Flow determination of a pump-turbine at zero discharge

G. Edinger¹, S. Erne², E. Doujak¹ and C. Bauer¹

¹Vienna University of Technology, Institute for Energy Systems and Thermodynamics, Department of Fluid-Flow Machinery, Getreidemarkt 9/302, 1060 Vienna, AT
²Voith Hydro GmbH & Co. KG, Linzer Straße 55, 3100 St. Pölten, AT
E-mail: gernot.edinger@tuwien.ac.at

Abstract. When starting up a reversible Francis pump-turbine in pump mode, the machine may operate at zero flow at a given gate opening. Besides reversal flow and prerotation in the draft tube cone, the onset of a fully separated flow in the vaned diffuser is observable at zero-discharge condition. In this paper, the occurrence of prerotation and reversal flow in the conical draft tube and the flow in one stay vane channel of a pump-turbine are examined experimentally and compared to numerical simulations. In order to assess the strongly three-dimensional flow in the stay vane channel, measurements with a 2D laser doppler velocimeter (LDV) were performed at various positions. The inlet flow in the draft tube cone, which becomes significantly at zero discharge in pump mode, is investigated by velocity measurements at two different positions. Pressure fluctuations in the draft tube cone induced by complex flow patterns are also recorded and analyzed. It is found that the swirl number at zero discharge does not significant differ from the values obtained at very low load pumping. Experimental investigations combined with CFD have shown that in the stay vane channel flow velocity components different from zero occur even at no discharge. Streamline plots show the fully separated flow structure.

1. Introduction
As renewable sources are nowadays growing at a very fast rate, the electrical grid becomes more and more affected by highly volatile energy production technologies. However, during peak demand, balancing facilities like pumped-storage hydroelectricity also contribute significantly to the electrical grid stability. The mode of operation of a pumped-storage pump-turbine has rapidly changed and makes it thus important to know about the specific loads during start-up. Pump-turbines are started in pump-mode up to ten times and more a day, which results in a higher load than compared to a single start a day. Radial fluid-flow machines with the discharge valve closed often operate for a certain time period at zero-flow conditions during start-up in pump mode. Starting a water-filled pump-turbine with opened guide vanes can result in similar conditions when the spherical valve upstream is closed. Characteristic flow structures at deep part-load conditions are addressed for example by Breugelmans and Sen [1] or Eisele et al. [2]. It is well known that such flow phenomena have a negative influence on the machine because of higher load caused by unsteady flow and pressure fluctuations. As stated by Sick et al. [3], a thorough understanding of such complex flow patterns might be valuable for lifetime assessment methods. Ancinger et al. [4] describe the region of instability at low flow rates \( Q/Q_N < 0.88 \) as a rough operation zone, at which the pump-turbine has to be shut down due to damage risk.
1.1. Part load flow

The flow patterns in the draft tube cone during part load operation in pump mode have been discussed in many publications. Investigations by Zhang [5] show a large backflow and prerotation zone at part load when the flow rate decreases below a critical flow rate. The recirculation zone increases when flow decreases, which was also found by Kaupert et al. [6]. Eisele et al. [2] describe the relationship between the instability of the head-capacity curve, rotating stall phenomena in the vane diffuser and the flow phenomena at the runner inlet such as prerotation and backflow. There is also a relationship between stall phenomena and the guide vane setting. The main focus of these investigations was the hysteresis of the head-capacity curve drop. Breuergmans and Sen [1] investigated centrifugal pumps with focus on the runner inflow conditions. They also found that these flow phenomena occur when the discharge drops below a critical flow rate. The two main criteria for backflow and prerotation are local flow separation and high pressure gradients in radial directions, as can be found in Güllich [7]. Güllich [7] furthermore detected that the increasing head-capacity curve at very low flow rates is strongly influenced by backflow and prerotation.

Although these and other numerous investigations have been carried out, no general method to predict these complex flow structures at the inlet has been found yet. Gentner et al. [8] describe rotating stall in the vane diffuser of a pump turbine occurring even when the flow in the runner is fully stable. They also confirmed the coincidence of instability of the head-capacity curve with onset of prerotation and backflow at the runner inlet at low flow rates. In Ancinger et al. [4], an operating point just before the instability of the head-capacity curve is investigated numerically and a rotating stall consisting of three stall cells can be found. These stall phenomena induce strong variations of the flow velocity in the stay vane flow passage. At zero discharge, the flow separation in the tandem flow passage has fully developed and the fluid is partially flowing back into the runner, which can be found in Güllich [7]. Zhang [5] also deals with rotating stall phenomena and showed flow separation in the vane diffuser at \( Q/Q_N = 0.74 \).

