A CENTRIFUGAL FAN TEST BENCH FOR VALIDATION DATA AT OFF-DESIGN CONDITIONS

Balazs PRITZ, Johannes WALTER and Martin GABI

Karlsruhe Institute of Technology, Institute of Fluid Machinery,
Kaiserstr. 12, 76131 Karlsruhe, Germany

SUMMARY

In order to have reliable results from CFD simulations a validation of the acquired simulation tool and the composed numerical model is crucial. Especially in off-design conditions of turbomachinery it is very challenging for CFD to predict a strongly whirling, highly unsteady flow field correctly. In the present paper the construction of a test bench for a low pressure centrifugal fan with spiral casing is described and the first results are presented. The main goal is to provide extensive validation data not only in the design operation point but also for off-design conditions (part-load and over-load).

INTRODUCTION

Nowadays with eco directives becoming more strictly the efficiency improvement of turbomachinery is getting more important than ever. Relieving experimental investigations computational fluid dynamics (CFD) could be the major tool to optimize the geometry of a turbomachinery in the future, as it can provide a detailed insight into the flow within the impeller. In order to have reliable results from CFD simulations a validation of the acquired simulation tool and the composed numerical model is crucial. Especially in off-design conditions it is very challenging for CFD to predict strongly whirling, highly unsteady flow fields correctly.

In the last decades numerous numerical investigations were presented in the field of turbomachinery [1]. This reflects the obvious demand for more detailed information about the flow field in different complex situations. The main benefit of the numerical investigation is that it can provide an unrestricted insight into the flow within the impeller. Surprisingly large is the number of investigations where no validation is carried out. Actually the real mission of CFD is to predict the flow features without manufacturing prototypes and conducting expensive and time-consuming experiments with restricted insight into the turbomachinery. Can CFD provide this today? Investigations on a higher level perform a study of dependency on resolution of the spatial discretization or on the applied turbulence model at least. Beyond these they assume that the rest of numerical model is correct. But there are a plenty of other possible sources of error. Exemplarily they assume implicitly that the flow solver (which is in most cases a black box) is bug free, the size of time step is sufficiently small, the time integration is sufficiently long, the boundaries of the computational domain are far away enough from the problem of interest, there is no unphysical reflection at the boundaries, the neglected geometrical parts or details are really insignificant, etc. In contrast other investigations, which perform a validation by comparing the results with experimental ones, show that generally a perfect matching cannot be reached. For turbomachinery working at the design point (best efficiency point - BEP) in many cases a very good agreement can be achieved as attached flow is present in the main part of the investigated domain. In off-design conditions the simulations usually cannot capture the physics of strongly separated flow. In cases where the performance characteristics are predicted promisingly well, or at least their tendency, comparison of the flow in detail show larger discrepancies.

Hence there is a demand for experimental data set freely accessible for the community which provides benchmarking in a wide range of the operation conditions with as much detail about the flow field as possible. In order to fill this gap at least on the field of centrifugal fans a project was started at the Institute of Fluid Machinery at the Karlsruhe Institute of Technology (KIT) to construct a test bench that can provide the desired data set. Based on the expertise of the institute acquired during the past decades on this field the test case of a low pressure centrifugal fan in spiral casing was chosen.

The main goal in constructing the test case was to provide an extensive validation data set not only in the design operation point but also for off-design conditions. The geometry was designed based on classical design guidelines. It is very similar to industrial centrifugal fans however an additional aim was, to provide a geometry which allows an acceptable mesh generation effort and high mesh quality for the numerical investigation. Due to the rotational periodicity of axial and centrifugal fans without casing, both types can be calculated by regarding only a single blade passage, which decreases the number of required cells and hence the necessary computational power. For centrifugal fans with spiral casing this simplification is not suitable and this construct induces additional issues as pressure pulsation, acoustics, pulsating radial force, periodic load of vanes etc. The present test case should provide availability to investigate such phenomena also. For the numerical investigation it is a quite challenging test case, thus it will open a competition for the flow solvers to test and improve their efficiency.

