Technical Paper

Session: 42-3

Session Name: EFRC

Internal flow in reciprocating compressors

Author:

Thomas Müllner
Research Assistant until 3/2014
TU Wien, Institute for Fluid Mechanics and Heat Transfer
1060 Vienna, Austria
currently: Robert Bosch
5400 Hallein, Austria

Co-Author 1:

Bernhard Streibl
CEO
Dr S^3 Simulation
und Strömungsberechnung
1110 Vienna, Austria

Co-Author 2:

Herbert Steinrück
Associate Professor
Institute for Fluid Mechanics and Heat Transfer
TU Wien
1060 Vienna, Austria

Summary

A key factor for the performance of a reciprocating compressor is the gas change. To achieve a good design, the physics of the flow in and out of the compressor have to be understood. In principle, this can be investigated by using commercial CFD software. However, for many applications simpler models turn out to be sufficient. For the prediction of the valve dynamics of compressors of barrel design, a one or in the case of multi- valve compressors a two-dimensional flow modeling is appropriate and have been implemented in the stand-alone tools Compressor1D and Compressor2D, respectively. Concerning the heat transfer from the gas in the compressor to the surrounding walls, these models are of limited use. During in- and outflow the heat transfer could be represented reasonably well since the characteristic flow velocity is given by the in- or outgoing mass flow. During expansion and compression, there is no obvious characteristic reference velocity for the internal flow. Thus, a three-dimensional flow simulation is required. As a consequence, a solver for the three-dimensional Euler-equations has been developed in a project of the EFRC R&D working group. The basic idea is that the heat transfer does not influence the flow and thermodynamic state of the gas in the compressor significantly. Thus, the flow and heat transfer problem can be decoupled. After the inviscid flow in the compressor is determined the heat transfer problem is solved using the boundary-layer approximation along the cylinder walls. For the flow calculation, the flow domain is split into parts; the compression chamber and the valve pockets. Due to the different topology of these domains different meshing strategies have been applied. In the cylinder, a structured mesh with dynamic layering accounting for the piston motion is used. In the valve pockets, an unstructured tetrahedral mesh is employed. The connection of both meshes has been implemented in a conservative way by constructing a common partition of the interface between the compression chamber and the valve pocket. The valve model is based on Costagliola [4]. For the easy use of the program, the user has to specify the main dimensions of the compressor in a graphical user interface. To specify the geometry of the compressor, a sample mesh for a valve pocket is provided. The user has to specify the actual dimensions of the valve pocket and define the direction of the valves in the plane of the cylinder head. The internal flow patterns and valve dynamics for selected compressors will be shown.

Introduction

In a reciprocating compressor, gas is compressed from a low pressure (suction pressure) to a high pressure (discharge pressure). Thus, a certain amount of mechanical work has to be supplied which can be easily calculated by simple quasi-static thermodynamic considerations. However, in practice losses of many different kinds (valve losses, pocket losses, inertia effects, piston masking, and others) occur, and deteriorate the theoretical limit. As a matter of fact, most of the losses can be attributed to the flow of the compressed gas through the compressor and models have been derived to estimate their effect, see e.g. E. Machu [6]. On the other hand, the availability of CFD-software makes it possible to simulate the flow and its interaction with the valves, c.f. Steinrück et al. [9]. It still takes a lot of effort to set up a simulation of a reciprocating compressor including the valve dynamics. In particular, the definition of the geometry, and the meshing of the valve pockets and the definition of the boundary conditions describing the passive valves is tedious. Moreover, the resolution and the prediction of the flow field is far from being trivial. Getting the flow structure at high Reynolds number right is even impossible. We refer to the well known driven cavity flow problem, wherein the 2D case an uncontroversial numerical solution exists only for Reynolds numbers up to about 20000. In reciprocating compressors, the Reynolds number is of the order 10^5 -10^6 . Thus, we can only expect to get the basic flow features correctly. Another issue is the long computation time which is usually a matter of hours or days for a complete cycle. Thus, there was the demand for simple, fast and reliable compressor models. To answer this need, the EFRC R&D working group funded projects to develop the simplified simulation tools Compressor1D, Compressor2D, see Aigner [1], and Compressor3D, see Müllner [7]. Since the one- and two- dimensional models have been presented at previous EFRC conferences, Aigner et al. [2], [3], we want to present here the main features of a simplified three-dimensional compressor model.

