2016

An Investigation Into The Dynamics Of Self-Acting Compressor Valves

Carsten Möhl
Technische Universität Dresden, Germany, carsten.moehl@tu-dresden.de

Christiane Thomas
Technische Universität Dresden, Germany, christiane.thomas@tu-dresden.de

Ullrich Hesse
Technische Universität Dresden, Germany, ullrich.hesse@tu-dresden.de

An Investigation Into The Dynamics Of Self-Acting Compressor Valves

Carsten MÖHL[1]*, Christiane THOMAS[1], Ullrich HESSE[1]

[1]Institute of Power Engineering
Bitzer Chair of Refrigeration, Cryogenics and Compressor Technology
Technische Universität Dresden,
Dresden, Germany

* Corresponding Author
carsten.moehl@tu-dresden.de

ABSTRACT

In principle, a reciprocating compressor stage consists of a cylinder, the working volume which varies periodically due to a piston moving inside a liner. Latter is sealed by two sets of valves. On one hand the suction valve for admitting the gas to be compressed and on the other the discharge valve for allowing the high pressure gas to be delivered to the downstream process. The typical type of valves used within compressors for refrigeration plants are reed valves. State-of-the-art computation of these valves is of zero or one-dimensional order; however the number of publications regarding 3D numerical simulation is increasing. Among these two ways the user has to compromise between computation time and accuracy. This paper tries to address one possible alternative as its focusing on the application of two-way fluid-structure-interaction (FSI) in a two-dimensional representation of a reciprocating compressor. The results are validated against an established simulation software as well as measurement data. Furthermore, some modeling peculiarities will be elaborated enabling better insight to users which are new to this subject.

1. INTRODUCTION

Mathematical modeling as well as simulation is widely applied in the design of compressors. The in-depth analysis of underlying processes enables complicated problems to be solved particularly fast with the help of computers. A considerable amount of previous works has been published on the mathematical modeling and simulation, starting with Michael Costagliola (Costagliola, 1949). He developed the fundamental set of equations on valve dynamics affecting the performance of reciprocating compressors back in the days, when computers were difficult to access and programmed via perforated paper tape. Despite some simplifications, like omitting the rebound effect when the valve hits the limiter, his work was the starting point for many others to come.

20 years later, the first model with the need for solving two nonlinear differential equations simultaneously was introduced by MacLaren & Kerr (MacLaren & Kerr, 1969). The analytical model – assisted by several empirical coefficients – included plenum as well as pipeline pulsations and lead to some of the first studies regarding valve lifetime by using a mathematical model. These models, which were supported by empirical data, were consecutively improved until the end of the 20th century. Due to the large amount, only some of the major key influencers shall be named and referred to. Simon Touber developed and published his model on valve dynamics with the focus on solving with a computer (Touber, 1976). He even compared the computation times as well as costs of analog and digital computers. His extensive theory, decomposition of various valve influences plus comparison of different types of valves lead to a well-rounded set of equations for valve dynamics description. Concluding there is the work of Leopold Böswirth (Böswirth, 1994). His research of the effects of valve flutter resulted in an improvement of equations characterizing the flow and a similitude theory for transient processes within the valve as well as the cylinder. The second edition of his self-published work was amongst other things extended by one chapter focusing on the simulation of valve dynamics using the software KV-DYN, which will be used for comparison later in this paper.

Following these zero and one-dimensional modelling methods, making use of the parallel improvement of computer hardware and thus the reduction in the average calculating time, the application of 3D simulation started approximately with the turn of the millennium. At first the computations were often conducted in 2D, like seen by

Georg Flade as he studied the opening process of plate valves (Flade, 2006), later on even considering the structural reaction of the suction and discharge valve in a hermetic compressor (Rodrigues, 2014). To overcome several hurdles, some of them will be addressed in the later part of this paper, in modeling a complete working cycle, the extensive usage of interrupt control and 4 different sub-models had to be applied in ANSYS CFX. Since there were no measurements published to compare to, a qualitative analysis has not been conducted.