From the numerous experimental investigations done for centrifugal fans we pick here two, which show some similarities with the present concept. The first one describes the geometry of the fan in detail, but, as in many cases, the flow field is measured only in the stationary parts [2, 3, 4]. The second one investigates the flow inside the impeller, but, as in many cases, the project was initiated from the industry and therefore the geometry is not available for the whole community [5, 6].

EXPERIMENTAL SETUP

The present concept in the first phase is to measure the characteristic and the efficiency curve of the fan with standardized methods conform to ISO 5801 [7] and to measure the flow field with Particle Image Velocimetry (PIV) in several different positions relative to the volute cut-off (tongue or corner) within the rotor vanes and in the spiral casing as well.

Test facility

The main focus of the investigations is on the reconstruction of the flow field by means of PIV (2D2C)[1]. In order to keep the seeding particles (DEHS oil droplets) within the system a closed loop test rig is constructed. The individual components of the test rig are shown in Figure 1.

The flow enters the inlet section (1) of the model fan (2) from the settling chamber (5). As the velocity is approximately zero in the settling chamber, the static pressure is approximately equal to the total pressure. By measuring the pressure here and in the discharge duct the pressure difference for the fan characteristics can be determined (3). The total pressure in the discharge duct can be calculated after the velocity has been determined from the volume flow rate measurement by an orifice (4). The model fan (MF) is driven by an electrical motor (M). The operating point of the model fan can be adjusted arbitrarily by means of a booster fan (6) and a throttle (7) close to the entrance within the settling chamber. Temperature (T) is measured in the inlet section and on the discharge side of the model fan as well.

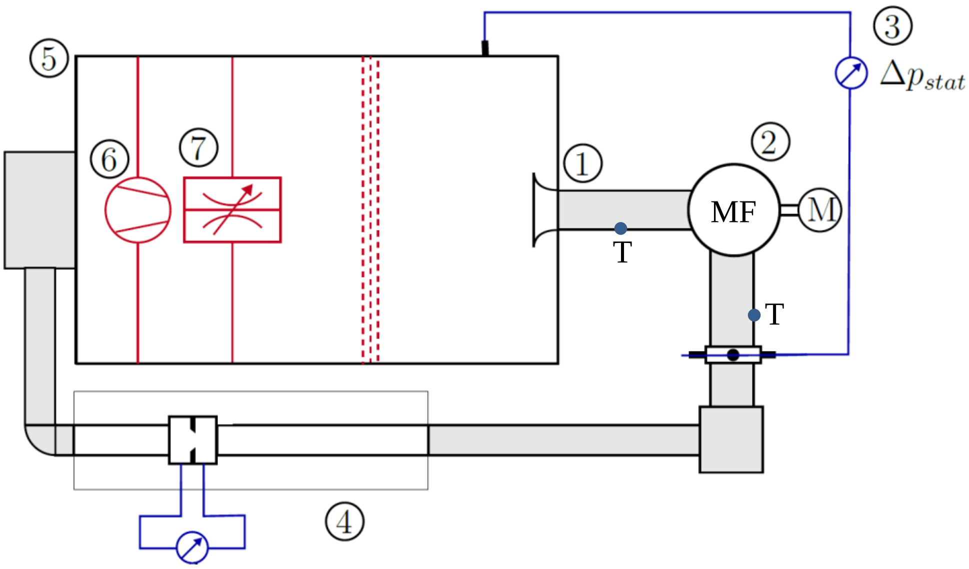

Figure 1: Test loop; 1- inlet section, 2- model fan, 3 – measurement of pressure difference, 4 – measurement of volume flow rate, 5 – settling chamber (with screens for flow homogenization), 6 – booster fan, 7 – throttle, MF – model fan, M - motor

Model fan

The test object is a centrifugal fan with nine backward-curved blades and a spiral casing (see Figure 2). The design of the fan is based on the classical design guidelines [8]. The generation of the geometry was performed with the in-house tool FanKIT [9]. The main characteristics of the centrifugal fan are summarized in Table 1 [10].