The main goals are to predict the valve lift dynamics (valve lift as a function of time), the indicator diagram, the time-dependent forces and moments on the piston, and the heat transfer to the casing.

Model assumptions

The main objective of the flow modeling of the gas in a reciprocating compressor is to predict the response of the gas in the compression chamber to the action of the piston and the valves. In particular, one is interested in internal waves as far as the valve dynamics and the time varying loads on the piston are concerned and in the global flow patterns to get an estimate of the heat transfer. Using a better geometrical model, say a 3D model instead of a 1D model does not necessarily improve the quality of the numerical solution. For example, consider a compressor of barrel design with only one suction and one discharge valve. The motion of the valve plate is affected by internal pressure waves running back and forth between the discharge and suction valve. It was shown that these waves could be well described by a one-dimensional wave propagation/flow model, see Aigner [1]. In a one-dimensional numerical model, the number of grid points can be chosen sufficiently large to compensate numerical diffusion and thus the numerical results are in reasonable agreement with measurements. In a 3D-dimensional approach, one is limited to a relatively large cell size to keep the computation within reasonable time limits. Thus, waves will be strongly damped by numerical diffusion. However, the benefit of a 3D-solution is that one gets the global flow pattern right. This is essential when one is interested in the heat transfer from the compressed gas to the surrounding walls. Thus, selecting the numerical model depends on the question one wants to answer. Of course, most of these problems can be overcome by using high-performance computing as done when simulating the flow and combustion process in internal combustion engines. So the definition of the problem was to develop simple tools to predict the internal flow and heat transfer in reciprocating compressors, which can assist the design of such machines.

Flow model

The gas flow is described by balance equations for mass, momentum, and energy written in conservative form. Neglecting viscosity, these are the well-known Euler equations. As a consequence, heat conduction and thus, the heat transfer to the walls of the compression chamber is also eliminated.

But this contradiction can be easily resolved by the fact that along the solid walls thin boundary layers form. Thus, the heat transfer can be calculated after the inviscid flow calculation a boundary layer calculation. The Euler equations have to be complemented by equations of state for the gas. Here, an ideal gas with constant heat capacities is assumed. It is described by $\varkappa$, the ratio of the specific heats and the molar mass. The numerical method for the solution is a finite volume method. Thus, the computational domain is divided into small finite volumes, and the mean values of the state variables (density, momentum, and energy) of each volume are used as computational quantities. Due to the conservative formulation, these mean values can be changed only by the flux of these quantities across the volume boundaries. Therefore, the mass, momentum, and energy conservation principle are strictly satisfied even for the discretized equations. In a numerical scheme, the fluxes across a cell boundary have to be computed as a function of the cell averages of the state quantities of the neighboring cells. Here, the Roe-method is used. It calculates the numerical flux from the cell average of the adjacent cells using an approximate Riemann solver. This is a robust, well-proven method of first order accuracy.

Description of the geometry

The flow region inside the compressor consists of the cylinder confined on one end by the movable piston and on the other side by the cylinder head, and the valve pockets, see figure 1. In the cylinder, a structured mesh for the finite volume method is used. It is given by a two-dimensional mesh in the plane of the cylinder head, see figure 3a), and is extruded in the direction of the cylinder axis. Due to the motion of the piston, these finite volumes are stretched in the axial direction. If the aspect ratio of the cells exceeds a critical value, a re-meshing in the axial direction will be performed and the cell averaged values of the density, momentum and energy have to be interpolated in a conservative way for the new mesh, see figure 2.