2. THE MODEL

As the main focus of this paper is on the application of two-way FSI simulation of reed valve dynamics and shall be the starting point for experiments to follow, the movement of the piston will be the driving force for changes of the working chamber volume and consequently the pressure. This enables the verification against well-established theories as well as current literature. Figure 1 depicts a sectional representation of a deformed reed valve within a reciprocating compressor. As shown in this drawing the limiter forces the reed, which is displaced due to the pressure difference between working and suction chamber, to bend and thus stress is induced.
Nowadays numerical simulation software makes it easy to setup a 3D representation of existing geometries. It is up to the user to decide, whether he tries to simplify the model utilizing symmetry, sectional representation or 2D slices. The resulting computation time biases the decision. As of now, there have been only a few published attempts to apply two-way fluid-structure-interaction (FSI) setups on reed valve dynamics. Considering that the one-dimensional models are still valid and often used, a 2D model could be the next best compromise between computation time and information gained. Thus – for this paper – the sectional representation of Figure 1 is extruded to a thickness of one mesh element in z-direction, leading to Figure 2 respectively Figure 4.

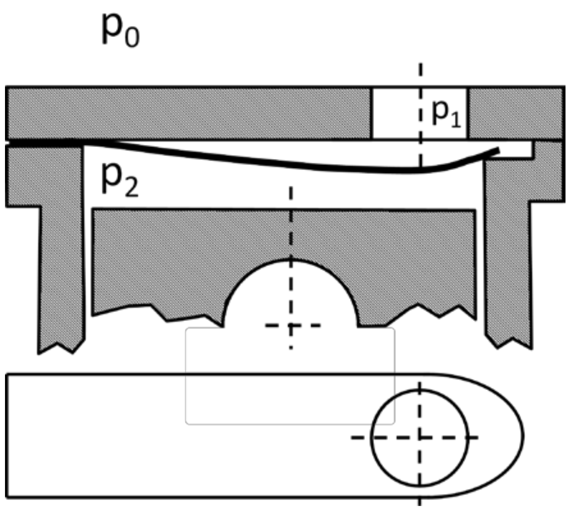


Figure 1: Sectional representation of reed valve deformation within a reciprocating compressor

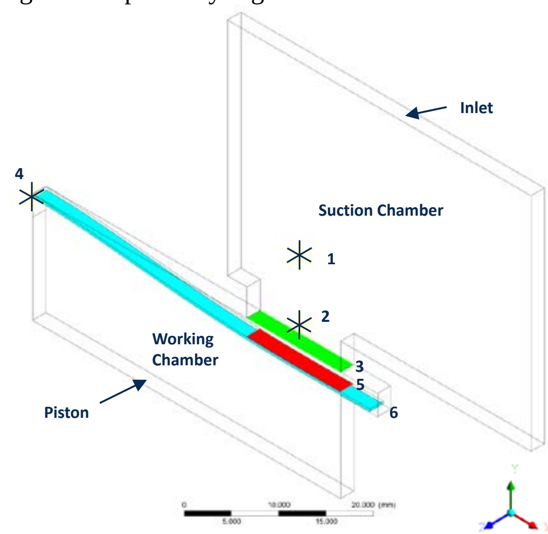


Figure 2: 2D representation of Figure 1 in CFX Post as well as definition of distinctive points, chambers and surfaces

As done in previous studies, the driving force for the necessary pressure difference will be the piston movement (Möhl, Langebach, & Hesse, 2014; Möhl, 2014). This enables to view the valve dynamics as a function of the crank angle und thus makes the data more comparable against literature. The losses due to streaming along the piston circumference will be neglected as there is no gap between the liner and the cylinder.
In consideration of the valve dynamics being a non-linear lift-off because of the fixed bearing as well as the fact that mesh elements cannot be created at run-time in ANSYS CFX, the initial position of the unbend lamella has to be shifted towards the piston. As depicted in Figure 3 a minimal gap (3, white border color) was introduced above the reed valve (2). This leads to streaming effects when the force is not yet able to move the lamella. Given that presently there is no oil fraction in the working fluid the flow can be interpreted as leakage.

