# Technical Paper

## Session: 49-2

## Session Name: Calculation 2

## Interactions between valve flow characteristics, valve pocket geometry and compressor performance

**Author:**

**Dr.-Ing. Reiner Schulz**
**Research and Development**
**Burckhardt Compression AG**
**8404 Winterthur, Switzerland**

**Summary**

The need to reduce overall pressure losses is one of the most important challenges in meeting the increasing demand for high-efficiency reciprocating compressors. Although in most compressor designs valve losses amount to a bit less than half of the overall pressure losses, the contribution of the valve pockets to the overall losses may be higher than the valve losses in some cases. Therefore pocket losses are far from being insignificant. The losses of both valve and valve pocket are dependent on the pocket geometry since the valve pocket affects the flow in the valve and vice versa.

To improve efficiency, both the evenness of flow through the valves as well as the clearance volume must be optimized. Due to the limitations of the mutually exclusive objectives of low clearance volume and evenness of flow in the usual configuration (cone with slot shaped passage), other arrangements are also taken into consideration for comparison. A double acting cylinder equipped with a new design of a poppet valve has been investigated using moving mesh CFD simulations, taking fluid structure interaction into account. To improve the accuracy of compressor performance analysis, including the prediction of absorbed power, correction factors for both valve and pocket pressure loss coefficients have been derived, however, only an illustrative example is presented within this paper.

***Keywords:*** Reciprocating compressor, Valve pocket, Valve losses, Computational Fluid Dynamics (CFD), Moving mesh

## 1. Introduction

Pressure losses in valve pocts can easily be determined for common geometries [1]. They may be expressed by a “pocket factor” based on valve losses. The impact of flow uniformity is neglected. This may be acceptable as long as only valves of similar types (e.g. plate valves of common proportions) are compared. However, the more valve losses are reduced, the more the flow is becoming unevenly distributed - in some cases even giving rise to recirculation through valves. Therefore, if valve types causing different flow distributions have to be compared, the accuracy of the traditional approach is limited. The modern approach based on CFD simulations, as described in many publications [2-9], is computationally too expensive to replace the tools for compressor / valve dimensioning which are based on zero or one- dimensional modeling. Therefore detailed CFD simulations should provide correction factors, which are applied easily to dimensioning tools.

In the first part of the paper steady state flow is assumed throughout. To illustrate the effect of non-uniform flow (or backpressure) on pocket pressure loss, different types of valve pockets have been investigated and the variations of valve pocket shapes are illustrated. The pressure loss of an arbitrary valve is approximated by an axisymmetric distribution of effective flow resistance (or porosity), taking into account non-permeable walls (e.g. centre bolt area). Radial flow in the valve is neglected for simplicity. Pressure loss coefficients for valve and compressor have been determined for different distributions of porosity. Inlet losses, caused by the redirection of flow entering the valve, are an additional important factor (however, they are not considered in this paper). By comparing these loss coefficients to the loss coefficients obtained for even flow (i.e. valve in a wind tunnel), the contribution of the unevenness of the flow to the pressure loss is obtained, i.e. additional loss coefficients due to the interaction of the parts are determined.

Steady state investigations will always be limited due to effects such as:

- Piston masking and squeezing effects near top dead centre;
- Pulsations and effects of gas inertia;
- Detailed flow in valve.

More complex transient flow, fluid structure interaction and moving mesh simulations provide valuable additional information such as gas forces acting on internal valve components and their (in some cases individual) opening and closing times. In the second part of the paper some transient, moving mesh simulations are summarized.

All simulations use air (i.e. ideal gas) as fluid. In order to minimize effects of numerical viscosity, the Monotone Advection and Reconstruction discretisation Scheme [10] is applied throughout. This scheme shows least sensitivity of solution accuracy to mesh structure and skewness. Turbulence is modelled by the standard high Reynolds Number k-ε model with non-equilibrium wall functions.

## 2. Simulations of Steady State Flow

### *2.1 Valve optimisation*

Valves are always subject to improvement; a common task is to reduce pressure losses for a given valve type and valve size. A modified design of a poppet valve had been proposed. The pressure loss of the proposed design was compared to the standard design. Two different arrangements were used as shown in Figure 2.1: The valve in a tube (as in a wind tunnel) and the valve between a valve cage dummy and a generic valve pocket representative of many compressor designs. The comparison of the two different models revealed that although the proposed new design performed more or less identically in a wind tunnel measurement, the performance in a compressor is worse than the standard design. Diagram 2.1 shows the comparison of the loss coefficients (static pressure differential divided by dynamic pressure) of the valves. To obtain more realistic data, all pressure loss assessments of valves are performed both in a generic cylinder and in a wind tunnel arrangement. The results also led to further design modifications and finally to the development of a slightly modified poppet valve with pressure losses reduced to 73% in comparison to the standard design in a valve pocket of a standard compressor (Figure 2.2).