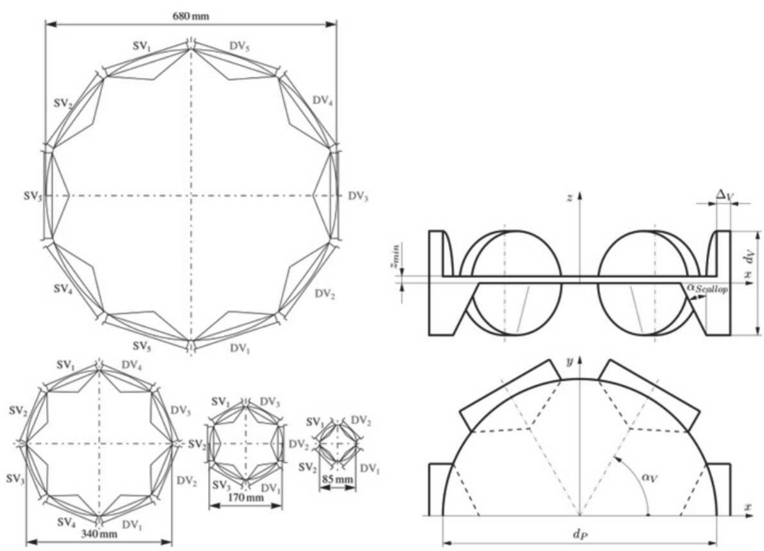

Figure 1: Compressor geometry

For the valve pockets, a reference geometry has been chosen. It consists of a cylinder of diameter d_V, which intersects the cylinder of the compression chamber, and a cone of height h_C. The axis of both, the cylinder and the cone are in the plane of the cylinder head. Due to its complex geometry, it is not possible to mesh it with a structured mesh. A prototype unstructured tetrahedral mesh was constructed for the valve pocket, see figure 3b). The prototype mesh will be mapped onto the actual valve pocket. Finally, the geometry of a compressor can be described by a few parameters: bore diameter, stroke, clearance, the number of valves, the direction of the valve pocket axis, the radius of the valve pocket, heights of valve-pocket cone and cylinder. At the interface of the valve pocket and the cylinder, the unstructured tetrahedral grid and the hexahedral grid in the cylinder have to be coupled in a conservative way. Therefore, a common partition of both grids at the common interfaces at the cylinder head and the circumference of the cylinder are constructed. In Fig 3c) the common interface with the quadrilateral surface mesh of the cylinder and triangular surface mesh of the pocket valve is shown.

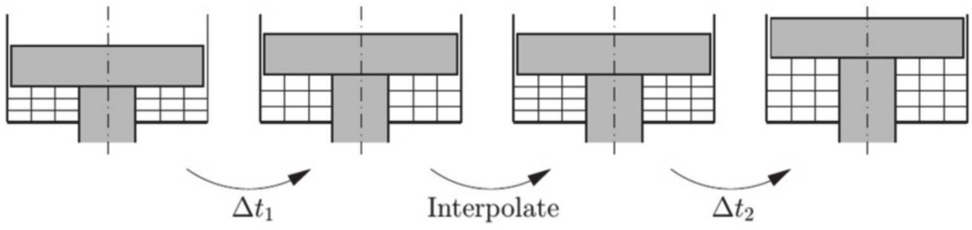

Figure 2: Dynamic mesh

Hence, in order to derive the interface facets, all the triangles of the valve pocket surface mesh and all the quadrilaterals of the cylinder surface mesh are decomposed into a priori unknown number of polygons, being the result of that intersection. Every single polygon is then part of a triangle and a quadrilateral, respectively. Recently, Fahs [5] identified five possible configurations for the intersection of triangles with convex quadrilaterals, giving polygons with either three, four, five, six or seven edges. Once the number of edges and the positions of the intersection points of every polygon is known, the area of every polygon can be computed easily by decomposing it into single triangles. It is also not difficult to determine the barycenter of the polygons, eventually required for a second order finite volume method. However, for a first order finite volume method, the numerical flux computation only requires the area and the unit normal vector, here taken in the area center.

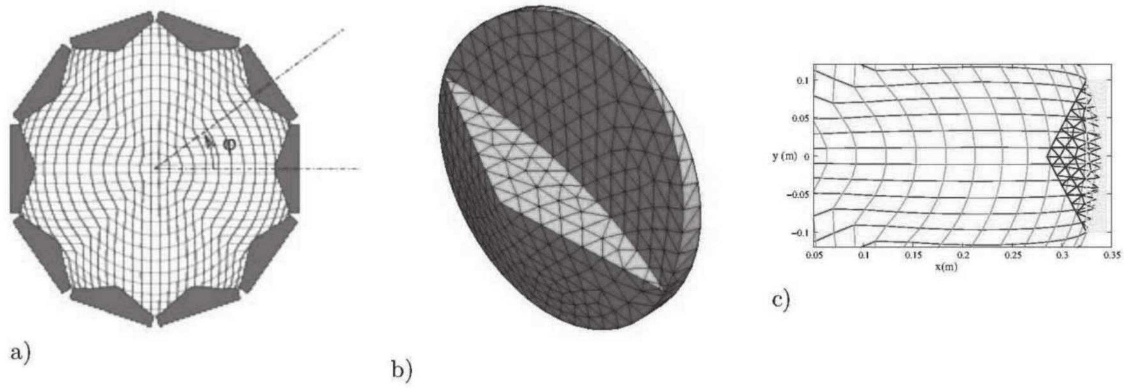

Figure 3: a) planar structured grid in plane of cylinder head, b) tetrahedral mesh in valve pocket, c) common partition of interface of structured and unstructured grid

