# Analysis of the Flow Conditions in a Dosing Pump with Regard to New Fuels


**Dipl.-Ing. Martin Petzold, Prof. Dr.-Ing. Jürgen Weber**

Institut für Fluidtechnik (IFD), Technische Universität Dresden, Helmholtzstrasse 7a, 01069 Dresden, E-mail: mailbox@ifd.mw.tu-dresden.de

**Dipl.-Ing. Etienne Dautry, Dr.-Ing. Olaf Ohligschläger, Dr.-Ing. Axel Müller**

Thomas Magnete GmbH, San Fernando 35, 57562 Herdorf, Email: info@thomas-magnete.com



## Abstract

Conveying new fuels with a high proportion of ethanol leads to an increased cavitation tendency inside a pump. This paper presents investigations of the flow conditions in a dosing pump using Computational Fluid Dynamics (CFD). For accurate spatial resolution of fluid mechanical details, a three-dimensional computational analysis of fluid-structure interaction in the outlet valve is surveyed. Comparisons are made by experimental testing. Noteworthy is a technique based on a laser Doppler vibrometer for examining the dynamics of the piston inside the fluid. The applied CFD cavitation model is parameterized and validated by experimental (optical imaging) and numerical investigations of the cavitating flow in an orifice flow. The objective of the developed method is to identify and reduce the potential locations for cavitation in order to ameliorate the high level of delivery accuracy.



KEYWORDS: dosing pump, cavitation, vibrometry, CFD, fluid structure interaction


## 1. Introduction

Dosing pumps, especially piston pumps (**figure 1**) are characterized as displacement units with a not too complex and robust design.

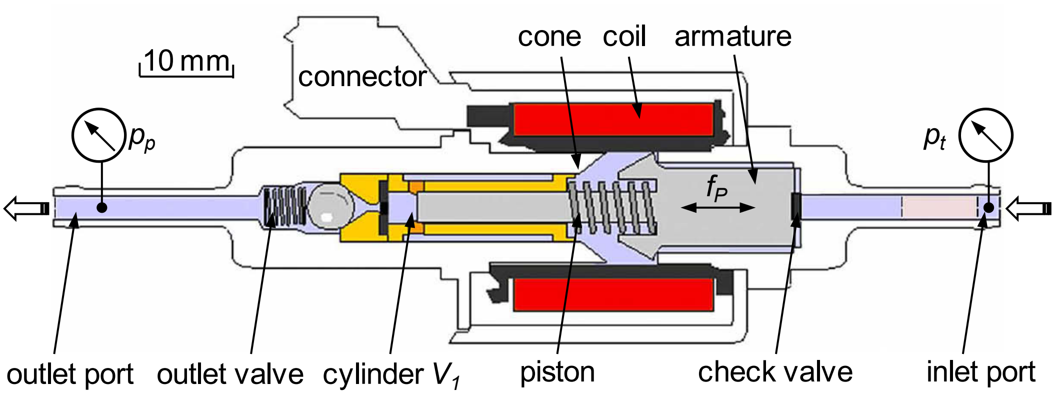


**Figure 1:** Dosing pump design

The pumps are designed to deliver an exact volume per stroke, independent of the pressure and temperature at the outlet port. The pump chamber is even with low-viscous fluids almost leak-proof, see /1/.

In motor vehicles, e.g. dosing pumps are part of the emission control system or used to convey a specific volume of fuel from a tank to the burner of an engine-independent heater. Especially under high temperature conditions or with fluids having low boiling temperatures, as gasoline or gasoline-alcohol blends, cavitation may occur due to the working principle and may change the operating behaviour of the pump. Hence, the fluid-structure interaction between the outlet valve and two-phase flow is influenced. As a result, both the noise level and metering accuracy are affected. Ethanol-containing fuels in particular have much higher vapor pressure than previously used diesel and gasoline fuels, thus, the potential of cavitation increases accordingly. To improve effects caused by cavitation, it is necessary to detect and analyze the critical areas. The experimental investigations of all relevant field variables inside the pump are due to the small dimensions of the pump volume rarely possible. With the application of CFD, we are able to display complex transient flow processes to acquire deep comprehension of significant physical processes and to reduce the number of prototype tests. The basis of simulating the pump operation is a detailed description of all relevant effects. The identification of cavitation zones requires a CFD model, which takes fluid-structure interaction into account. First, however, preliminary experimental and numerical investigations are needed to analyze and calculate the cavitation. Based on these investigations even elementary dosing pumps are empowered to meter fluids in a very precise manner.

## 2. Operation of the dosing pump

Most oscillating displacement pumps are equipped with a pressure and suction valve, working in opposite directions during the suction and delivery phases. In the analyzed pump, the suction valve is replaced by cross holes (**figure 2**). In this case, the opening of the inlet is a function of the pistons position. An adequate design of the cross holes results in a reduction of moving parts and increases the robustness of the pump with consistent conveying accuracy.