A special feature of the fan is the rotating diffuser (from D_2 to D_3), which was found to be advantageous for fans without casing [11]. For fans with casing the effect of a rotating diffuser is not straightforward and it is also a subject of the upcoming investigation series.

For the optical accessibility the hub of the impeller and the main parts of the casing are made out of acrylic glass (see Figure 2). The shroud and the blades have advanced geometry. They are manufactured in one part by a three-dimensional printing technology. The geometry of the blades is milled into the hub and the two parts are glued together.

---

[1] A measurement with stereo PIV (2D3C) was abandoned as the blades limit very strongly the optical access of the cameras in inclined positions.

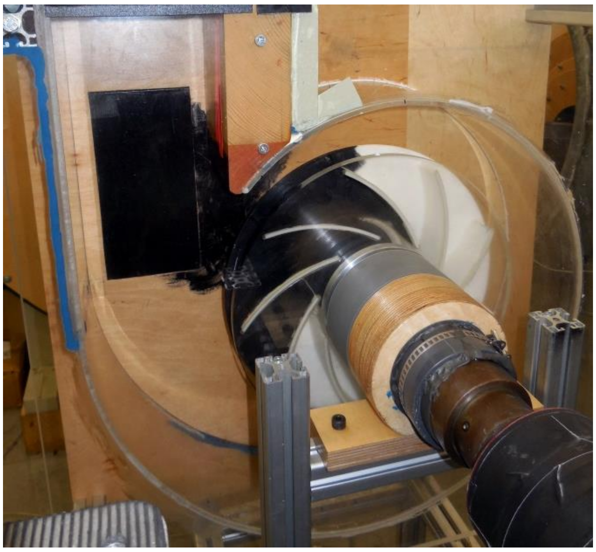

Figure 2: The model fan in the spiral casing

Table 1: Fan characteristics

Definition	Symbol	Value
Number of blades	z	9
Blade shape	-	Backward-curved, circular arc
Inlet angle of blade	β_{s1}	30.2°
Outlet angle of blade	β_{s2}	40.7°
Impeller inlet height	b_1	58.5 *mm*
Impeller outlet height	b_2	38.2 *mm*
Impeller inner diameter (blade leading edge)	D_1	138 *mm*
Diameter of blade trailing edge	D_2	306 *mm*
Impeller outer diameter	D_3	325 *mm*
Volute shape	-	logarithmic

In order to accomplish measurements in different positions relative to the volute cut-off, the test rig is designed so that the spiral casing can be rotated around the axis, while keeping the laser and camera in a fixed position.

A very important aspect when providing validation data for numerical simulations is that boundary conditions must be well defined, and as simple as possible. The simulation domain can end where the static pressure is measured in the discharge duct, and the measured value can be set simply for the outlet boundary condition. The inlet boundary conditions are more critical. In the case of industrial fans the flow is predominantly turbulent. Turbulent inlet boundary conditions are however very challenging. If turbulent values are measured and prescribed for the simulation, statistical models (Reynolds averaged Navier Stokes simulations), on the one hand, often fail to correctly predict the evolution of turbulent properties downstream of the inlet boundary. On the other hand, time resolving methods (large-eddy simulations - LES) need time dependent data set at the inlet. The techniques to provide such data sets are under development and they need generally high computational effort. Therefore the inlet section of the fan was designed to get well defined inlet conditions for the numerical simulations with negligibly small turbulence intensity. In order to accomplish this condition it was very important that the fan receives air directly from the settling chamber. The inlet section of the fan therefore contains a nozzle and a pipe. The nozzle is placed within the settling chamber, where the flow velocity has very low values. Even if there are some fluctuations in the settling chamber the nozzle accelerates the flow strongly, therefore an almost laminar flow is expected downstream of it. The pipe is long enough that possible flow disturbances (swirl) of the fan cannot reach upstream into the settling chamber, and short enough that the developing flow does not become turbulent. The chosen length (L_{IS}) of the pipe is approximately eight times the diameter (D_{IS}, see Figure 3).