*Figure 2.1: Models used for investigations: Wind tunnel type (top), cylinder with valve pocket (bottom)*

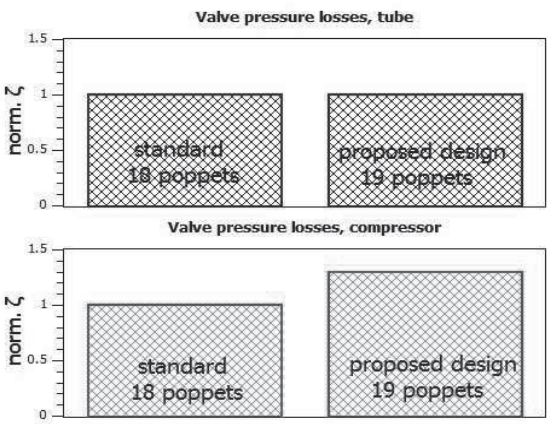


*Diagram 2.1: Comparison of normalized pressure loss coefficient of two poppet valve designs. Loss coefficient of valve placed between valve pocket and valve cage (top) and in a wind tunnel arrangement (bottom)*

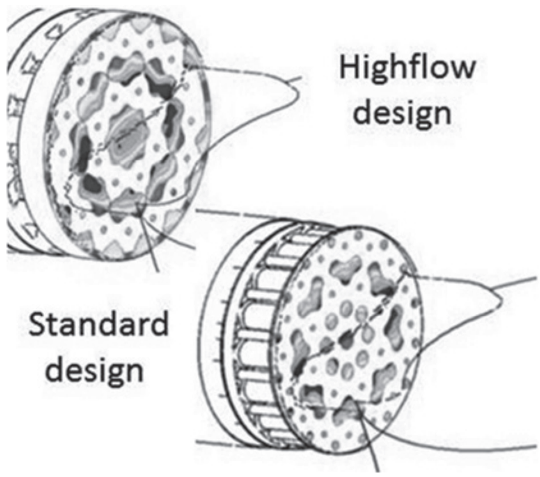


*Figure 2.2: Velocity plots, section at valve outlet. Highflow design with pressure loss reduced to 73 % (top) in comparison to standard execution (bottom)*

### 2.2 Variation of pocket geometry

Systematic series of valve pocket geometry variations have been investigated. The series have been simulated in order to quantify the effects of interdependency between valve pressure loss characteristics (i.e. not only the overall, averaged pressure loss as determined in a wind tunnel arrangement but especially the impact of the distribution of flow resistance) and valve pockets of different shapes. At the moment, the parameter var-iations are limited to geometries with one suction / discharge valve per cylinder. The series uses a conventional arrangement with a right angle between cylinder and valve axis (Figure 2.3). Different shapes of the passage between cylinder and valve pocket are compared.

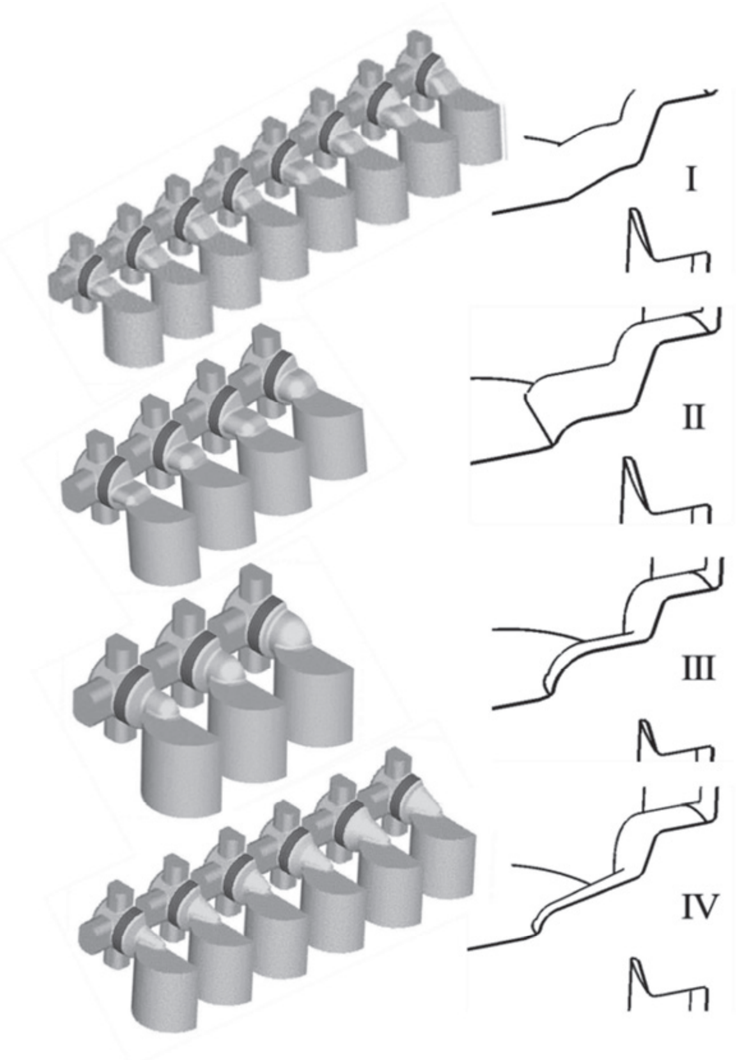


*Figure 2.3: Geometry variations of conventional valve pocket arrangement: Type 1 (combination of wedge and cone), Type II (combination of block and cone), Type III (combination of cylinder and cone) and Type IV (combination of two cones; from top to bottom)*

In Figure 2.4 velocity plots are compared. The shape of the passage between the cylinder and the cone adjacent to the valve is clearly visible as a velocity distribution at the section at the valve inlet. In this simulation, the valve pressure loss amounted to half the pressure loss of the valve pocket. The lower the pressure loss of the valve, the higher becomes the unevenness of the flow. In order to describe the pressure loss coefficients of the generic valve pockets, the following approach will be applied: The pressure loss coefficients are tabulated for each design dependent on several distributions of local pressure resistance of the valve, the shape of the designs are described by parameters of characteristic dimensions (form factors), and the pressure loss coefficient is approximated dependent on these parameters as well as the distribution of the resistance in the valve by least square methods (i.e. curve fitting).