The focus of the investigations in this paper is to describe how the above mentioned and well known flow phenomena influence the zero-flow condition in a pump-turbine reduced scale model.

2. Experimental setup

The experimental investigations were conducted in the Hydrodynamic Laboratory of the Institute for Energy Systems and Thermodynamics. Figure 1 shows a scheme of the test rig. A reduced scale model pump-turbine (2) was investigated on a closed loop test rig. The reduced scale model has a specific speed \( n_q \) of 41 min\(^{-1}\) and 7 runner blades. The stay ring of the spiral case has 20 stay- and guide vanes. One feature of this specially designed test rig is the installation of a spherical valve (5) between spiral case outlet and head water tank (6) to perform a start-up sequence at closed spherical valve. Accurate measurements of the head capacity curve were realized by the usage of a needle valve (8), installed at the manifold just before the flow enters the tail water tank (1). The hydraulic torque acting on the runner was measured by means of torque flange and a hydrostatic bearing (3). This arrangement enables the consideration of the friction losses caused by the bearings of the turbine shaft. The model was driven by a variable speed DC motor generator (4). The head was determined by a differential pressure transducer connected to the draft tube inlet and using the spiral case outlet. This corresponds to \( p_2 - p_1 \), shown in fig. 2. The flow rate was recorded by an electromagnetic inductive flow meter (7) in the manifold of the test rig. For the velocity measurements, a 2D LDV in backscatter mode was used to measure the flow velocities simultaneously in two perpendicular directions. In order to ensure an optical access to the flow regions of interest, the draft tube cone was made of acrylic glass with an octagonal shape on the outside. Furthermore, one side was milled at the same angle as the opening of the diffuser to get a parallel wall in the direction of the turbine axis.
Figure 1. Scheme of the closed loop test rig: 1 tail water tank, 2 pump-turbine model, 3 hydrostatic bearing, 4 DC motor generator, 5 spherical valve, 6 head water tank, 7 flow meter, 8 needle valve.

Figure 2 shows the two planes (E1 and E2) for the LDV measurements. At these planes, the tangential velocity $C_\theta$ and the meridional velocity $C_m$ were measured at several points along a radius. The azimuthal position of this velocity measurements was on the opposite of the nose vane of the spiral case. Transient wall pressure measurements ($p_{w,t}$) were also conducted in the conical part of the draft tube. As can be seen in fig. 2, four piezoresistive pressure transducers were placed circumferentially and equally spaced on the measuring plane E1 in the diffuser. For the measurements in the stay vane channel, the spiral case was equipped with two windows. Each of these windows can be covered with a glass pane or an aluminum cover which ensures the exact hydraulic contour of the inner side of the spiral case. Using the aluminum covers, a possible major influence of the global performance data can be identified or excluded. The primary and secondary velocity obtained by experiments are based on a one measurement campaign as repeated measurements have shown very similar results. The uncertainty of the used measuring chains are listed in tab. 1.

3. Computational model

For the simulations, the incompressible Reynolds averaged Navier-Stokes equations were numerically solved by using the open source CFD toolbox OpenFOAM-1.6-ext. A $2^{nd}$-order accurate scheme was chosen for time-stepping, where a purely linear discretisation scheme was applied to the convection term in the momentum equation. The velocity-based moving mesh
Table 1. Measurement uncertainties.

| measured value | $T$ | $n$ | $Q$ | $p_{w,t}$ | $p_2 - p_1$ | $r$, $x$, $y$ | $c_m$, $c_t$, $c_z$ |
|----------------|-----|-----|-----|--------|-------------|--------------|------------------|
| uncertainty ($\pm$) | 0.5 | 0.02 | 0.16 | 0.002 | 0.002 | 0.3 | 0.5 |

considers a static mesh region (spiral case, draft tube and guide vane) and a rotating mesh region (runner).