Valve model

The discharge and suction valves are self-acting, spring loaded plate valves. For all three compressor simulation tools, the same valve model based on the paper by Costagliola [4] is used. The gas flow through the valve is assumed to be quasi-steady. This assumption is valid if the transit time of a gas particle through the valve is short compared to the opening time of the valve, which is the case under usual operating conditions. Thus, the mass flow through can be expressed by the well-known outflow formula by Saint-Venant Wantzel, see Zierep [10], as a function of the effective flow cross-section Φ_V which is, in turn, a function of the valve plate lift x_V.

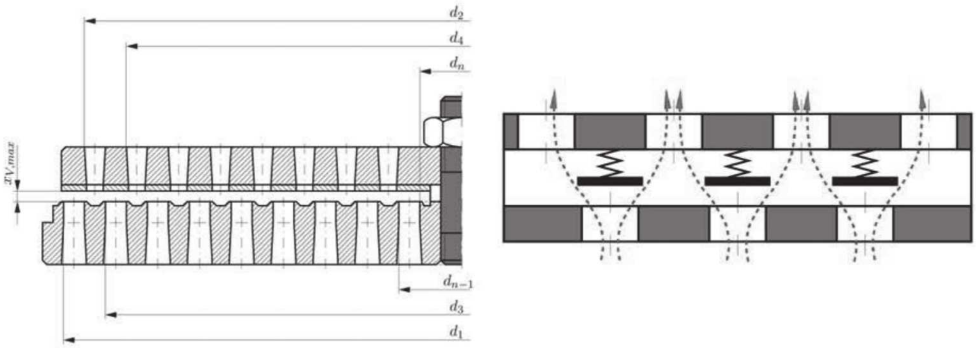

Figure 4: Flow paths through a plate valve (left), mass spring model of plate valve (right)

Moreover, a model for the state of the valve is needed. That is an equation of motion for the valve plate of mass m_V . The gas pressure acts on the so-called force areas A_f on both sides of the valve plate. Springs with spring constant k_V and initial deflection L_V press the valve plate onto the seat and thus define a closing pressure Δp_{closed} , see figure 4.

Heat Transfer model

Since the Reynolds number for the gas flow in a reciprocating compressors is large, we can expect that the flow is almost inviscid in the interior, and turbulent boundary layers form along the walls. In a state of the art commercial CFD software, one would use a more or less elaborate turbulence model to describe the Reynolds stresses in the flow region. Rigid walls suppress the turbulent fluctuations and thus, special wall treatments are required. Very close to the wall viscous stresses dominate, and this region is called viscous sub-layer. In the case of an attached flow, the flow profile has to be matched to the so-called defect layer of the boundary-layer. It has been argued by Prandtl and later be confirmed by measurements that in the overlap region of the defect layer and the viscous wall layer the velocity and temperature profile is given asymptotically by a logarithmic profile.

Thus, there is the choice to resolve the viscous sub-layer or to use wall functions to describe the flow and temperature field close to the wall.

In our approach, the external flow field is given by the numerical solution of the Euler-equations. Thus, it remains to solve the boundary-layer equations to obtain the heat transfer coefficients. Since the Mach number of the gas flow near the wall is small, we can consider the boundary-layer flow as incompressible. We remark that in the passage between the valve pocket and the cylinder the flow velocity may become supersonic. But this is not the case for the flow along the walls.

For the solution of the boundary-layer equations several methods are possible: using turbulence models plus wall treatment or integral methods are among them. Since we are interested only in an overall heat transfer coefficient for each face, we will use a very simple integral method. Following Gersten Schlichting [8], we will use logarithmic velocity and temperature profiles close to the wall. With these assumptions, we derive the relation for the average heat transfer coefficient α

$$\alpha = \rho c_p \bar{u} \frac{1}{Pr_t} \left(\frac{\kappa G(\Lambda, D)}{\ln \mathrm{Re}} \right)^2 ,$$

where u is the mean velocity of the Euler flow along the considered face and the Reynolds number Re is formed with this velocity and the piston diameter. The value of the turbulent Prandtl number Pr_t is 0.9 and of the von Karman constant κ is 0.4. ρ and c_p denote the density and isobaric heat capacity of the gas, respectively. The function $G(\Lambda, D)$ is approximated by 1.5, see Gersten Schlichting [8].