The motion of the piston is realized by an integrated electromagnet. When the coil is not energized, the displacement volume is filled with the fuel to be delivered. After current is applied to the coil, the armature moves the piston against the force of a spring. If the pressure rises above the set pressure, the outlet valve opens and the piston expels the medium from the displacement volume. Meanwhile, the inlet to the displacement volume is closed and the liquid continues to flow into the armature space. Once the supply of electricity is interrupted, the spring pushes the armature and piston back. At the beginning of the armature´s backwards movement, the piston overlaps the inlet holes and a partial vacuum is created in the increasing displacement volume. After opening the cross holes, liquid enters the open inlet and a new working cycle begins.

The analyzed pump has a displacement volume of $V_1$ = 27 mm$^3$ and is used up to a maximum outlet pressure of $p_p$ = 2 bar. With the adjustable stroke frequency $f_p$ = 0 - 25 Hz, the average flow rate can be varied continuously. The coil is placed under a voltage of $U_{nominal}$ = 12 V (square wave signal).

## 3. Investigation of the cavitation behaviour

The phenomenon cavitation characterizes the formation of cavities in a liquid, which then implode immediately. Happening usually when a liquid is subjected to rapid changes of pressure, cavitation can be divided into three types basically. A pressure decrease in a supersaturated solution causes the formation of gas bubbles called gas cavitation. Inertial cavitation (vapor bubbles) occurs if the static pressure falls below the thermodynamic saturated vapour pressure $p < p_v$. In the case of pseudo cavitation, already existing gas bubbles grow. For technical flows, usually all three types of cavitation occur simultaneously and a clear demarcation is often not possible /2/. In case of natural ventilation, automotive fuels can dissolve up to $\alpha_U$ = 21 vol% of air /3/. The maximum acceptable vapor pressure of gasoline E10 (for $T$ = 37,8°C) is $p_V$ = 90 kPa /4/. Compared to hydraulic oil or water, these properties lead, even at low pressure drops, to the formation of gas-/vapour bubbles.

### 3.1. Dominant cavitation zones in the pump

If one distinguishes cavitation considering the reason for the pressure drop, two dominant effects appear (figure 2).

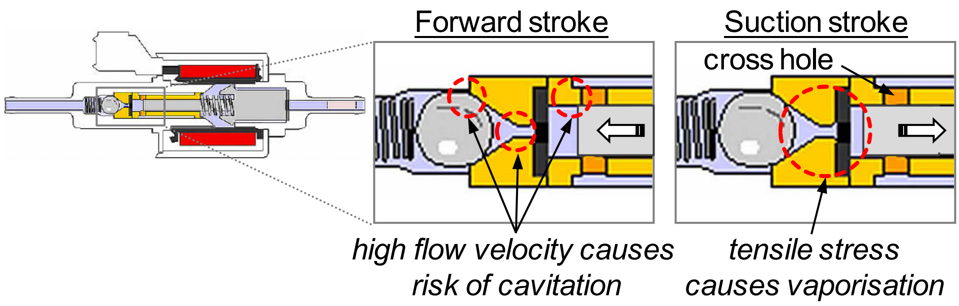


**Figure 2:** Risk of cavitation during a working cycle

At the beginning of the suction stroke, the piston overlaps the inlet holes. The reverse travel of the piston decreases the pressure within the liquid chamber by enlarging the internal volume of the chamber. As long as the cross holes are closed, no fuel flows into the volume. Tensile stress and cavitation nuclei cause the vaporisation of the liquid. The focus of this work is the analysis of cavitation during the forward stroke, since it significantly affects the dosing accuracy. The forward stroke of the piston reduces the internal volume of the liquid chamber, displacing the metered fuel out the discharge check valve. Assuming a low back pressure, a sufficiently high pressure gradient across the cross holes or the outlet valve causes high flow velocities and the occurrence of cavitation. This leads to the formation of finely distributed gas-/vapour bubbles, which can grow to an accumulation of bubbles. Bubbles in the suction duct adversely affect the suction performance. Accumulation of bubbles at the outlet port may move discontinuously to the next hydraulic consumer and reduce the conveying accuracy. Bubble collapse leads to noise emission and near a wall to cavitation erosion and component damage. The mentioned reasons demonstrate the importance of understanding and preventing cavitation in the pump.

## 3.2. Flow model for fluid analysis

To investigate the cavitation more precisely, the experimental analysis of the relevant areas inside the pump is not suitable, since they are difficult to access. For this reason, an orifice flow model has been developed. The occurring jet cavitation correlates to a typical fluid flow in the pump (see 3.1.) and helps to analyze the cavitation behaviour.

