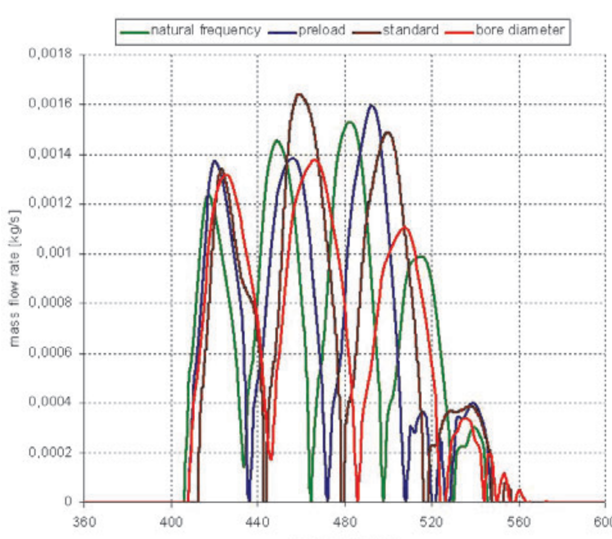

Figure 16 Effect of parameter variation on mass flow rate.

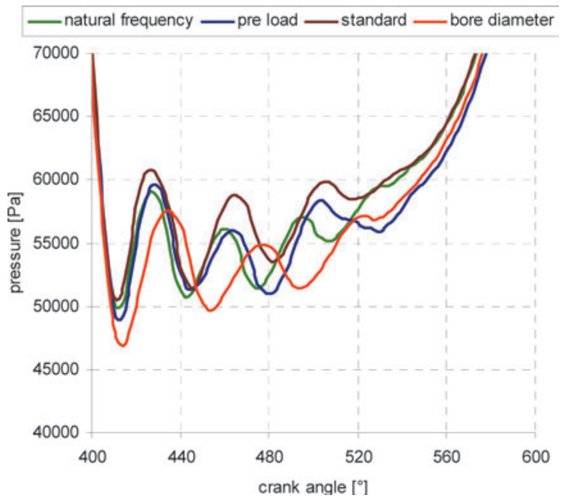

Figure 17 Effect of parameter variation on suction pressure.

6. CONCLUSION

A numerical method which allows the 3 dimensional simulation of the suction valve behaviour has been presented. The method has been validated by the use of a measuring method already available in literature. The validation has shown sufficient agreement, between the measured and the simulated results. Furthermore to show the abilities of the method, different changes of the main suction valve parameters has been done, and the effects of the changes has been evaluated by experimental and simulation methods. It has been shown that the method is able to reproduce changes onto the main suction valve parameters in a sufficient manner. No doubt about the fact, that more complex methods for example more detailed contact models or other effects, can further improve the results. But this method has shown its ability to capture the main effects during the valve dynamics with reasonable high accuracy and in a relatively short time, which enables its use in industrial applications.

REFERENCES

[1] EU Commission Directive 2003/66 EG

[2] Costagiola M., 1950.: The theory of spring-loaded valves for reciprocating compressors, Journal of Applied Mechanics, vol. 17, p. 415 – 420.Forbes Pearson S. 1996, Uses of Hydrocarbon Refrigerants, Proc. Aarhus Conference, IIF/IIR: 439–446

[3] Nieter J.J. and Singh R., 1984.: A computer simulation study of compressor tuning phenomena, Journal of Sound and Vibration 97 (1984), pp. 475–488

[4] Hayashi S., Hayase T. and Kurahashi T., 1997.: Chaos in a hydraulic control valve, Journal of Fluids and Structures 11 (1997), pp.693–716

[5] Böswirth L., 2000.: Strömungen und Ventilplattenbewegung in Kolbenverdichterventilen, Eigenverlag, Vienna (2000)

[6] Matos F.F.S, Prata A.T. and Deschamps C.J. 1999.: Numerical analysis of the dynamic behavior of plate valves in reciprocating compressors. In: International Conference on Compressors and Their Systems. London, 13–15 September 1999, pp. 453–462.

[7] Junghyoun Kim, Semyung Wang, Sungwoo Park, Kioh Ryu and Jongpill La, 2006.:Valve dynamic analysis of a hermetic reiprocating compressor, Proceedings of the International Compressor Conference Purdue, Purdue University, IN, USA

[8] Baudille R., Biancolini M.E., 2006.: Modelling FSI prolbems in Fluent: a dedicated approach by means of UDF programming, Proceedings of the European Automotive CFD Conference, FISITA Congress 2006, F2006M235

[9] ASHRAE. (2005) 'Method of Testing for Rating Positive Displacement Refrigerant Compressors and Condensing Unit', ANSI/ASHRAE Standard-23.

[10] Lang W., Almbauer R.A., Burgstaller A., Nagy D.: Coupling of 0-, 1- and 3-d Tool fort he Simulation of the Suction Line of a Hermetic Reciprocating Compressor, Proceedings of the International Compressor Conference at Purdue, 2008, West Lafayette, USA, Paper 1272

[11] Corberan, J.M., Gascon, M.L. (1994) 'TVD Schemes for the Calculation of Flow in Pipes of Variable Cross-Section'. Mathematical Computational Modelling, Vol.21. No.3 pp.85–92.

[12] Zienkiewicz, O.C., and Taylor R.L., 2000.: The Finite Element Method, Third, fourth and fifth editions, McGraw-Hill, London, 1977 and 1989/91, Butterworh and Heinemann, Oxford, UK, Boston, US.

[13] Newmark N.M., 1959.: A Method for Computation of Structural Dynamics, Proceedings of the ASCE, 85 EM3, pp. 67–94.

[14] Sieber G., 2001.; Numerical Simulation of Fluid-Structure Interaction using loose coupling methods, Dissertation, Technische Universität Darmstadt, Germany

[15] Glück M., 2002.: Ein Beitrag zur numerischen Simulation von Fluid-Struktur-Interaktionen – Grundlagenuntersuchungen und Anwendung auf Membrantragwerke, Dissertation, Technische Fakultät der Universität Erlangen-Nürnberg, Germany

[16] Nagy D., Almbauer R.A., Lang W., Burgstaller A.: Valve lift measurement for the validation of a compressor simulation model, Proceedings of the International Compressor Conference at Purdue, 2008, West Lafayette, USA, Paper 1274