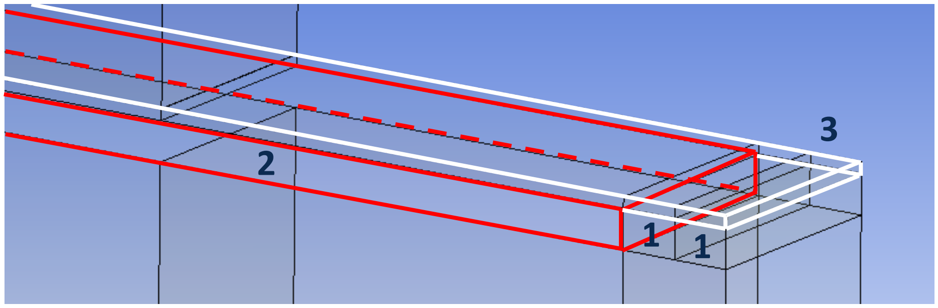


Figure 3: Geometry conditioning for valve dynamics computation

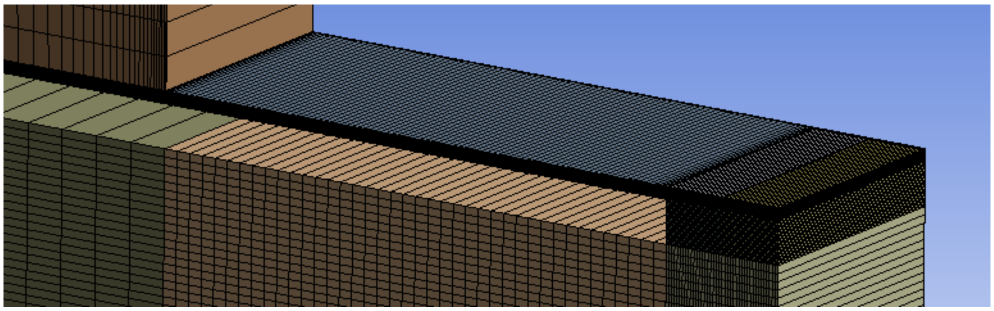


Figure 4: Mesh for initial conditions

Furthermore, the space at the tip of the reed valve should be split up in two distinct volumes for the definition of mesh movement, like shown in Figure 3 (1). The part connected to the outer wall shall only move parallel to its boundary, as the other one is defined as unspecified in motion. The drawback of this kind of modeling is preserving the mesh stiffness and quality. Figure 5 shows the coloring of the mesh according to the orthogonality angle at the tip. The reddening describes the angles in relation to the desirable 90 °. For numerical stability and reliable results one should aim for rectangular elements, even after deformation. Unfortunately, the non-linear movement complicates this task. Therefore, the user needs to influence the mesh stiffness. Advisable is the application of subdomains and the implicit addressing via Ansys CFX expression language (CEL). Table 1 shows example codes for the mesh stiffness interaction as well as the corresponding results. As they may vary according to the conditioned geometry the user should evaluate the results and evaluate different regimes. In this case the application of wall-based criteria should be avoided. This is due to the dimensions, in which this kind of boundary draws the mesh elements out of the deformation zone.

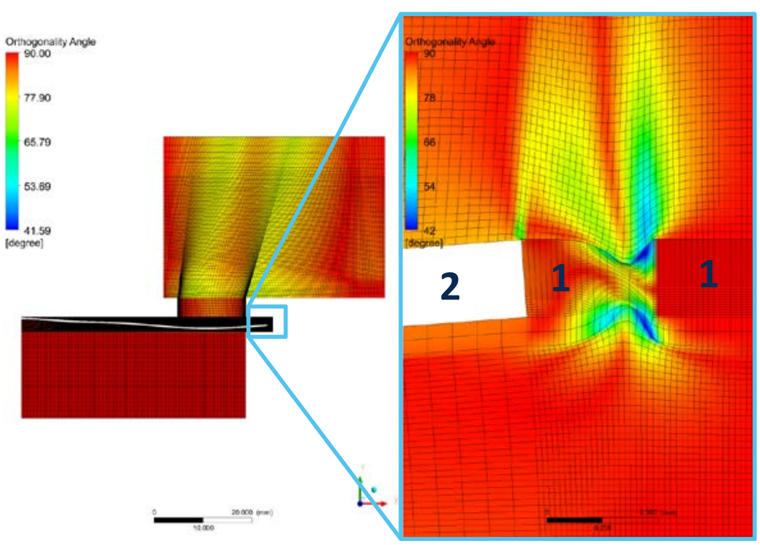
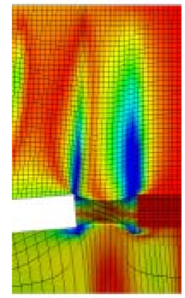
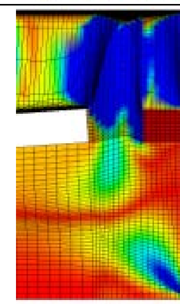
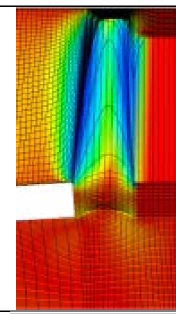
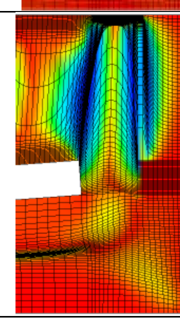