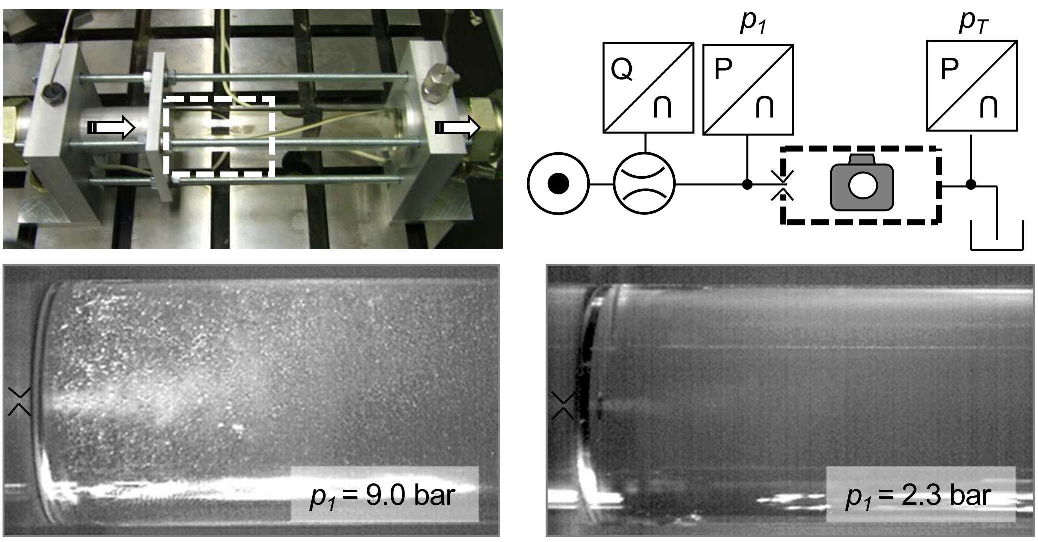


**Figure 3:** Experimental flow rig and high-speed records of the cavitating flow

The methods applied are photographic imaging with a high-speed camera (Basler A 504 K) and a stroboscopic flash light, relating to measurements of the pressure and flow rate. This data is intended to parameterize and validate a CFD multiphase model. **Figure 3** gives a simplified overview of the test rig. It is utilized to investigate various orifice designs. A piston pump in conjunction with an infinitely adjustable pressure relief valve maintains a constant pressure level at the inlet. The outlet pressure is kept at atmospheric pressure. An open tank represents the behaviour in an automobile and increases the amount of air dissolved in fuel. Upstream of the orifice, a volumetric flow sensor and a straight pipe are installed. With the help of two static pressure sensors, the flow rate can be measured as a function of the pressure drop. A transparent glass pipe allows observing the cavitation regions in the fluid flow. In order to obtain reproducible results, existing air in the system has to be removed before measurement. Moreover, during a series of measurements, the pressure drop should be reduced from maximum to minimum. Figure 3 presents the optical imaging results, recorded with a rate of 200 images per second, for different system pressures. The bright cloud is a mixture of fuel, steam and unresolved air.

### 3.3. Parameterization of the cavitation model

Taking cavitation into account, the commercial code Ansys Fluent is applied to simulate the orifice flow.

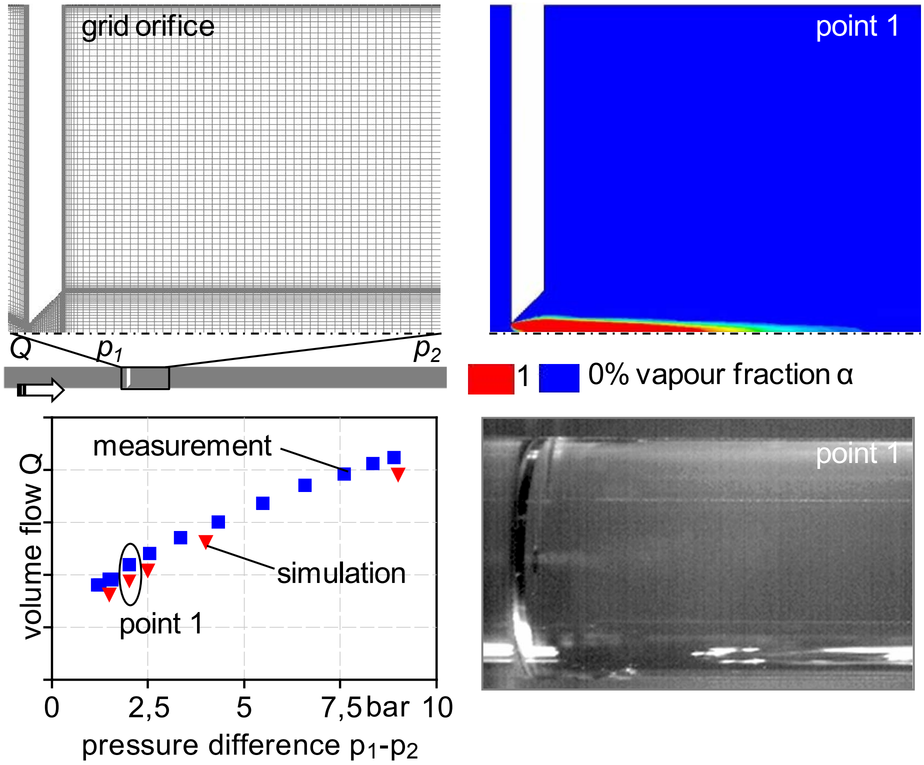


**Figure 4:** Comparison of the simulated and measured flow rate as a function of pressure drop and optical imaging of the cavitation

For this analysis, a rotationally symmetrically 2D grid is generated. The numerically most interesting flow region is located directly behind the narrowest flow cross section and requires a high grid resolution.