Results

The program has been tested for four test cases shown in figure 1. The data for the 10-valve compressor is listed in table 1 and will be discussed in detail. In Figure 5 the maximal Mach-number in the valve pocket is shown as a function of the crank angle for the first two cycles of the simulation. We remark that the first cycle depends weakly on the initial condition of the simulation. Time-periodic behavior is obtained after the first cycle. During outflow (at a crank angle of 320°) the Mach number in the discharge pocket rises up to 0.75. The flow stays sub-sonic. Only when increasing the compressor speed to 1200 rpm supersonic flow conditions occur for a short period, and the present code is capable of dealing with it.

piston diameter	680mm	# suc. valves	5	valve radius	100 mm
min. gap width	1.5mm	# disch. valves	5	height of valve cone	40 mm
stroke	150 mm	α	2.0	height of valve pocket	19 mm
speed	800 rpm	β	$1.8 \cdot 10^5 \mathrm{m}^{-2}$	max valve lift	2.5 mm
Molar mass	29 kg/kmol	force area	$0.0178\,\mathrm{m}^2$	mass valve plate	0.21 kg
ratio of spec. heats	1.4	spring stiff.	3750 N/m	$fe_{1\mathrm{mm}}$	5.938 m
kinematic vis.	$2 \cdot 10^{-5}\,\mathrm{m}^2/\mathrm{s}$	initial deflection	0.75 mm		
suc. press.	1 bar	suct. density	$1\,\mathrm{kg/m}^3$	disch. press.	1 bar

Table 1: data of test case

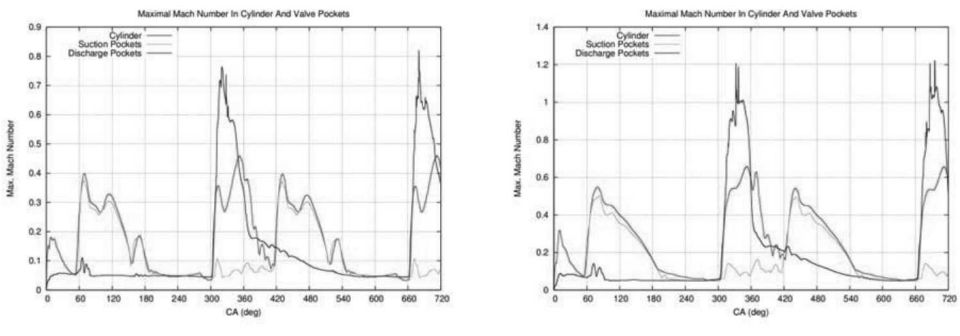

Figure 5: Maximal Mach number for 800 rpm (left) and 1200 rpm (right)

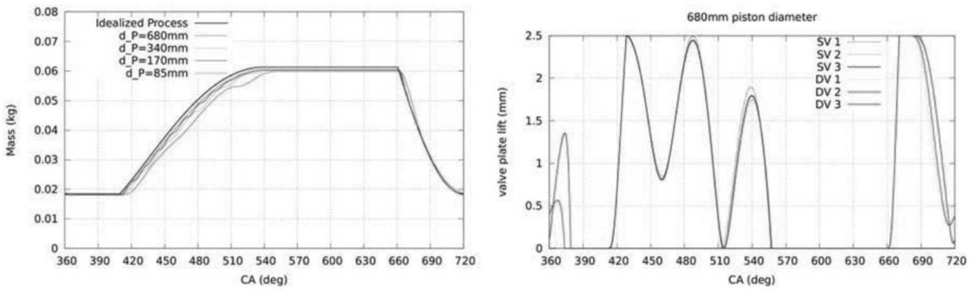

Figure 6: Valve dynamics: mass flow during discharge for all four test cases compared with the ideal compressor process (left), valve lift of the suction and discharge valves of the 10-valve compressor (right)

The operating point of the four test cases has been chosen such that they can be considered as different stages. Thus, they refer to the same idealized compressor cycle. The total mass flow through the discharge valves shown in figure 6a) is close to the idealized compressor process. However, it can be seen that the compressors with piston diameter 340 mm, 170 mm and 85 mm show oscillations in the mass flow. This is due to fluttering of the suction and discharge valves. By changing the springing, the eigenfrequency of the spring-mass system of the plate valve can be decreased to avoid fluttering, see Müllner [7].