As a result, the pressure loss of the valve pocket can be calculated dependent on dimensions and the pressure loss of the valve and the distribution of flow resistance.

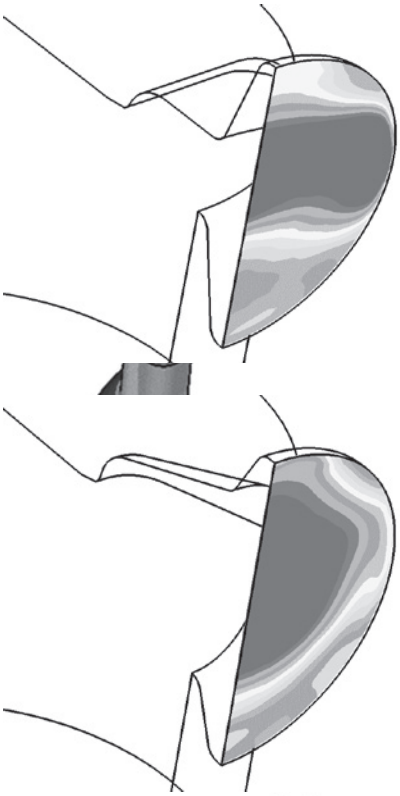

Figure 2.4: Velocity plots of selected valve pockets. Type II (combination of block and cone, top) and Type IV (combination of two cones, from top to bottom)

As the resistance distribution of the valve can be calculated or estimated from the geometry, the impact of different valve types on the main pressure losses in a compressor can be taken into account, e.g. in zero dimensional codes used for compressor dimensioning.

### 2.3 Results: Interaction between pressure losses of valve pocket and valve

Within this paper, the impact of the distribution of flow resistance in the valve on pressure loss coefficients is illustrated for a single design and a small number of idealized distributions only. The valve is approximated by porous cells; within these cells flow is possible only in the direction of the valve axis. . The radial distributions of flow resistance applied are plotted in Diagram 2.2 over the normalized radius.

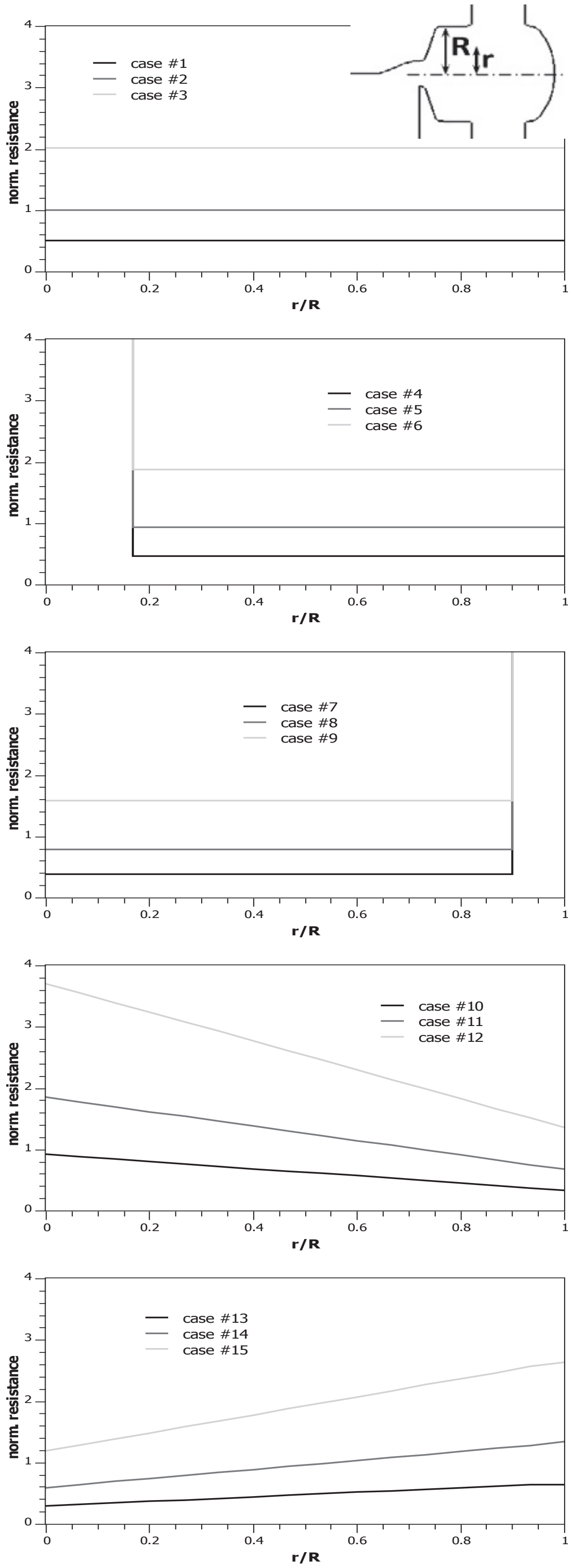


*Diagram 2.2: Radial distributions of flow resistance in valve. Constant resistance, constant with central bolt, constant with rim, linear with high resistance in centre, linear with low resistance in centre (top to bottom)*

Three levels of flow resistance of the valve are applied to an incompressible flow with a Reynolds Number of 5.0E4 (based on nominal valve diameter and averaged velocity). In a wind tunnel arrangement, all the distributions plotted in the same colour in Diagram 2.2 have the same pressure loss of the valve.