At the inlet and outlet of the domain, a zero-discharge boundary condition was prescribed to ensure a closed system. An important aspect of the numerical setup is the initial flow field, which was obtained from a weak start-up procedure of the pump-turbine runner after attaining a steady-state configuration. As high-velocity flows are expected to appear primarily in the runner channel, stay- and guide vane channels and draft tube cone, mesh areas of lower mean velocity were locally coarsened. The frequently used two-equation turbulence closure model $k$-$\omega$ SST was applied, which is mostly adapted in industrial applications (Menter and Esch [9]). As the $k$-$\omega$ SST retains computational stiffness and a sufficient predictability of turbulent shear flow, the model was used as the basis for further adaptions. In a pump-turbine operating at zero discharge, azimuthal momentum of the flow is enhanced upstream and downstream of the rotating runner so that the fluid is rotating in the vaneless spaces between runner and stay vanes. Therefore, the turbulence model is sensitized to the influence of rotational effects on the turbulence based on mean flow gradients and the runner rotation as proposed by Smirnov and Menter [10]. Such effects imposed by streamline curvature and system rotation are taken into account by the coefficient $f_{r1}$, entering the RANS model via the production terms in the equation for $k$ and $\omega$.

Following Menter et al. [11], the turbulence closure model is extended by a DES formulation (detached eddy simulation), which applies the standard SST formulation in the near-wall region and behaves like a LES-type model in the free shear flow. To ensure the DES is disabled in the coarsened mesh regions, the switch $F_\Delta = 0$ was defined in the coarsened RANS region. A zonal distinction between the turbulence flow regime in the refined mesh region is made by the delimiting coefficient $F_{DES}$ in the destruction term $\beta^*k\omega$ of the $k$-equation. The transport equations of the modified RANS model $k$-$\omega$ SST-CC DES read

$$\frac{\partial k}{\partial t} + C_j \frac{\partial k}{\partial x_j} = \mathcal{P} f_{r1} - \beta^* k \omega (1 + F_\Delta [F_{DES} - 1]) + \frac{\partial}{\partial x_j} \left[ \nu_{eff} \frac{\partial k}{\partial x_j} \right] \tag{1}$$

$$\frac{\partial \omega}{\partial t} + C_j \frac{\partial \omega}{\partial x_j} = \frac{\gamma}{\nu_t} \mathcal{P} f_{r1} - \beta^* \omega^2 + \frac{\partial}{\partial x_j} \left[ \nu_{eff} \frac{\partial \omega}{\partial x_j} \right] + (1 - F_1) C_D k \omega \tag{2}$$

where $F_{DES}$ delimits the DES formulation by comparing the RANS length scale and DES length scale:

$$F_{DES} = \max \left( \frac{\sqrt{k}}{\beta^* \omega C_{DES} \Delta} (1 - F_1), 1.0 \right) \tag{3}$$

In eq. 3, the grid influence on the RANS boundary layer flow is reduced by using the SST blending function $F_1$, suggested by Menter [11]. The local grid spacing is $\Delta = (\Delta_x \Delta_y \Delta_z)^{1/3}$, assuming all cell edges have approximately the same length. As in Shur et al. [12], the DES constant is blended between RANS and DES zone $C_{DES} = (1 - F_1) C_{k-\epsilon} + F_1 C_{k-\omega}$, where $C_{k-\epsilon} = 0.61$ and $C_{k-\omega} = 0.78$. 
4. Results

4.1. Global flow quantities

For a qualitative assessment of the zero-discharge condition, global flow quantities are used. The speed factor and the torque factor according the international standard IEC 60193:1999 are chosen to properly draw a comparison between experiments and CFD:

\[
E_{nD} = \frac{gH}{n^2 D_1^2} \quad \text{(energy factor)} \quad (4)
\]

\[
T_{nD} = \frac{T}{\rho n^2 D_1^5} \quad \text{(torque factor)} \quad (5)
\]

| model       | \( T \) | \( H \) | \( E_{nD} \) | \( T_{nD} \) | \( \Delta E_{nD} \) | \( \Delta T_{nD} \) |
|-------------|---------|---------|-------------|-------------|-----------------|-----------------|
| EXP         | 248.8   | 26.73   | 19.31       | —           | —               | —               |
| \( k-\omega \) SST-CC DES | 258.5   | 26.78   | 19.34       | 0.90        | +0.2            | +3.9            |

Table 2 compares the global quantities obtained from the numerical simulation and experiments. The \( \Delta \)-values indicate the deviation of the simulation compared to the measurements. The \( k-\omega \) SST-CC DES model predicts the head very accurate, also the deviation of the torque is very small. An influence of the optical access panes on the global performance data in the experiments was not detected, however a influence on the velocity field in the stay
Flow in the draft tube cone