Figure 5: Mesh orthogonality at the reed tip (less reddening, smaller angle)

Table 1: 4 examples for mesh stiffness manipulation, equally scaled

#	Stiffness Parameter Regime	Result
(1)	$\text{Stiffness} = if(\text{ inside()@SubdomainTip } == 1, 1e8\,[m\hat{}2/s], 1[m\hat{}2/s])$	
(2)	Increase near Boundaries	
(3)	Increase near Small Volumes	
(4)	$\text{Stiffness} = if\left(\text{inside()@SubdomainTip } == 1, 1e8\left[\frac{m^2}{s}\right], \text{Stiffness2}\right)$ $\text{Stiffness2} = if\left(\text{inside()@SubdomainAroundReed } == WallFunction, 1\left[\frac{m^2}{s}\right]\right)$ $WallFunction = 1000\,[\text{m\^{}5 / s}] / (\max(0.1[\text{mm}], \text{Wall Distance})\hat{}3)$	

Subsequently to the fluid part of the simulation the structural interaction needs to be setup as it will be solved iteratively. The main focus is on limiting the lift so the reed valve does not hit the wall as the simulation will instantaneously stop. So the definition of a minimal gap between the contact pair is essential. In this study 0.1 mm were used between the limiter and the bottom side of the lamella. Furthermore, the correct material data for the solids should be applied as they tend to differ in tensile strength, fatigue strength as well as ductility (Chai, Zetterholm, & Walden, 2004). For the computations conducted here, the necessary parameters can be taken out of the corresponding datasheet (Sandvik, 2015).

3. RESULTS AND VALIDATION

Preceding the verification of the numerical data, the position of the bottom side (numbered as 6 and shown in turquois in Figure 2) is diagrammed in Figure 6. Due to the increasing pressure difference across the lamella, the reed valve moves and furthermore bends as it has a fixed bearing. The dashed perpendicular line shows the center of the suction port (point 2 in Figure 2). The time difference between the drawn curves is 50 time steps or 6.4° with respect to the crank angle (CA). While the upper four resemble convex bending functions, the lower two show a concave behavior due to the limiter that stops at 2.5 mm. As state-of-the-art valve dynamics measurements only capture points by design there is literally no data published to validate against. At our facilities at the Technische Universität Dresden currently a new measuring principle is evaluated, that enables to track the movement of a laser line on a moving object. Since the test rig dedicated to validate this computational data is still in production, only data acquired in preliminary tests is available at the moment. Figure 7 shows a provisional setup, where the lift of a discharge valve without a limiter was recorded. Nevertheless the measured data fits the computed ones well.

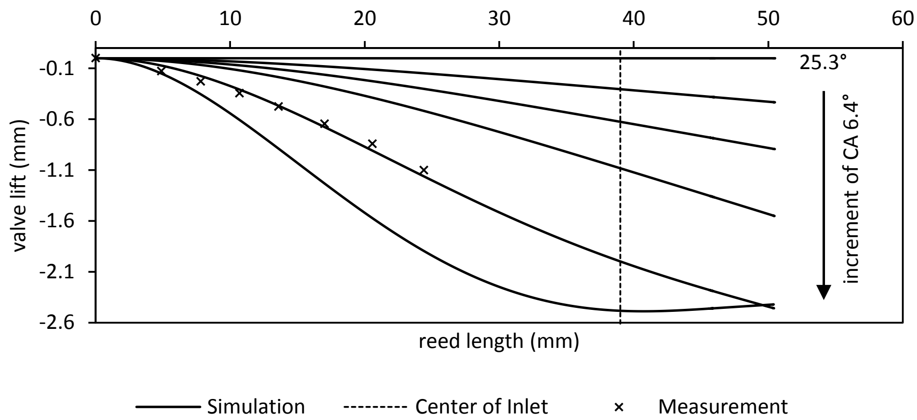


Figure 6: Valve lift vs. reed length

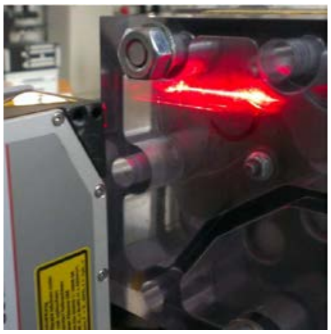


Figure 7: Application of a laser profile scanner for valve lift measurements

As the movement of the reed valve appears to be as indicated by our experiments, a more detailed analysis can be conducted. (Böswirth, 1994) presented the flow coefficient C_{D0} as a function of the ratio of valve lift to bore radius. Furthermore, he showed one function and a straight slope for approximation. Figure 8 shows the recreation of his chart extended to the data computed. As depicted in Figure 9 the deviation is sufficiently small, with a maximum of -0.37 %.