However, if the same valve is investigated in a model with cylinder, valve pocket and valve cage (Figure 2.5), the different distributions result in different pressure losses of the whole combination. The pressure losses are usually described as ratio of static pressure differential divided by dynamic pressure (i.e. total minus static pressure); in this way a pressure loss coefficient is obtained for the valve. We define the pressure loss coefficient for steady flow through a valve with equal cross sectional area of the in – and outlet as:

$$\zeta = \frac{\iint \rho v p_s \, dA_{in} - \iint \rho v p_s \, dA_{out}}{\iint \rho v (p_t - p_s) \, dA_{in}}$$

The same approach may be applied to the parts up- and downstream of the valve (another approach is to calculate the ratio of the area of an orifice required for the mass flow under isentropic conditions to the actual section).

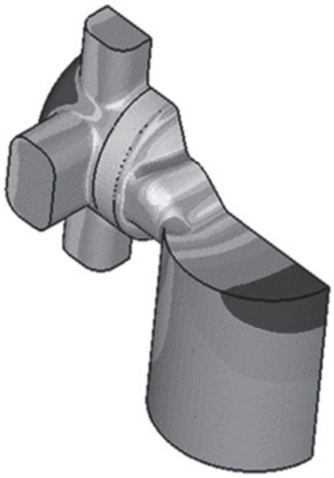

*Figure 2.5 Geometry of example valve pocket*

As mentioned before, the pressure loss coefficients are not only dependent on the geometry, but to a smaller portion also on the combination of valve pocket, valve and valve cage. If we denote the pressure loss coefficient of the valve in a wind tunnel arrangement as $\zeta_{v0}$ and in the valve pocket as $\zeta_{v1}$, the pressure loss coefficient of the pocket with inlet - outlet boundary conditions as $\zeta_{p0}$ and combined with a valve $\zeta_{p1}$, the pressure loss coefficient of the valve cage with inlet - outlet conditions $\zeta_{vc0}$ and combined with a valve $\zeta_{vc1}$, then the effects of a combined pocket – valve – valve cage assembly in comparison to the separate parts on pressure loss may be expressed by a factor

$$f_i = \frac{\zeta_{i1} - \zeta_{i0}}{\zeta_{i0}}, i = v, p, vc$$

In Diagram 2.3, these factors are plotted for the design as shown in Figure 2.5 and the 15 distributions of flow resistance of the valve from Diagram 2.2. As can be seen from Diagram 2.3, these factors may exceed the 10% range for pocket and valve, and hence are not negligible. For the valve cage the effect is much higher, however, the contribution of this component to the overall pressure losses is usually very small. When valves of different types (e.g valves with few straight-through modules or an optimized poppet valve) are interchanged with standard valves of common proportions, or between each other, then the compressor performance may behave in an unexpected manner because the change in valve pocket pressure loss may well overcompensate the nominal performance differential of the valves.

The effects illustrated are negligible when, and only when, similar valves (i.e. plate valves of similar proportions of the central bolt etc.) are compared.