As mentioned above, the flow in the diffuser becomes strongly three-dimensional when decreasing the discharge. In order to characterize the occurring prerotation and backflow, a flow angle is defined as

$$\beta = \arctan \left( \frac{C_\theta}{C_m} \right),$$

(6)

According to eq. 6, a straight diffuser flow corresponds to the flow angle $\beta = 0$, where the range $0 < \beta \leq \pi/2$ means prerotating flow. Reversal diffuser flow is present when the flow angle is in the range of $\pi/2 < \beta \leq \pi$. Counter rotating flow ($\beta > \pi$) was not considered in experiments and simulations. Figure 6-left shows a scheme of the velocity components for determining $\beta$.

In order to characterize the strength of the swirling flow, the dimensionless integral quantity called swirl number $S$ is defined, as suggested by Beér and Chigier [13]. This factor is the angular momentum flux related to the axial momentum flux multiplied by a characteristic radius. Neglecting turbulent velocity fluctuations as in Bender [14] as well as pressure terms following Weng [15] leads to the definition of the swirl number

$$S = \frac{2\pi \rho \int_0^R |c_m|c_\theta r^2 dr}{2\pi \rho R \int_0^R c_m^2 r dr} \approx \frac{\sum_{i=1}^n |C_{m,i}||C_{\theta,i}r_i^2\Delta r_i}}{R \sum_{i=1}^n C_{m,i}^2 r_i \Delta r_i},$$

(7)

where the magnitude of the meridional flow velocity is considered. Due to a point-by-point measurement of the velocities, the single terms of the summation can be interpreted as local swirl numbers. This local swirl numbers are calculated by a numerical integration on the basis of ring areas of the diffuser cross section. In eq. 7, $n$ stands for the number of ring areas, which corresponds to the number of measurement points along the radius. Figure 6-right shows the method of determining the swirl number according to eq. 7. The radial component of the flow velocity is not considered. Figure 7 shows the development of the flow angle $\beta$ from the center of the diffuser.
of the draft tube diffuser to the wall. The error bars represent the variation due to the standard deviation of the measured velocities, calculated from the Gaussian error propagation without taking the covariance into account. Since the flow is unsteady, the range of variation is relatively high and can be interpreted as the standard deviation of the flow angle. The dashed line at $\beta/\pi = 0.5$ marks the onset of backflow. Referring to fig. 7, a comparison of measurement data for two different operating points can be seen. The zero discharge flow condition is compared to a deep part-load operating point ($Q/Q_N = 0.2$). As can be seen from the diagram, similar flow conditions for both operating points appear. Even at $Q/Q_N = 0.0$, there is a strong flow. For these flow conditions, the global swirling strength can be identified by $S = 1.08$ and $S = 0.63$ respectively. Local calculated swirl numbers are shown in fig. 8, wherein the error bars represent the standard deviation of the local swirl number. In the region of $1-r/R = 0.2...0.1$, a significant peak of the local swirl number appears, caused by the stagnation of the meridional velocity $C_m$. Due to the small values of $C_m$, the standard deviation increases similarly. Compared to other operating points at deep part-load, a strong swirling flow in the diffuser comes apparent at $S > 1$, presented in tab. 3.

At E2, S shows a slightly higher value than E1, which might result from the cross-sectional contraction. Focusing on the numerical simulations, the values of S are calculated for both E1 and E2 which show a rather poor agreement with the experimental values. As can be seen in fig. 10, the peak in the local swirl number is much higher than in the experiments. Figure 9 shows the simulated flow angle over the radius, which agrees well with the data from experiments. Backflow and prerotation are well predicted by the $k$-$\omega$ SST-CC DES model.

### Table 3. Swirl numbers of different operating points and measurement sections.

| $Q/Q_N$ | 0.0 (EXP) | 0.0 (CFD) | 0.2 (EXP) | 0.3 (EXP) |
|---------|-----------|-----------|-----------|-----------|
| $S$ at E1 | 1.03 | 1.12 | 0.61 | 0.45 |
| $S$ at E2 | 1.08 | 1.03 | 0.63 | 0.47 |