The Schnerr and Sauer model, which is used to simulate a two-phase fluid and is based on the Rayleigh-Plesset equation, takes the evaporation and condensation of a bubble into account /5/. The flow, with possible coexistence of liquid and vapour (modeled as ideal gas), is treated as a homogeneous mixture. No distinction between air and vapor is made. The phase boundaries between fluid and vapor/air are not resolved individually. However, for analyzing the flow field in the pump, the cavitation model is better suited than a single phase model, as it calculates the pressure and velocity distribution more realistically (necessary for fluid-structure interaction) and allows the qualitative comparison between different flow paths with regard to cavitation. The partially empirical constants of the cavitation model are validated for cavitation effects in water. Thus, the visualized and measured results of the test rig are used to adjust the fluid parameters on the behaviour of fuel (**figure 4**). The comparison of simulations with data shows good quantitative agreement of the pressure drop - flow rate characteristic and acceptable agreement with the imaging data (with respect to the cavitation inception). The determined fluid parameters are the basis of numerical simulation of the pump operation.

## 4. Methods of flow analysis

### 4.1. Experimental investigation

With regard to the delivery accuracy of the pump, the measurements of the pressure inside the displacement volume as well as the piston and ball dynamics are investigated.

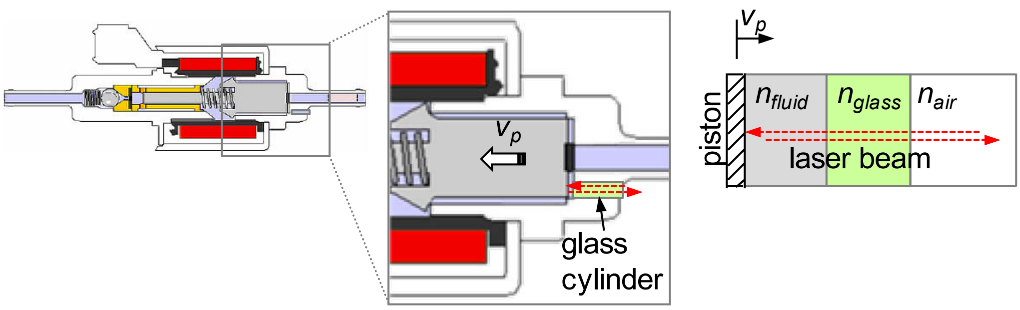


**Figure 5:** Dynamic motion measurement of the piston submerged in a fluid

The final positions of the magnetically driven piston can be determined by the analysis of the current characteristic and the voltage signal. In order to define the dynamics of the piston very precisely, a laser Doppler vibrometer (Polytec OFV-503; OFV-5000) for non-contact motion analysis is used. Most often, the surrounding medium of the moving object is ambient air. In this work, the laser is used to accurately measure the dynamic of a submerged solid object in fuel. The investigation undertaken into measuring the movement of an object inside a fluid is useful for examining many hydraulic components, for example the opening characteristics of valves. Often, the moving component must be equipped with additional devices, which change the mass, affect the flow path or pressure level, and cause additional friction or leakage.

In **figure 5** the experimental setup is depicted. The piston of the experimental prototype pump is optically accessible through a glass window. Therefore, a transparent and bubble free fluid is needed. The laser beam is focused on moving piston and parallel to the direction of movement.

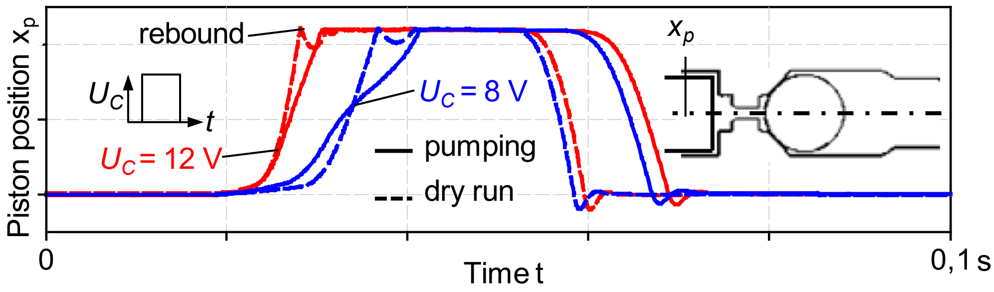


**Figure 6:** Measured piston dynamic for different coil voltage

Due to the motion, the Doppler effect causes a change in frequency of the laser beam. The output signal of the vibrometer depends on the optical path difference between the measurement beam and a reference beam. It is a continuous analog voltage and is directly proportional to the velocity of the object. If the object moves into a fluid, the index of refraction $n_{fluid}$ must be taken into account by dividing the measured velocity $v_{p.measured}$ value by this factor /6/.