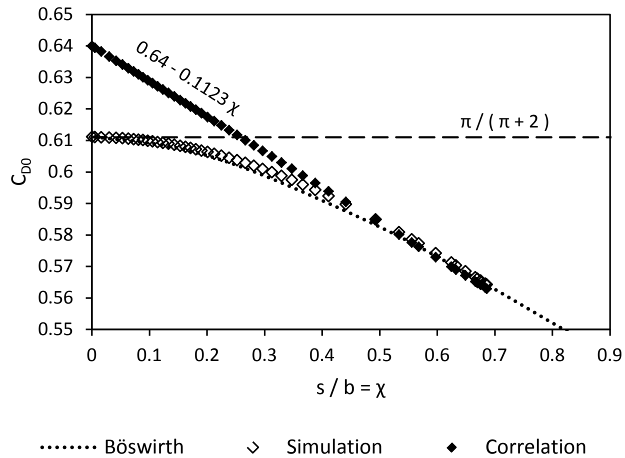


Figure 8: Flow coefficient vs. ratio of valve lift to bore radius (recreation of (Böswirth, 1994))

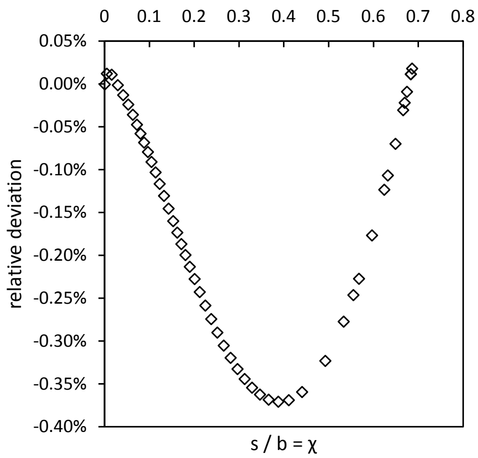


Figure 9: Relative deviation of C_{D0} computation vs. results presented in (Böswirth, 1994)

For further analysis, the steady state model implemented in KV-DYN will be used (Böswirth, 1994). This software enables the computation of valve dynamics according to the basic valve theory. The valve input parameters used are listed in Table 2. It appears that the coefficients are of a linear kind, even though Figure 8 showed that the flow coefficient C_{D0} for instance is a function of valve lift with respect to the time. Nonetheless the zero and one-dimensional computation is still state-of-the-art and widespread in the field of compressor design. The first nine parameters are pure hardware based and resemble the input data for ANSYS. The latter seven coefficients can be determined by rules of thumb, correlations or user experience. Applying these below-mentioned parameters yields to curves for valve lift, acceleration as well as the corresponding pressure difference as a function of the crank angle.

Table 2: Input parameters for KV-DYN

Symbol	Parameter	Value	Unit
b	bore diameter	7	mm
A_P	port area	38.48	mm²
L	circumference	21.99	mm
l	thickness of valve plate	4	mm
s_{max}	limiter position	2.50	mm
C	spring constant	1165.8	N/m
X_v	pretension	0	mm
m	mass	2.32	g
f_0	resonance frequency	112.90	Hz
Phi	angle for oil sticktion	0	°
PF	pocket factor	1	-
C_G	coefficient for gas spring effect	2	-
J	coefficient for gas inertia	3.94	-
C_D	flow coefficient	0.6	-
c_p	force coefficient the valve plate	1	-
d_Z	damping of the valve plate	0	-