#### 4.3. Pressure fluctuations in the draft tube diffuser

Pressure fluctuations at part load operation bring up a further issue. As a result of unfavorable flow conditions, fluctuations in pressure can cause unusual load conditions, heavy vibrations and noise emitted by the machine. The power spectrum of the recorded pressure signals $p_{w,t}$ of one transducer mounted in the draft tube diffuser (see fig. 2) is plotted in fig. 11 for different operating points. A significant increase of the amplitudes at very low and zero flow rates can be observed. At $Q/Q_N = 0.5$, the fluctuations are almost negligible compared to very small flow.
rates, as it is the case for higher flow rates. If a pump or a pump-turbine is operated at such low flow rates, this fact should be taken into account. The characteristic blade passing frequency at $\frac{f}{f_\Omega} = 7$ could not be detected, as probably the distance of measurement plane E1 to the runner inlet was too large. The peaks at very low frequencies, especially at $\frac{f}{f_\Omega} < 1$ can be interpreted as slow moving vortex structures in the draft tube.

**Figure 9.** Flow angle in the draft tube cone at E1.

**Figure 10.** Local swirl number at E1.

Further flow measurements were carried out in the stay vane channel opposite the nose vane. In fig. 12, the measurement lines (dotted) and directions of the measured velocities are explained. Figure 13 shows time-averaged velocity components of secondary flow in the stay vane channel along the three sample lines. The standard deviation of the mean value at a significance level of 95% is used to characterize the confidence level of the mean value of the experimental data. Positive and negative values might indicate fully separated flow, presumably caused by the onset of rotating stall phenomena. The different measurement planes can be seen in fig. 3, where the plane middle represents the reference plane of the machine at $B/b = 0.5$. Figure 13-right shows that the axial velocity component $C_z$ is around zero in the direction of the mean flow. At line 3 from $1 - x/X = 0.4$ to 1, a peaking standard deviation was obtained as only few counts were detected by LDV. Especially at small $1 - x/X$ values, an increasing $C_t$ towards pressure side of the stay vane is observable, as shown in fig. 13. Along line 3, the streamwise development of the tangential velocity shows a reversed direction of $C_t$, which is in good agreement with numerical

**Figure 11.** Waterfall plot of power spectra in the draft tube cone for the operating points $Q/Q_N = 0.0$ to $Q/Q_N = 0.5$.

### 4.4. Stay vane channel flow

Further flow measurements were carried out in the stay vane channel opposite the nose vane. In fig. 12, the measurement lines (dotted) and directions of the measured velocities are explained. Figure 13 shows time-averaged velocity components of secondary flow in the stay vane channel along the three sample lines. The standard deviation of the mean value at a significance level of 95% is used to characterize the confidence level of the mean value of the experimental data. Positive and negative values might indicate fully separated flow, presumably caused by the onset of rotating stall phenomena. The different measurement planes can be seen in fig. 3, where the plane middle represents the reference plane of the machine at $B/b = 0.5$. Figure 13-right shows that the axial velocity component $C_z$ is around zero in the direction of the mean flow. At line 3 from $1 - x/X = 0.4$ to 1, a peaking standard deviation was obtained as only few counts were detected by LDV. Especially at small $1 - x/X$ values, an increasing $C_t$ towards pressure side of the stay vane is observable, as shown in fig. 13. Along line 3, the streamwise development of the tangential velocity shows a reversed direction of $C_t$, which is in good agreement with numerical
predictions. In general, unsteady CFD captures the trend of the secondary channel flow to a reasonable degree of accuracy. In order to assess the streamwise flow in the stay vane channel, the pseudo-meridional component of the velocity at $1 - x/X = 1/6$ along a line $0 \leq y \leq Y$ is evaluated, shown in fig. 14. The data points and curve again represent the mean value of the velocity component. Figure 15 compares the mean velocities obtained from measurement and simulation. In experiments, the pseudo-streamwise velocity $C_m$ is almost linearly decreasing from the suction side to the pressure side of the stay vane channel, which is not accurately resolved by CFD. Figure 16 presents a streamline plot of the mean velocity in the reference plane with sample and measuring lines highlighted. When comparing with fig. 13 and fig. 15, a reasonable agreement of simulations and experiments can be observed. As plotted in fig. 16, three dimensional vortex structures of fully separated flow in the stay vane channel become apparent. The streamline plot shows a stagnation point attached to the pressure side of the stay

**Figure 12.** Stay vane flow passage with measurement lines.

**Figure 13.** Secondary velocities $C_t$ and $C_z$ in the reference plane of the machine.
Figure 14. Sample line for $C_m$ measurement and comparison.

Figure 15. Comparison of $C_m$, simulation to experiment.