$$v_p(t) = \frac{v_{p.measured}(t)}{n_{fluid}} \tag{1}$$

$$x_p(t) = \int v_p(t)dt \tag{2}$$

In **figure 6** the piston dynamic is presented for different coil voltages $U_c$. Increasing the voltage leads to higher magnetic force applied to the piston. Hence, increasing the voltage level $U_c$ form 8 V to 12 V, the piston accelerates much faster and finishes the forward stroke earlier. During pumping mode, the rising pressure inside the displacement volume causes an additional pressure force. Acting in the opposite direction to the original direction of movement this additional force reduces the piston speed. Moreover, it is obvious that the liquid influences the piston damping. For an unusual dry run (no fluid inside pump), a clear rebound stands in contradiction to the measured piston behaviour for the pumping mode. The suction stroke mainly depends on the spring force. The elastic properties of the piston stop can be determined very accurately.

Further investigations deal with the highly dynamic motion ball seat valve. Although the ball valve is quite small and low-cost, delivery accuracy and noise behaviour are highly depending on their reliability. Furthermore, the valve affects the efficiency factor of the pump. The requirements concerning the valve are: a long service life, a reproducible and reliable behaviour, low pressure drop during forward stroke as well as tightness during suction stroke /7/.

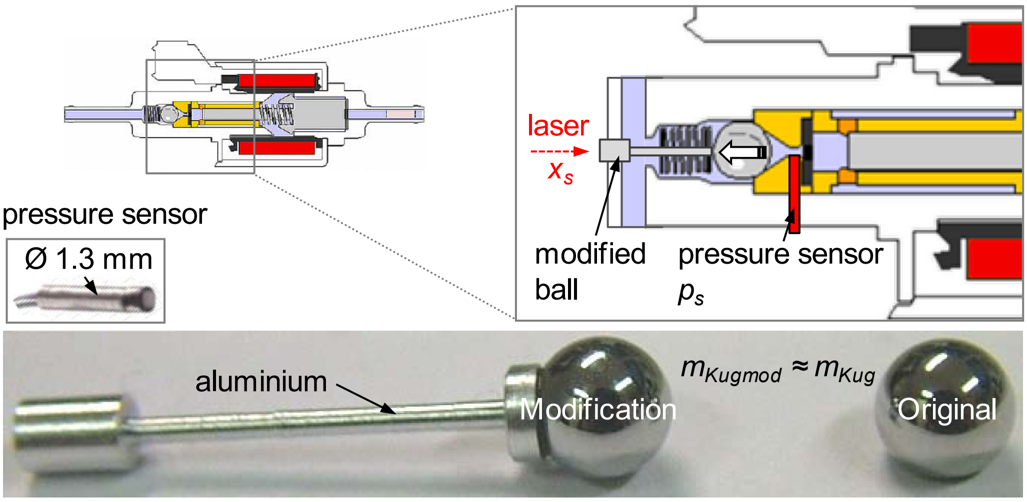


**Figure 7:** Measurement of the ball motion and the cylinder pressure

To understand the valve kinematic, a test rig is used. The magnitudes of the ball diameter $d_s$ < 5 mm and the expected ball stroke $x_s$ < 0.2 mm expose the demands on the measurement technique. Due to the inaccessible position of the ball valve, the non-contact vibrometer measurement of the ball motion is difficult to realize. For this reason, the ball is modified by a rod and guided to the outside of the pump, see **figure 7**. To measure at almost realistic operating conditions, special attention has been given to maintain the ball mass and the flow conditions around the ball. By means of focusing the laser beam on the side surface of the rod, the valve motion is recorded synchronously to the pressure. The pressure is measured by a subminiature pressure sensor (measurement specialties EPIH-17; $p$ = 0-5 bar), specifically designed for dynamic and high frequency measurements.

The graphs in **figure 8** summarize the most relevant quantities of an operation cycle. A voltage step is applied to the actuator. The characteristic current drop during forward stroke is due to the motion-induced electromagnetic field and caused by the increase of the inductance with a decreasing air gap. The dynamic behaviour of the ball valve is shown as a function of the displacement volume pressure and the piston dynamics. According to the low mass and very minor forces, the ball movement reacts sensitively to all types of disturbance.