For the 10-valve compressor, the suction valves do not stay fully open during intake. They are completely opened only for two short moments. They even close for a moment at 515 deg crank angle. The discharge valves behave well. It is interesting to note that the central discharge valve DV_3 opens again after the piston has reached the dead center. For the lateral valves DV_2 and DV_1, this secondary opening is less pronounced.

Calculating the average velocity on each face of the compression chamber and using the simplified boundary-layer heat transfer model the heat transfer coefficients on the different faces of the compression chamber are estimated. As expected, the heat transfer is large during the discharge between 300 and 400 deg. During intake, it is only of moderate size while it is almost negligible during compression, see figure 7b).

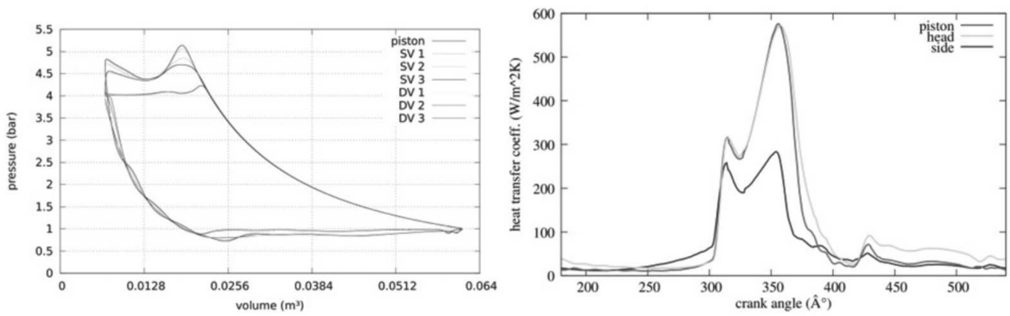

Figure 7: Indicator diagram (left), heat transfer coefficients (right)

In the indicator diagram, figure 7, the pressure at the piston, the suction valve, and the discharge valve is shown. It can be clearly seen that the pressure at the suction valves is markedly larger than the pressure at the piston. In the valve pockets of the discharge valves, the pressure is only slightly above the discharge pressure. In figure 8 a)-c), the pressure at the piston is shown shortly before and after the opening of the discharge valves. At CA=660 deg the pressure is almost uniform. At the interfaces to the valve pockets, the pressure is slightly smaller. Shortly, after the discharge valves open at CA=662 deg, the development of a pressure wave traveling transversally through the cylinder can be observed.

During re-expansion internal waves can be observed in the indicator diagram and at the snapshots of the pressure field at the piston, figure 8 d)-f). An estimate of the time needed for traveling back and forth in the cylinder t = 2d/c gives a crank angle difference of about 20 deg. for the complete passage back and forth in the cylinder. Indeed, the pressure snapshots between 380 and 390 deg show the wave has once passed the cylinder.

The velocity field along the piston is shown in figure 9. At a crank angle of 360 deg the piston is at the dead center. At 370 deg the re-expansion has already started, and the gas flows from the valve pockets back into the cylinder. Since the pressure in the suction valve pockets has been larger than that in the discharge valve pockets the flow velocity along the piston at the suction side is considerably larger than at the discharge side. During intake (450 deg) we observe the inflow from the suction side. At the dead center (540 deg) only relics from the intake motion are visible. During compression, there is a slow redistribution of the gas into the valve pockets visible. When the discharge valves open at 670 deg the outflow motion into the discharge valve pockets sets in.