First investigations show that, as the flow is steady in this section, it is possible to simulate the flow separately in this region by a two-dimensional axisymmetric simulation and extract results from a given position to provide velocity distributions at the inlet boundary for the three-dimensional simulations of the fan [12]. The uncertainty of the velocity profile at the inlet insignificantly influences the flow within the fan.

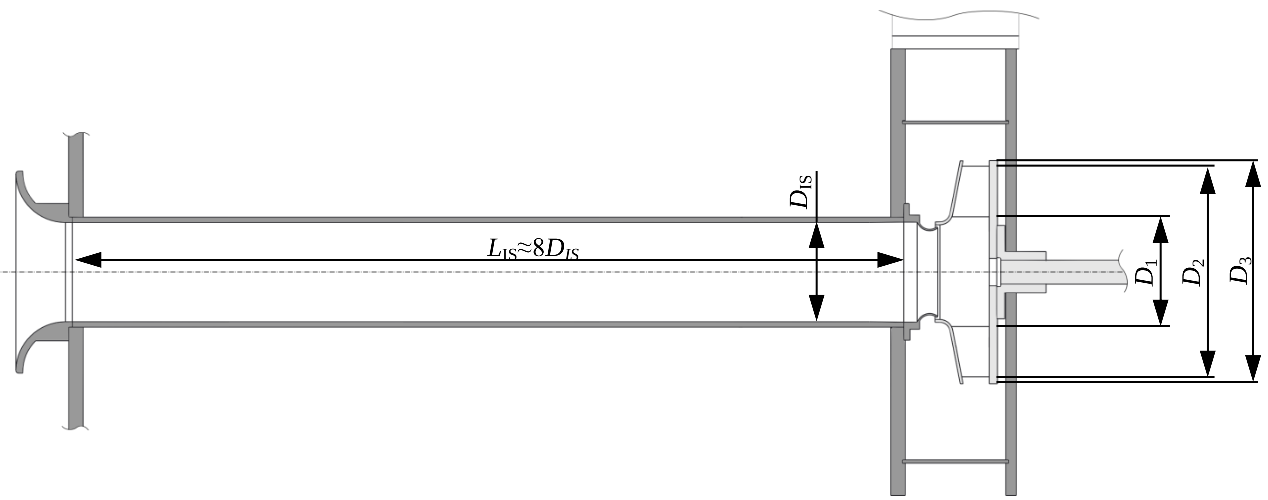

Figure 3: Inlet section, suction nozzle and impeller

A special feature of the configuration is that in the simulation almost every loss mechanism can be included[2]. Consequentially the efficiency curve from measurement and numerical simulation are very good comparable. Two exceptions are the mechanical loss of the bearing between the impeller and the shaft power measurement and general leakage of the test rig. Therefore the power consumption of the bearing was reduced as much as possible and documented carefully in dependence of radial and axial forces acting on the shaft [13]. Furthermore, the tightness of the test rig was improved in order to make the amount of leakage losses become marginal compared to the flow rate through the impeller [13].

FIRST RESULTS

Performance measurements

The characteristic curves of the fan are presented in non-dimensional form in Figure 4. They were obtained on the basis of the measurements of pressure difference ((3) in Figure 1), volume flow rate

[2] E.g. friction losses on each side of the impeller, leakage between suction side and discharge side of the impeller.

((4) in Figure 1) and shaft power (on the driving shaft between (MF) and (M) in Figure 1). The pressure coefficient Ψ_f, the flow coefficient φ and the efficiency η are defined according to Equations (1), where Δp_{tot} is the pressure rise, u_2 is the circumferential velocity at the trailing edge, $\dot{V}$ is the volume flow rate and P_{Shaft} is the shaft power.

$$\Psi_f = \frac{\Delta p_{tot}}{\frac{\rho}{2} \cdot {u_2}^2} \qquad \varphi = \frac{\dot{V}}{\frac{\pi}{4} \cdot {D_2}^2 \cdot u_2} \qquad \eta = \Delta p_{tot} \cdot \dot{V} / P_{shaft} \qquad (1)$$

The measurement was conducted first at the rotational speed of 600 *rpm* not to overstress the glued components of the impeller. Later higher rotational speeds were tested at 800 *rpm* and 1000 *rpm* to investigate Reynolds-number effects. A very good match of the pressure coefficient curves can be stated. However, the efficiency curves show some deviations, which cannot be cleared straightforward using the scaling law of Reynolds-number dependency and it needs further investigations.