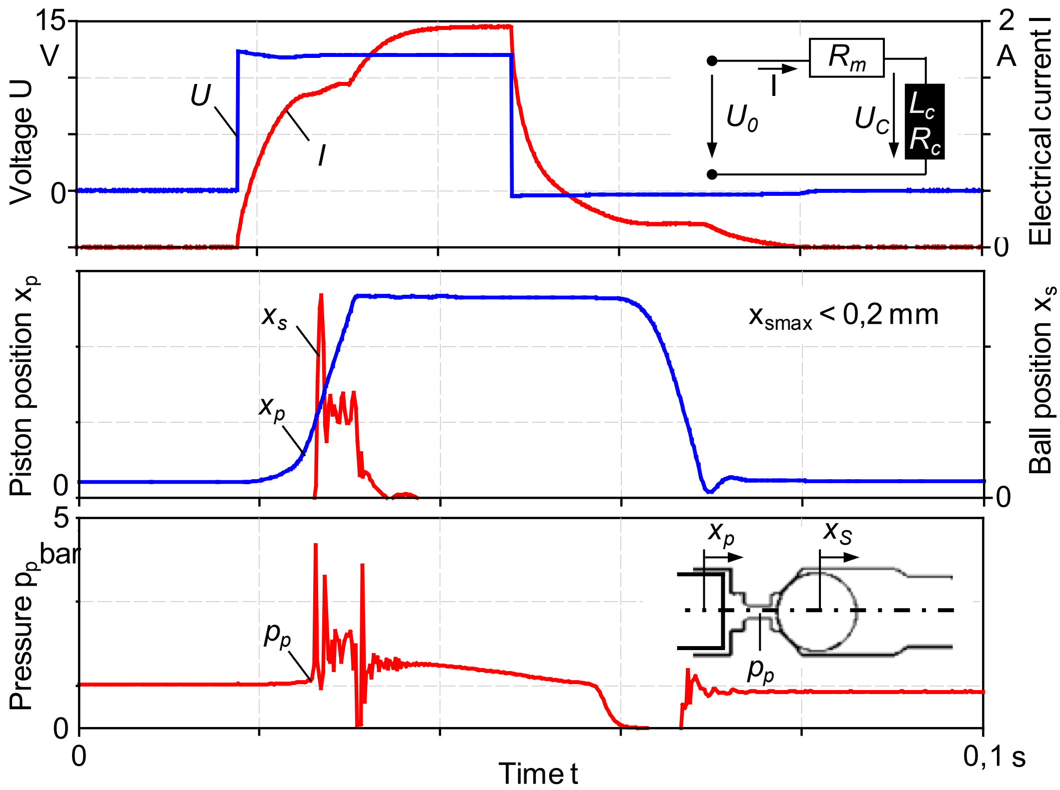


**Figure 8:** Measurement of the ball motion and the displacement chamber pressure

### 4.2. Numerical approach by CFD-simulation

The numerical simulation of the pump is a useful tool for optimizing the flow paths, especially in a product development process. With the help of computational fluid dynamics which takes the piston motion and the fluid driven ball valve into account, a profound understanding of the flow effects can be achieved. So, the dynamic simulation can significantly reduce testing time and costs. The aim of this work is to investigate the valve kinematics and the interactions with the surrounding fluid by analyzing the fluid-structure interaction (FSI). An overview of FSI is given by /8/. For analyzing the ball motion, the assumption of a 2-way interaction with rigid body motion can be made. The deformations of the ball or the housing are not modeled. For all numerical simulations, the cavitation model is used as described in chapter 3.3.

Considering the dominant direction of the ball movement (one degree of freedom), the following equation of motion is determined:

$$m_s a_s = \sum F_{CFD} + \sum F_{external} \tag{3}$$

The fluid flow-induced forces $F_{CFD}$ (here: pressure force, force due to the viscous friction) and external forces $F_{external}$ (here: spring force) affect the valve dynamics.

To model the solid motion of the piston and the ball, the mesh has to change according to the present position. Due to the symmetrical design of the pump, a numerical quarter section model with two symmetry planes as boundaries can be used. The CFD code provides different approaches to deal with moving/deforming meshes. Thus, at each time step the grid is automatically updated as a function of the new position of the moving parts (**figure 9**).