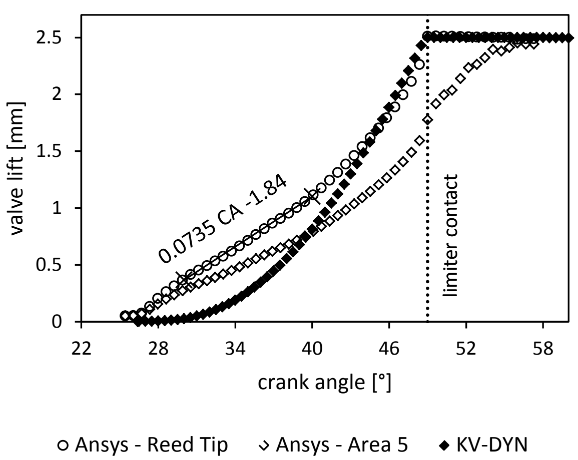


Figure 10: Comparison of simulated data – valve lift

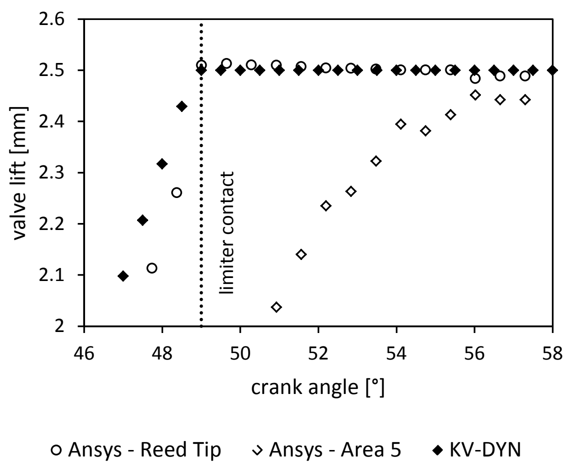


Figure 11: Detail of Figure 10 focusing on the limiter contact

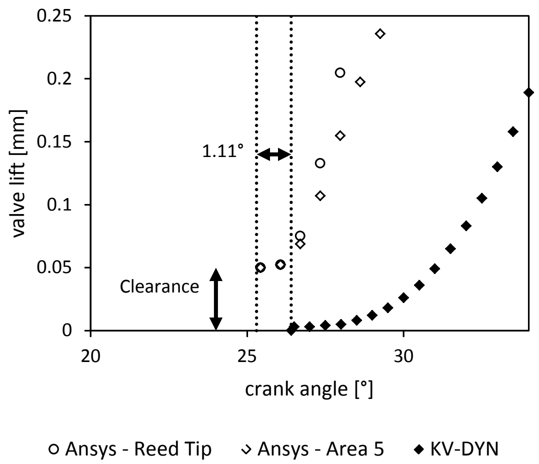


Figure 12: Detail of Figure 10 focusing on the lift-off, also depicted: clearance for mesh displacement and delay angle

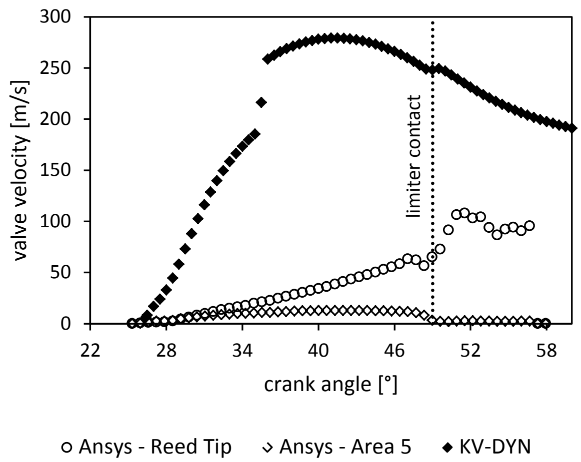


Figure 13: Comparison of simulated data – valve velocity

Figure 10 shows the simulation data for the valve lift created with both ANSYS CFX and KV-DYN. The first part shows the trend that is described by the reed tip movement. Secondly the movement of the valve can be observed analog to a simple valve plate. Therefore one should track a sample area approximately the size of the suction port. In this case the red surface labeled 5 in Figure 2 has been used. As it bends while moving, an average was used for the charts. While the movement of this duct-sized area was mainly introduced for information purposes, Figure 11 shows an oscillation close to the limiter between 54 ° and 57 °. Additionally as the time base was not the same, the limiter contact at 49 ° was used for synchronization purposes.
The curves of the reed tip and KV-DYN show a similar trend. However, they differ in acceleration during the opening process as shown in Figure 12. The valve lift-off in the ANSYS simulation starts approximately 1.1 ° in advance to KV-DYN, accelerates faster in the beginning, forms an almost linear zone from 30 ° to 40 ° and then speeds up again until it hits the limiter. At first glance this behavior conflicts with the often used and published curves (Aigner, Meyer, & Steinrück, 2013; Flade, 2006; Pereira, Deschamps, & Ribas, 2008). However (Flade, 2006) compared his data to other methods of calculation. The valve lift function of the 2D CFD computation showed the smallest gradient. Nonetheless, each model in his comparison has a similar slope. One possible explanation for this difference can be found in the quasi-stationary basis of KV-DYN. The emerging pressure gradients within each chamber caused by the prompt opening process require an accurate resolution in time and space. With the application of linear coefficients this accuracy cannot be achieved. Instead a valve lift dependency should be implemented which results in a higher computation time. By the utilization of higher mathematical method orders as realized in ANSYS, a proper time resolution is ensured due to the complexity of the simulation setup. This complexity results from the parallel solving of multiple mesh displacements, shifts and deformations, thermodynamic states as well as structural strains and stresses.