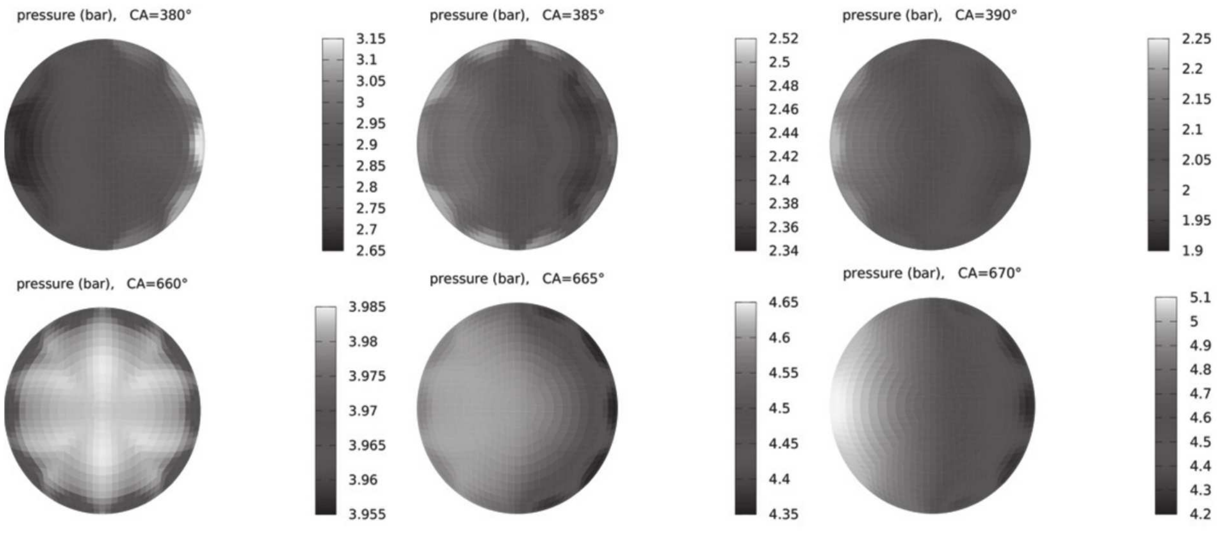

Figure 8: Pressure at the piston

Comparison with Compressor1D

In Compressor1D, the Euler-equations are averaged over the cross section of the compressor perpendicular to the diameter between the two valves. Thus, a wave propagation problem through a channel with varying cross section has to be solved. It has been shown that the restriction to a one-dimensional flow is reasonable during the interesting phases of the compressor cycle. However, due to the nature of the one-dimensional flow model the passage of the flow from the cylinder into the valve pocket cannot be accurately described due to the discontinuous change of the flow cross-section at the interface between the cylinder and the valve pocket. In the following, we compare simulation results of Compressor1D with Compressor3D for a 2-valve compressor with a 220 mm bore. In figure 10a), the motion of the valve plate during the opening of the discharge valve is shown. The one- dimensional model predicts a higher impact velocity of the valve plate. This can be attributed to a larger pressure loss at the interface of the valve pocket to the cylinder than in the three-dimensional case.