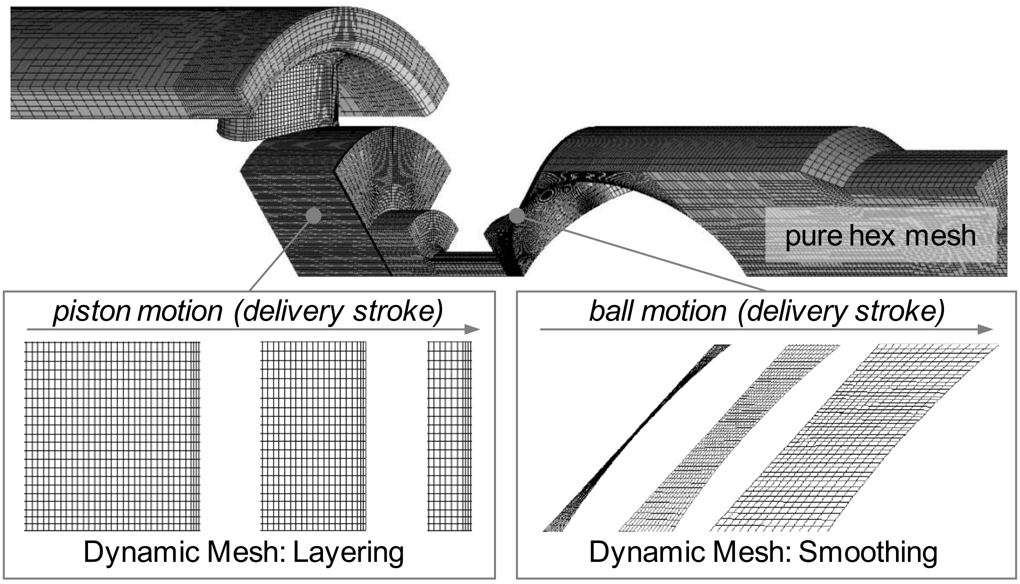


**Figure 9:** Dynamic mesh update methods

Dynamic layering is used to consider major volume changes of the displacement chamber. In this case, layers of cells adjacent to a moving boundary are added or removed. As the displacement of the ball is small, a spring smoothing approach for the dynamic mesh update is suitable. The fluid inside the opening gap has a low Reynolds number and, thus, laminar flow is assumed. A pressure inlet and pressure outlet is used. The approach neglects the leakage gap between the piston and the cylinder.

A first model verification is done by analyzing the stationary valve behaviour. For this purpose, the pressure drop and ball position needs to be calculated as a function of the constant volume flow. The impressed volume flow is realized by the piston movement, assuming an infinite uniform motion. The piston velocity, used for simulation, correlates to the measured, mean velocity of the piston during opening time of the valve. **Figure 10** shows the high accuracy in stationary behaviour to be achieved with the CFD model.

Due to the force-dependent motion of the ball, the flow conditions are changing all the time; this is why the fluid flow may be not purely analyzed on the basis of the stationary valve behaviour. For determination of the real flow conditions, a realistic ball kinematics is essential. The calculation takes the dynamic impact of the forces into account, summarized in equation (2). In reality, the piston moves due to the acting magnetic force. For simulation this is simplified by using an experimentally determined movement considered as a $x_p(t)$-profile.

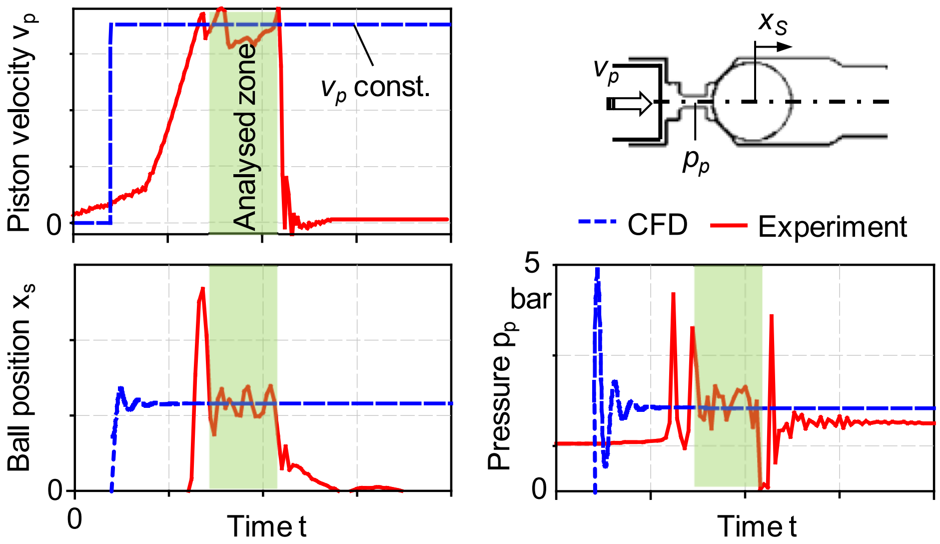


**Figure 10:** Calculation of the stationary ball position for a constant piston velocity compared to measurement results

In **figure 11** the comparison of the experimental and numerical results is shown, taking the dynamic pump behaviour into account. After the cross holes are closed, the pressure pushes the valve. Hence, the movement is a result of the fluid forces applied to the valve. The inertia of the valve body and the surrounding medium leads to a peak pressure, followed by high amplitude of the valve motion and the oscillation of the spring-mass system along the axis. For both simulation and measurement, it is obvious that after the opening oscillation is damped out, the ball follows the given flow rate caused by the constant piston movement with good coincidence. Analyzing the differences between simulation and measurement of two points, apart from the simplified flow volume and possible production tolerances, the following should be noticed:

A) Potential adhesive and friction forces in the experiment are neglected in the simulation. Furthermore, due to the low mass and very minor forces, the ball movement reacts sensitively to all types of disturbance. Small bubbles in the fluid or a non-axial ball movement may influence the valve behaviour. Therefore, even two consecutive valve openings are never the same.

B) As illustrated in figure 10, the piston does not move with constant velocity, as assumed in simulation. In reality, the movement of the piston is influenced by the position dependent magnetic and pressure force which causes axial vibration during forward stroke.