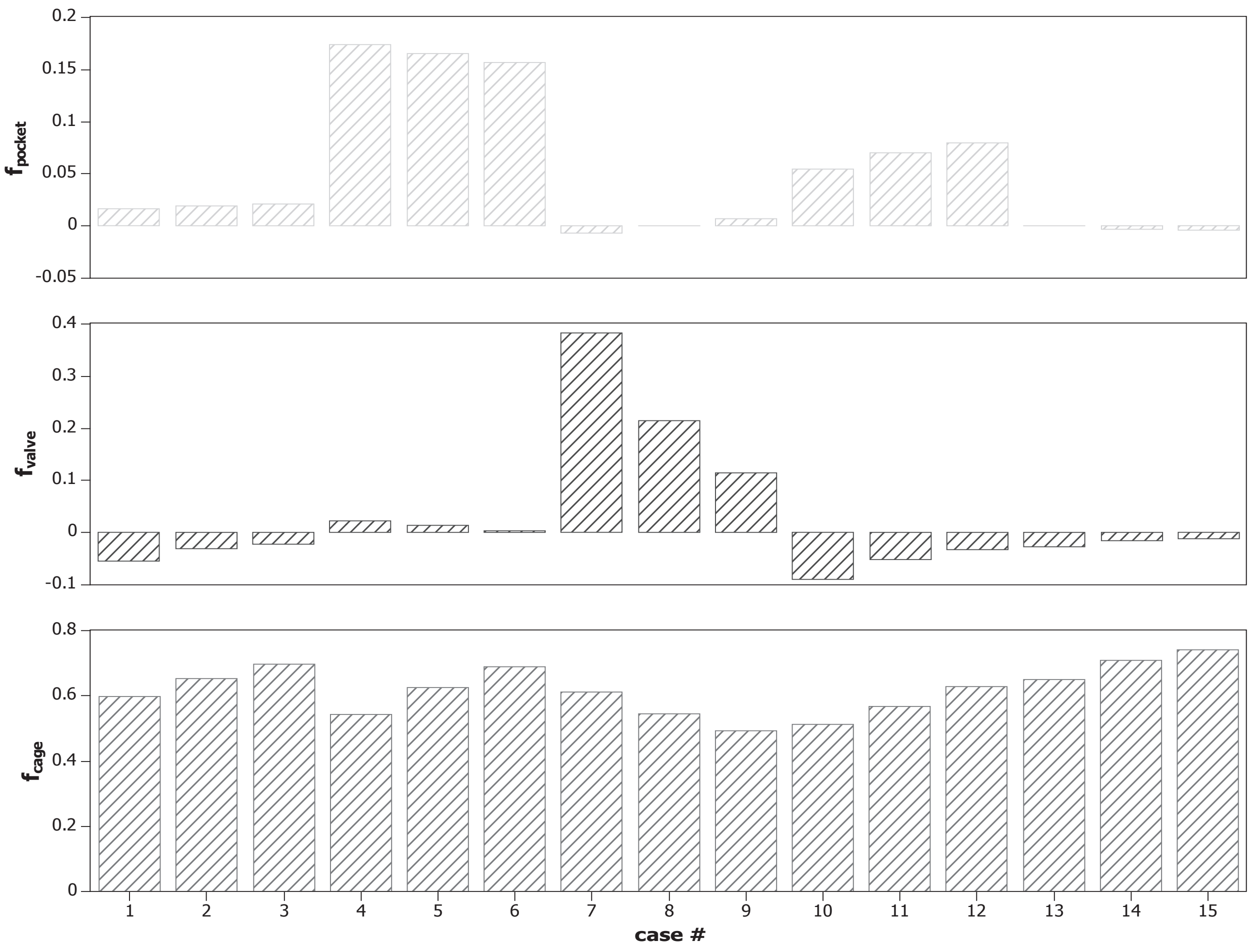


*Diagram 2.3: Pressure loss correction factors caused by the interaction of valve pocket, valve and valve cage for the geometry in Figure 2.5 and the distributions plotted in Diagram 2.2 for pocket, valve and valve cage (from top to bottom)*

## 3. Moving mesh simulations

### *3.1 Moving mesh simulations with prescribed motion*

A simple model of a compressor cylinder is depicted in Figure 3.1 (left), comprising the cylinder, an inlet and discharge valve pocket, two simple valves, valve cages and collectors. A velocity plot is shown in a cut plane, the piston is at the bottom dead centre.

The geometry of the parts located up and downstream of the compressor cylinder is taken into account by boundary conditions derived from a one-dimensional model. In this model, the valves are a crude approximation of the real geometry, and the opening and closing is prescribed depending on crank angle. The valves are simply switched on or off by coupling or de-coupling the mesh parts. Although such a simple geometry does not give flow details within the valve itself, the resulting time dependent flow in the compressor parts is an useful input for subsequent investigations with a model of higher resolution. By mapping the results of the simple model onto a refined model as initial condition, computational efforts are reduced to a minimum.

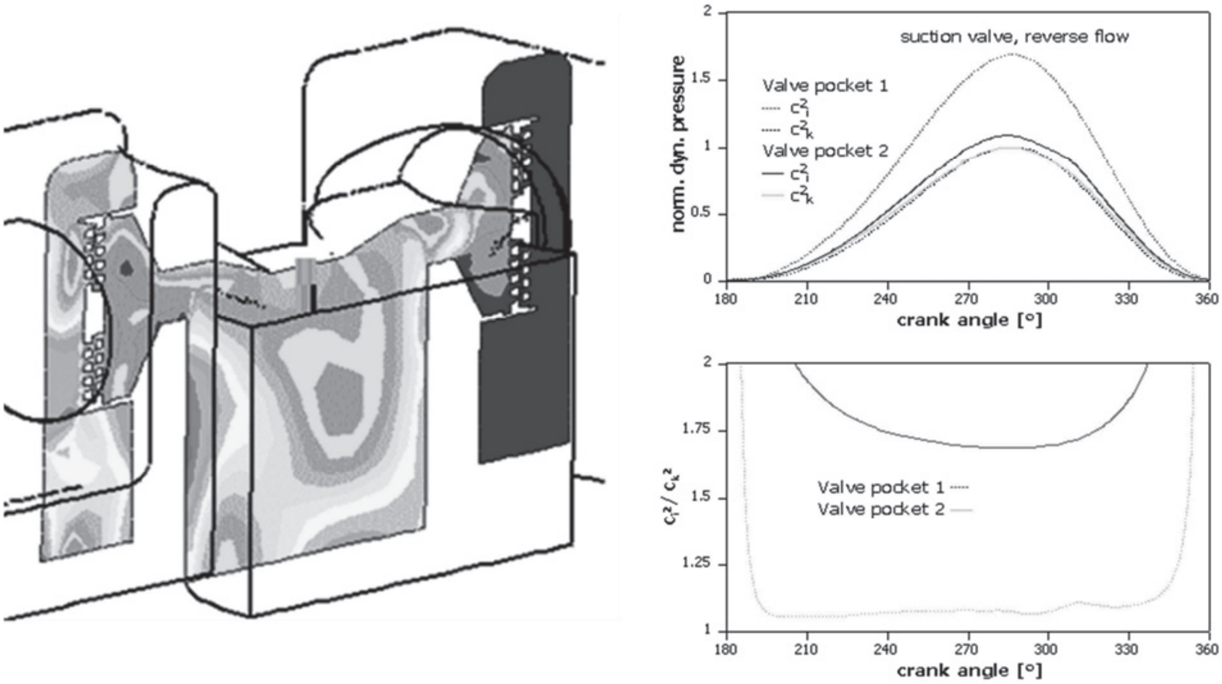


*Figure 3.1: Compressor cylinder moving mesh model(left), and comparison of dynamic pressures (diagrams right): Pressures calculated from continuity or integration of the momentum equation at suction valve outlet for two different valve pocket geometries (top), ratio of pressures dependent on crank angle (bottom)*

The purpose of the investigations was to determine forces and torques acting on a reverse flow control system. Due to the geometry of the valve pocket the unloader is not only pushed back but parts of the unloader mechanism are also subjected to torque. As a first indication, the momentum flux into the suction valve during reverse flow conditions derived from the model has been plotted vs. the crank angle (Figure 3.1, right).