Figure 16. Streamline plot in reference plane.

Figure 17. Streamline plot at stay vane channel cross section at $1 - x/X = 1/6$.

Vane and reversal flow near the stay vane channel outlet due to strong rotation of the flow. The fluid might also rotate in the stay vane channel as shown in fig. 17.

5. Conclusion

During start-up of reversible Francis pump-turbines, zero flow condition at a given gate opening can arise. In this paper, the zero flow condition is examined experimentally and numerically to gain a better insight into the specific flow patterns. It is shown that at no flow strong prerotation and reversal flow at the runner inlet occur. The inflow conditions can be characterized by a swirl number $S > 1$ and a flow angle $\beta > \pi/2$, which provides information about the reversal flow. Significant peaks in the wall pressure power spectrum similar to very low load pumping can be observed. The flow in a stay vane channel is also considered. LDV measurements and numerical simulations showed a complex three dimensional flow pattern indicated by negative and positive values of secondary and meridional velocity components. This might be the result of fully separated flow in the tandem cascade of the machine shown in streamlines plots. For a better characterization further transient analyses are necessary. As a result of all this unfavorable flow conditions, the caused loads should be considered for lifetime cycle assessments.

Nomenclature

\begin{align*}
  a & \quad \text{m} \quad \text{gate opening} \\
  A/A_{max} & \quad \text{dimensionless amplitude} \\
  b & \quad \text{m} \quad \text{measurement position} \\
  B & \quad \text{m} \quad \text{stay vane channel height} \\
  c & \quad \text{m s}^{-1} \quad \text{velocity} \\
  T_{\nu D} & \quad \text{torque factor} \\
  u_1 & \quad \text{m s}^{-1} \quad \text{circumferential speed} \\
  x, y & \quad \text{m} \quad \text{measurement position} \\
  x_j & \quad \text{m} \quad \text{cartesian coordinate} \\
  X, Y & \quad \text{m} \quad \text{length of sample line}
\end{align*}
Acknowledgments
The authors gratefully acknowledge the support and assistance of VOITH Hydro. This work was partially funded by the FFG - Austrian Research Promotion Agency (project no 827497).

References
[1] Breugelmans F and Sen M 1982 Proc. 11th Int. Pump Symp., Texas A&M Univ pp 165–180
[2] Eisele K, Muggli F, Zhang Z, Casey M, Sallaberger M and Sebestyen A 1998 Experimental and Numerical Studies of Flow instabilities in Pump-Turbine Stages, Proc. 11th Symp. Hydraulic Machinery and Cavitation, The Environmental Engineering Society of Singapore, Singapore p 8
[3] Sick M, Oram C, Braun O, Nennemann B and Coutu A 2013 Proc. Hydro 2013, Innsbruck, Austria p 11.03
[4] Anciger D, Jung A and Aschenbrenner T 2010 IOP Conf. Ser.: Earth and Environ. Sci. 12 012013
[5] Zhang Z 2010 IOP Conf. Ser.: Earth and Environ. Sci. 12 012010
[6] Kaupert K A, Holbein P and Staubli T 1996 J. Fluids Eng. 118(4) 685–691
[7] Gällisch J 2010 Centrifugal Pumps (Springer) ISBN 9783642128233
[8] Gentner C, Sallaberger M, Widmer C, Braun O and Staubli T 2012 IOP Conf. Ser.: Earth and Environ. Sci. 15 032042
[9] Menter F and Esch T 2001 16th Brazilian Congress of Mechanical Engineering (COBEM)
[10] Smirnov P and Menter F 2009 Journal of Turbomachinery 131 041010 (pages 8)
[11] Menter F, Kuntz M and Langtry R 2003 Turbulence, heat and mass transfer 4 625–632
[12] Shur M L, Spalart P R, Strelets M K and Travin A K 2008 International Journal of Heat and Fluid Flow 29 1638–1649
[13] Beér J and Chigier N 1972 Combustion aerodynamics Fuel and energy science series (Applied Science Publishers Ltd) ISBN 9780853345138
[14] Bender C 2011 Messung und Berechnung des Resonanzverhaltens gekoppelter Helmholtz-Resonatoren in technischen Verbreitungssystemen - (Karlsruhe: KIT Scientific Publishing) ISBN 9783866446588
[15] Weng M 2002 Experimentelle und Numerische Untersuchung von Gleichstromzyklen Ph.D. thesis RWTH Aachen