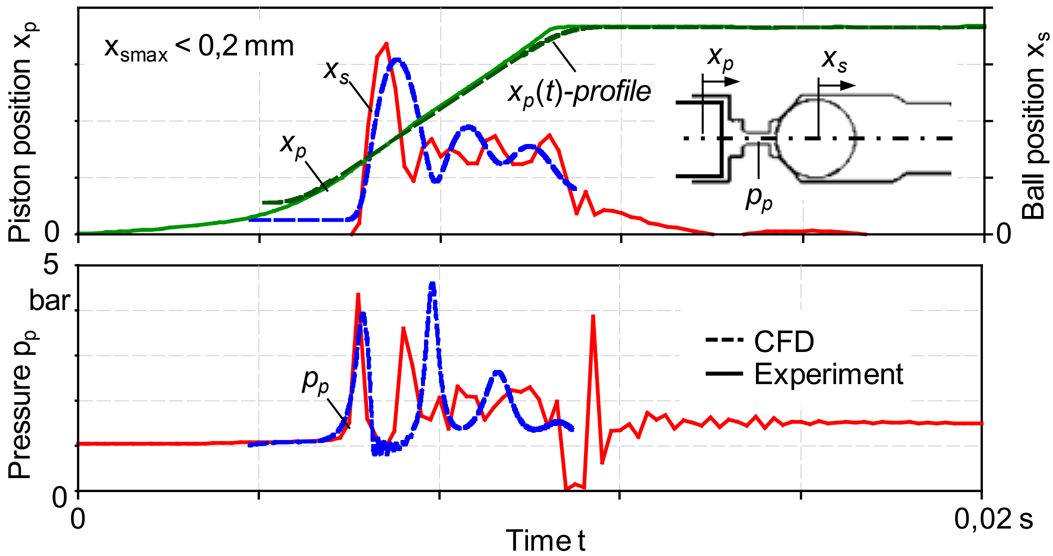


**Figure 11:** Calculation of the ball motion and the pressure in the displacement volume compared to measurement results for a working cycle

## 5. Conclusions

The operation behaviour of a dosing pump may change by a high cavitation tendency of the conveying fluid. The experimental investigations of all relevant field variables inside the pump are due to the small dimensions of the pump volume rarely possible. This article explains a methodology to locate and reduce potential cavitating areas in a dosing pump in order to elevate the high level of delivery accuracy.

To simulate cavitation, a CFD two-phase model was parameterized and validated with the help of experimental, preliminary studies. Due to the consideration of fluid-structure interaction for the outlet valve, it was possible to precisely determine local flow processes at any position for an entire working cycle. Experimental testing was performed to validate the CFD model. Mentionable is a technique based on a laser

Doppler vibrometer for examining the dynamics of the piston inside the fluid. Further research may take the fluid-structure interaction for the piston into account.

## 6. References


/1/ Rolland, T.; Müller, A.; Kappler H.: An experimental approach to the capability of metering units for future fuels, European Fuel Cell Forum, Luzern 2011, B1302

/2/ Wustmann, W.: Experimentelle und numerische Untersuchung der Strömungsvorgänge in hydrostatischen Verdrängereinheiten am Beispiel von Außenzahnrad- und Axialkolbenpumpen, Dissertation, TU Dresden, 2009

/3/ Gilles Birth, I.: Optische Untersuchung der Düseninnenströmung von Sitzlochinjektoren für die Benzindirekteinspritzung, Dissertation, Universität Karlsruhe (TH), 2008

/4/ DIN 51626-1: Kraftstoffe für Kraftfahrzeuge – Anforderungen und Prüfverfahren – Teil 1: Ottokraftstoffe E10 und Ottokraftstoffe E5, Entwurf November 2010

/5/ N.N.: Ansys Fluent 12.1 Documentation

/6/ Marsili, R.; Pizzoni, L.; Rossi, G.: Vibration measurements of tools inside fluids by laser Doppler techniques: uncertainty analysis, Measurement, 2000, No. 27, pp 111-120

/7/ Blendinger, S.: Strömungsinteratkionen, Kinematik und Verschleiß fluidgesteuerter Pumpenventile, Dissertation, Universität Erlangen-Nürnberg, 2010

/8/ Erhard, M.: Numerische Strömungsberechnung (CFD) im Ventilentwurf, Ölhydraulik und Pneumatik, 2011, Nr. 11-12, S. 440-445


## 7. Symbols

| | | |
|---|---|---|
| $a$ | acceleration | mm/s$^2$ |
| $f$ | frequency | Hz |
| $F$ | force | N |
| $m$ | mass | kg |

| | | |
|---|---|---|
| $n$ | refractive index | - |
| $p$ | pressure | bar |
| $p_v$ | saturated vapour pressure | bar |
| $t$ | time | s |
| $T$ | temperature | °C |
| $U$ | voltage | V |
| $v$ | velocity | mm/s |
| $V_1$ | displacement volume | $mm^3$ |
| $Q$ | volume flow | l/min |
| $x$ | position | mm |
| $\alpha_U$ | volumetric proportion of air | - |