As can be seen from the diagram, the dynamic pressure calculated from the continuity equation does not necessarily give a good estimation of the momentum flux. The ratio of dynamic pressure, as determined from a mass flow averaged integration of momentum flux, to the dynamic pressure, calculated from the velocity based on the continuity equation, is dependent on the valve pocket geometry as well as the crank angle. In order to obtain the averaged dynamic pressure relevant for the estimation of forces, the velocity determined from continuity must be corrected adequately, the dynamic pressure from momentum is nearly two times higher than the value calculated with the help of the continuity equation.

In a subsequent investigation of flow through a highly detailed model of a ring type suction valve closing with a prescribed motion, the initial and boundary conditions derived from the model described above  have been applied.

Figure 3.2 (left) gives an impression of the valve quarter model showing recirculation zones inside the valve caused by the sharp redirection of flow. These recirculation zones block a significant portion of the flow section of plate or ring valves, not only in under reverse flow conditions. The valve quarter model has been mirrored and a cylinder has been added to investigate the forces acting on the rings during closing in reverse flow (Figure 3.2, right).

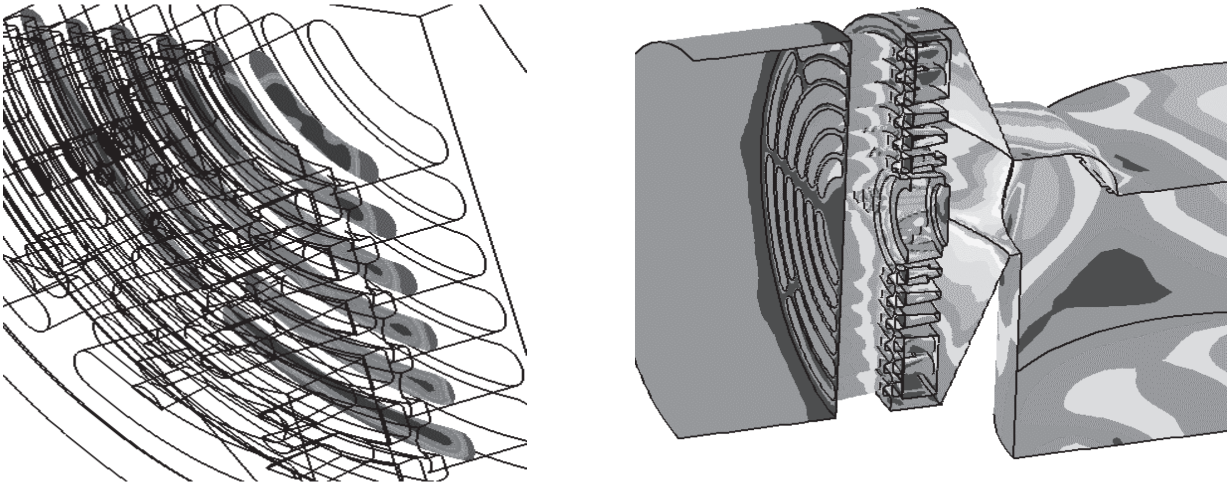

*Figure 3.2: Recirculation in a suction valve of ring type, reverse flow condition (compression stroke).Section plot of axial velocity (backflow in regions coloured red, right), internal view (i.e. velocities projected onto the wall, left)*

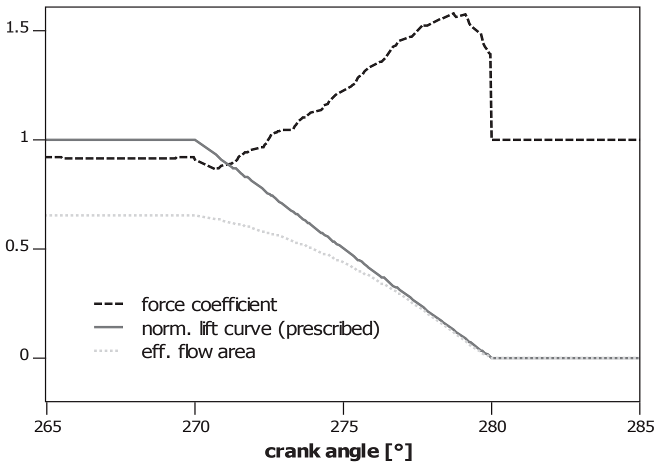


*Diagram 3.2: Normalized lift curve, resulting effective flow area and force coefficient dependent on crank angle.*

Note that the unloader bell is omitted in this model. The simulated force coefficient, the normalized valve lift curve and the effective flow section of the valve dependent on crank angle are shown in Diagram 3.2. The force coefficient is defined as force divided by the area effective in a closed position, and by the pressure loss of the valve. The normalized effective flow area is derived from the isentropic flow through an orifice.