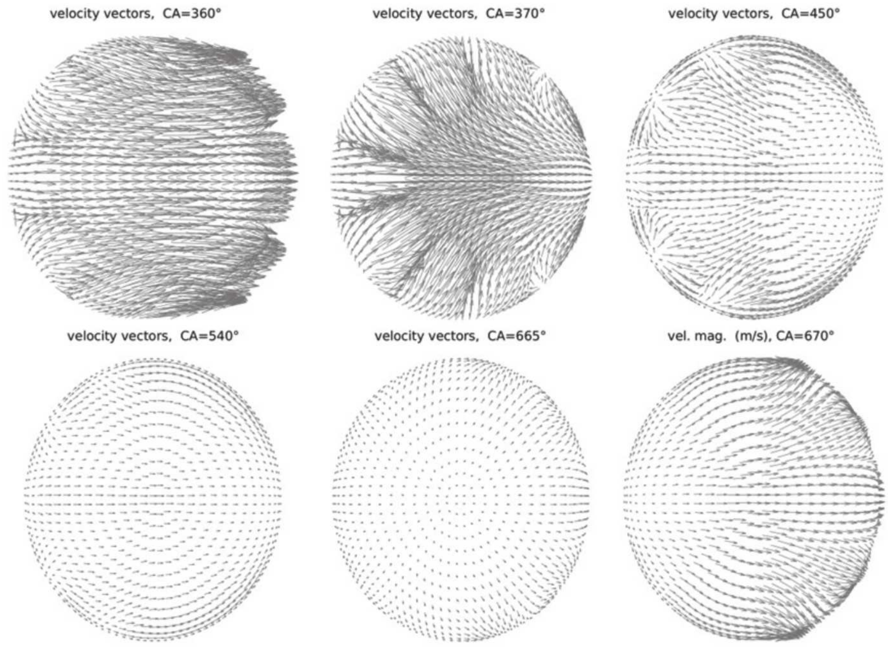

Figure 9: Velocity field along piston

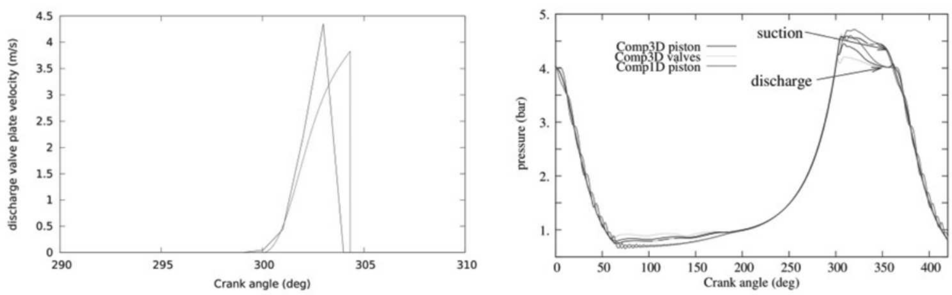

Figure 10: Comparison Compressor1d/Compressor3D for a 2-Valve compressor: plate velocity of discharge valve (left), pressure at different positions in the compressor (right)

In figure 10b), the pressure at different positions as a function of the crank angle is shown. In the three-dimensional case, the mean pressure at the suction and discharge valve is indicated with a cyan-colored line. The blue lines correspond to the pressures at the piston close to the valve pockets. At the beginning of the discharge, a pressure difference across the discharge valve pocket can be observed. This difference decreases until the end of discharge. In the one-dimensional model, this pressure difference is obviously larger. On the suction side, there is no pressure difference across the valve pocket. The one-dimensional model predicts on the suction side a larger pressure than the three-dimensional model.

During inflow, the 1D model predicts at the piston a lower pressure than the 3D model which can be again attributed to a significant pressure loss across the passage from the valve pocket into the cylinder. Moreover, the one-dimensional model predicts only slightly damped pressure waves running back and forth across the cylinder. These waves are not present in the 3D-dimensional model. On one hand there is no damping mechanism in the one-dimensional model, except numerical diffusion, which is, however, small due to the relatively large number of grid points and on the other hand in the 3D case waves are damped geometrically and by the numerical diffusion due to the relatively small number of cells per diameter to keep the computational time within reasonable limits.

Conclusions

A three-dimensional model for the simulation of reciprocating compressors based on the Euler-equations has been presented. The interaction of the flow with the valve dynamics and the occurrence of internal waves have been shown. Moreover, an educated guess of the heat transfer coefficients has been given, and a comparison with the existing one-dimensional model has been made.

The use of these models is to obtain an educated guess of the compressor performance with a ready-to-use-tool. To study details of the flow and heat transfer, a detailed analysis using a state of the art tool will be unavoidable.

Acknowledgement

The work has been funded by the EFRC-R&D working group.

References

[1] R. Aigner, Internal Flow and Valve Dynamics in a Reciprocating Compressor, Ph.D. thesis, TU Wien, 2007.
[2] R. Aigner, G. Meyer, and H. Steinrück, Valve dynamics and internal waves in reciprocating compressors. In 4th EFRC Conference, pages 169–178, 2005.
[3] R. Aigner and H. Steinrück, Modeling fluid dynamics, heat transfer and valve dynamics and in reciprocating compressors. In 5th EFRC Conference, pages 169–178, 2007.
[4] M. Costagliola, The theory of spring-loaded valves for reciprocating compressors. J. Appl. Mech., pages 415–420, 1950.
[5] H. Fahs, High-order discontinuous Galerkin method for time-domain electromagnetics on non-conforming hybrid meshes. Mathematics and Computers in Simulation, 107:134–156, 2015.
[6] E. Machu, Problems with high-speed short stroke reciprocating compressors: Increased power requirements due to pocket losses, piston masking and gas inertia, eccentric loads on the piston. In Proceedings of Gas Machinery Conference USA, 1998.
[7] T. Müllner, Flow Patterns and Valve Dynamics in Multi-Valve Reciprocating Compressors. Ph.D. thesis, TU Wien, 2015.
[8] K. Schlichting, H. Gersten, Boundary-layer theory. Springer, 8th edition, 2000.
[9] H. Steinrück, R. Aigner, and G. Machu, Transversal waves in a reciprocating compressor. Acta Mechanica, 201(1):231–247, 2008.
[10] J. Zierep, Grundzüge der Strömungslehre. Springer, 1997.