In contradiction to the state-of-the-art mathematical models a two-way fluid-structure-interaction was numerically solved in this paper. Therefore data describing stresses within the reed valve is available. Figure 14 depicts the case of a fully opened lamella that has hit the limiter as well as the corresponding stresses. Figure 15 presents the data as a function of time as well as the reed length. Thus it appears that there are two main relaxation areas. The first is at the limiter contact position while the second one shifts towards the mounting of the lamella as the forces increase. For a clear presentation of the loads the von-Mises stress criterion will be applied, given that the multiaxial stress state is reduced to an uniaxial equivalent stress. Thus the computed data has to be compared against the yield strength given in (Sandvik, 2015), namely 1450 MPa with a manufacturing tolerance of 60 MPa. As the peak value is approximately 495 MPa a safety factor of about 2.81 to 3.05, depending on the tolerance, can be calculated.

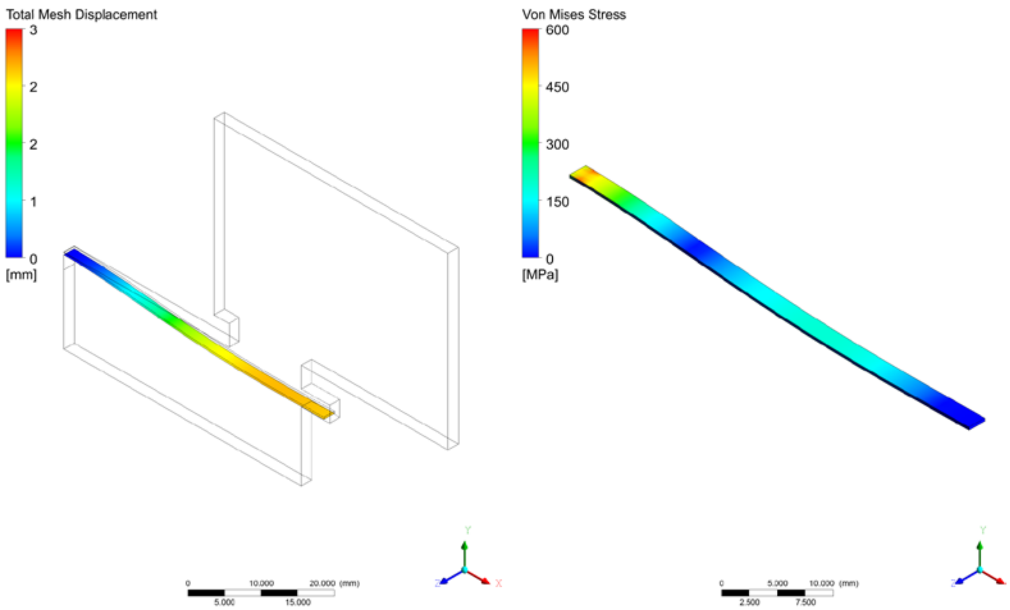


Figure 14: Parallel display of displaced reed valve and corresponding stresses

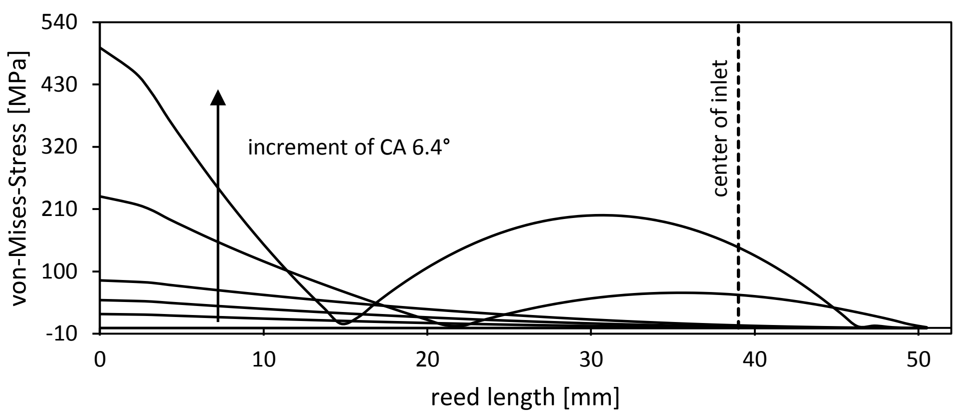


Figure 15: von-Mises-Stresses vs. reed length

4. CONCLUSIONS

Within this paper a two-dimensional two-way fluid-structure-interaction model has been presented. The computed data was then validated against literature as well as established methods. Further it could be shown that there was a fair agreement between the valve lift curves though the quasi-stationary evaluation lacks the transient resolution of an oscillation process of a reed valve. Although the bare simulation time of the data presented here was approximately 07:22 hours on an Intel® Core™ i5-2500 CPU @ 3.3 GHz with 16 GB of RAM, it enables the analysis of the entire lamella starting with the tip. Information like impact velocities in combination with different designs of seats for chamfer studies come to mind as well as the later inclusion of oil fractions in the refrigerant. Nonetheless, additional measurements should be conducted verifying the simulated data and possibly contribute in improvements regarding the lower dimensional computation methods.

REFERENCES

Aigner, R., Meyer, G., & Steinrück, H. (2013). Ventildynamik und interne Wellen in Kolbenkompressoren. *Workshop Kolbenverdichter*, *17*, 159–184.