### 3.2 Moving mesh simulation with fluid structure interaction: Double acting compressor cylinder with poppet valves

The third moving mesh model described within this paper comprises a double acting cylinder, two poppet valves of a new design, and the compressor parts up- und downstream of the valves. Figure 3.3 gives an idea of the whole model. The poppet valves are located in the top compression chamber, the bottom chamber is equipped with simple valve dummies (i.e. valves switching on or off as described above). In the CFD simulation poppet valves with seven poppets are investigated, one poppet located centrally and six poppets in a circle around. A new design of the poppet valve results in a very high flow section, especially in the valve guard, and hence the pressure loss in this part of the valve is reduced. In this CFD model, in contrast to the models described before, the motion of the valve bodies is not anymore prescribed as a function of the crank angle, but by solving the equations of motion for each individual poppet.

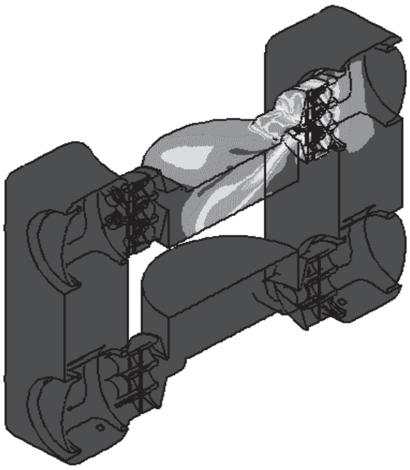

*Figure 3.3: Moving mesh compressor model with poppet valves in top compression chamber*

A multi cycle simulation was performed and the individual movement was simulated from the second simulated cycle onwards. In Diagram 3.3 the normalized lift dependent on crank angle is shown for two cases. On the left, the poppets are equipped with a spring causing premature closing of the suction valve. As can be seen from the diagram, the poppets at the top and bottom position close faster at the beginning, but are then the ones remaining open for the longest duration. The spring force at maximum lift, $F_{sm,}$ is equal to 1.0 for this model. In the Diagram at the right, even stronger springs are used ($F_{sm}$ ~ 8.0), resulting in an oscillation of the poppets. As a main result of the investigations, not only the individual lift curves are calculated but also the friction work is determined.

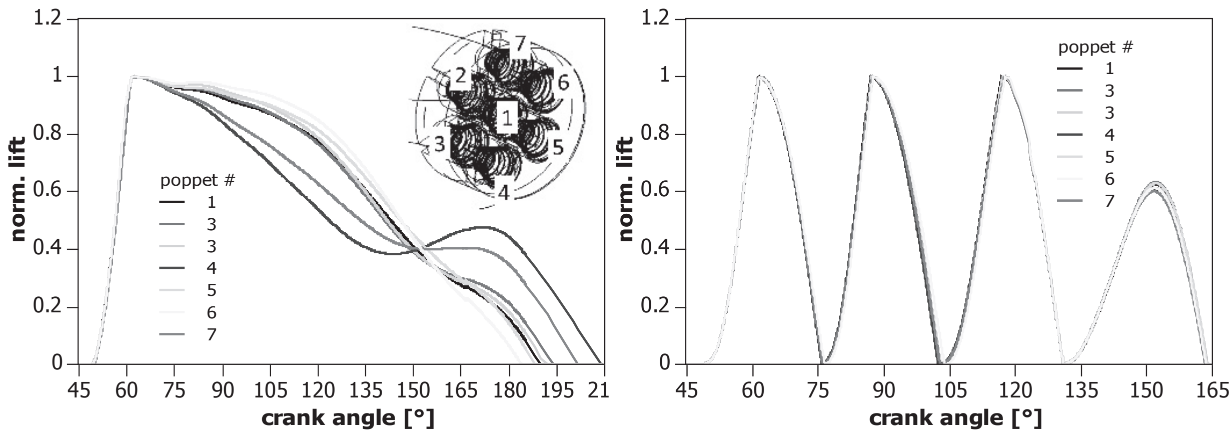


*Diagram 3.3: Individual lift curves of poppet valve bodies as simulated in CFD model. Springs ($F_{sm}$=1) resulting in poppets closing prematurely (left) and stronger springs ($F_{sm}$=8) resulting in synchronous oscillation (right)*

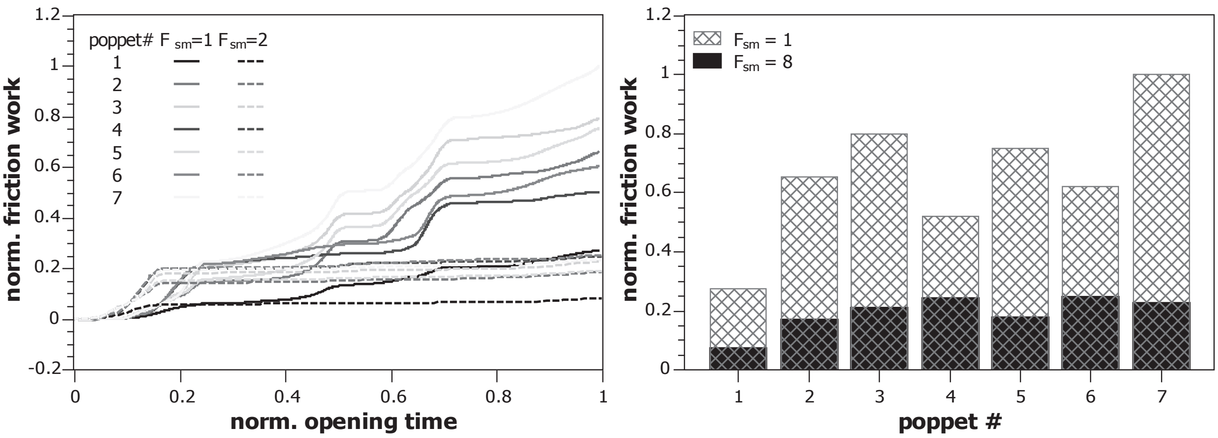


*Diagram 3.4: Friction work in poppet guide determined from lateral forces acting on poppets (left) over time and comparison of total friction work per cycle (right)*