Böswirth, L. (1994). *Strömung und Ventilplattenbewegung in Kolbenverdichterventilen*. Mödling: Eigenverlag.

Chai, G. C., Zetterholm, G., & Walden, B. (2004). Flapper Valve Steels With High Performance. *International Compressor Engineering Conference*, 8.

Costagliola, M. (1949). *Dynamics of a reed type valve*. Massachusetts Institute of Technology. Retrieved from http://dspace.mit.edu/handle/1721.1/12752

Flade, G. (2006). *Weiterentwicklung der Berechnungsmethoden für Kolbenverdichterventile auf der Basis zweidimensionaler Strömungssimulation*. Technische Universität Dresden.

MacLaren, J. F. T., & Kerr, S. V. (1969). An analytical and experimental study of self-acting valves in a reciprocating air compressor. *ARCHIVE: Proceedings of the Institution of Mechanical Engineers, Conference Proceedings 1964-1970 (vols 178-184), Various Titles Labelled Volumes A to S*, *184*(318), 24–33. http://doi.org/10.1243/PIME_CONF_1969_184_515_02

Möhl, C. (2014). A Comparative Analysis of Numerical Simulation Approaches for Reciprocating Compressors.

Möhl, C., Langebach, R., & Hesse, U. (2014). Numerische Simulation eines Hubkolbenverdichters unter Berücksichtigung der Ventilbewegung. *Kälte-, Luft- Und Klimatechnik*, 28–35. Retrieved from http://www.ki-portal.de/12305/numerische-simulation-eines-hubkolbenverdichters-unter-berucksichtigung-der-ventilbewegung/

Pereira, E. L. L., Deschamps, C. J., & Ribas, F. A. (2008). Performance analysis of reciprocating compressors through computational fluid dynamics. *Proceedings of the Institution of Mechanical Engineers Part E Journal of Process Mechanical Engineering*, *222*(4), 183–192. http://doi.org/10.1243/09544089JPME194

Rodrigues, T. T. (2014). ANSYS Mechanical Coupling Control in FSI Simulations to Reduce Computational Time. In *ESSS Conference & ANSYS Users Meeting*. Retrieved from http://www.ansys.com/staticassets/ANSYS/staticassets/resourcelibrary/presentation/brazil-2014ugm-mechanical-coupling-control-in-fsi-simulations.pdf

Sandvik. (2015). Sandvik 7C28Mo2 Compressor Valve Steel. Retrieved from http://smt.sandvik.com/en/materials-center/material-datasheets/strip-steel/sandvik-7c27mo2-flapper-valve-steel/

Touber, S. (1976). *A contribution to the improvement of compressor valve design*. TU Delft. Retrieved from http://repository.tudelft.nl/view/ir/uuid%3A4c12fa64-32c6-4fd5-9ebc-e733621c96ac/