The friction work is calculated as integral of the lateral forces times the poppet displacement and is plotted as function of the crank angle in Diagram 3.4. Not surprisingly, the friction work of the oscillating poppets is much higher than the friction work of the poppets equipped with a weaker spring. In both cases, the centrally located poppet renders the lowest friction. The determination of the time-dependent flow and valve motion is relevant both for the flow characteristics of the valve as well as for an assessment of the wear to be expected during operation. Due to the asynchronous movement of the valve bodies the effective flow area is dependent on time, both during opening and closing phases, and hence differs from the assumptions used for zero- or one-dimensional compressor performance or valve selector simulation tools.

## 4. Conclusion

Investigations aiming at reducing pressure losses of small sized poppet valves resulted in an improved valve design and triggered subsequent investigations of flow through the cylinder, valve pocket, valve and valve cage. Not surprisingly, if the overall pressure losses in a reciprocating compressor have to be minimized, the valve pocket geometry has to be taken into account. Not only is the pressure loss in the valve pocket significant, and in some cases even higher than the pressure loss of the valve itself, but also valve and valve pocket are influencing each other as the resulting flow is dependent on both the distribution of flow resistance in the valve and the pocket shape. If valve performance is rated on the basis of a wind tunnel arrangement (instead of a cylinder – valve pocket – valve arrangement) only, misleading results may be obtained. The impact of the design of the valve pocket on compressor performance cannot be over-emphasized. In order to be able to better predict the effects of valve pocket design on valve performance, two sets of half models – in total 28 different geometries – have been simulated so far.

The pressure loss coefficients of the pocket, valve and valve cage are usually treated as constants, however, as illustrated by a simple example, are also slightly dependent on the distribution of flow resistance in the valve, which can be calculated or estimated from the geometry. By introducing the distribution of flow resistance of the valve (described by additional parameters) the usual approach of pressure loss calculation required for compressor dimensioning and implemented in zero-dimensional codes is preserved, but with improved predictive capabilities–the conventional method is correct only if similar valves (e.g. plate valves with identical centre bolt diameter etc.) are compared among each other.

Steady flow investigations can only give an approximation of flow in a reciprocating compressor. Time-dependent motion effects like piston masking the valve etc. have to be included, and additional effects of transient flow like pulsations have to be taken into account. Moving mesh in-cylinder simulations are a tool common in internal combustion engine development; due to the close relatedness of reciprocating compressors and internal combustion engines a proven methodology has been applied.

Fluid structure simulations of moving valve internals create a deeper insight into valve dynamics and are a prerequisite for wear prediction. In order to predict valve wear, not only the forces acting on the valve internals need to be known but–since a guide does not wear as long as the valve does not move – the temporal evolution of forces, torques and velocities has to be determined and input into a wear model.

More cases need to be studied in detail. However, the investigations presented in this paper demonstrate how advanced modelling used in CFD today can contribute to increasing the reliability of compressor and valve performance predictions.

**Acknowledgments**

I want to thank all colleagues from Burckhardt Compression for the fruitful discussions and many suggestions, in particular Luzi Valär from Research and Development as well as Roland Aigner from the Design department. Furthermore, the contribution from external computing resources, in particular Mr. Gropp from CPU24/7, must not be forgotten.

**Bibliographic details**

[1] Bauer, F.: *Valve losses in Reciprocating Compressors*, 1988 International Compressor Engineering Conference at Purdue University, July 1988, West Lafayette, Indiana, USA

[3] Balduzzi, F., Ferrara G., Maleci, R., Babbini, A. and Pratelli, G.: *A Parametric CFD Analysis of the Valve Pocket Losses in Reciprocating Compressors,* paper no. PVP2013-97929, ASME 2013 Pressure Vessels and Piping Conference

[4] Pratelli, G., Babbini, A., Balduzzi, F., Ferrara, G., Maleci, R., Romani, L.: *CFD Evaluation of Pressure Losses on Reciprocating Compressor Components,* 8th Conference of the EFRC, Düsseldorf, September 2012

[5] Traversari, R., Faretra, M., Bucci, D., Bassini, S.: *Thermo-fluid-dynamic design of Reciprocating Compressor Cylinders by Fluid Structure Interaction Software simulation,* 7th Conference of the EFRC, Florence, October 2010

[6] Schiavone, M., Manfrone, F., Giacomelli, E.: *Evaluation of the Coefficients used for simulation for Cylinder Valves for Reciprocating Compressors,* 5th Conference of the EFRC, Prague, March 2007

[7] Manfrone, F., Raggi A.: *Analysis of the Movements of the Valve Sealing Elements,* 8th Conference of the EFRC, Düsseldorf, September 2012

[8] Aigner R., Meyer G., Steinrück H.: *Valve Dynamic and Internal Waves in a Reciprocating Compressor*, 4th Conference of the EFRC, Antwerp, June 2005

[9] Meyer G.: *Simulation der Strömung in einem Kolbenverdichter*, Diplomarbeit, TU Wien, 2005

[10] CD-adapco: *Methodology Manual, StarCD*, 2015