Uncertainty of the curves will be presented at later time after the final design of the test facility is found.

An important aspect in the development of the test facility was to simultaneously conduct numerical simulations. Latter are described in more details in [12] and [14]. On the one hand, it was very important to test the boundary conditions as soon as possible. On the other hand they could help to eliminate some sources of error in the measurements. The numerical investigations plotted in Figure 4 are described in more detail in [15]. Furthermore, numerical simulations could help to design the measurement setup, hence to identify regions of interest indicating where and what should be measured.

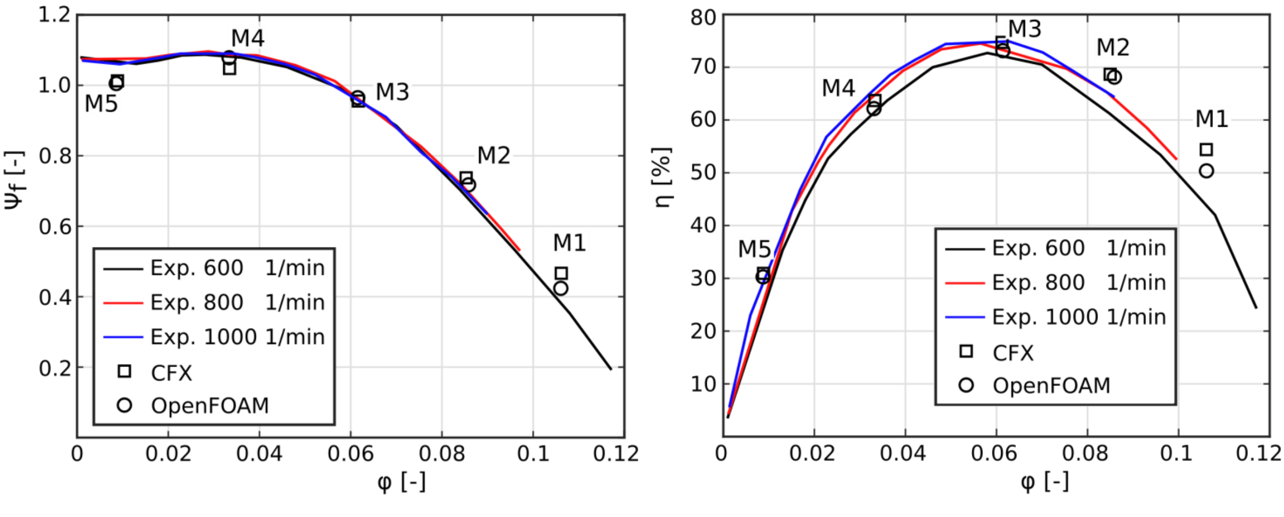

Figure 4: Pressure head and efficiency of the fan conducted from measurements at different rotational speeds and from numerical simulations

PIV measurement

The first measurements of the flow field by means of PIV were conducted at the rotational speed of 600 *rpm*. The region near to the volute cut-off was chosen to test the abilities of the experimental setup. One measurement plane was defined in the rotor vane at approximately the middle inlet height of the vane in axial position (at *h* = 24 *mm* to the hub, see Figure 5). The position of the vane relative to the volute cut-off was defined by the trailing edge of the blade with its suction side in the vane. It was rotated 40° from the vertical position opposite to the rotational direction (see Figure 5). Additionally a second plane was defined in the discharge duct at the same axial position. The two planes have a small overlapping region. The laser illuminates the measurement areas from the left in

"View A" of Figure 5. Measurements were carried out in the operation points M1 to M5 (see Figure 4). The measurements in the two planes were carried out subsequently.

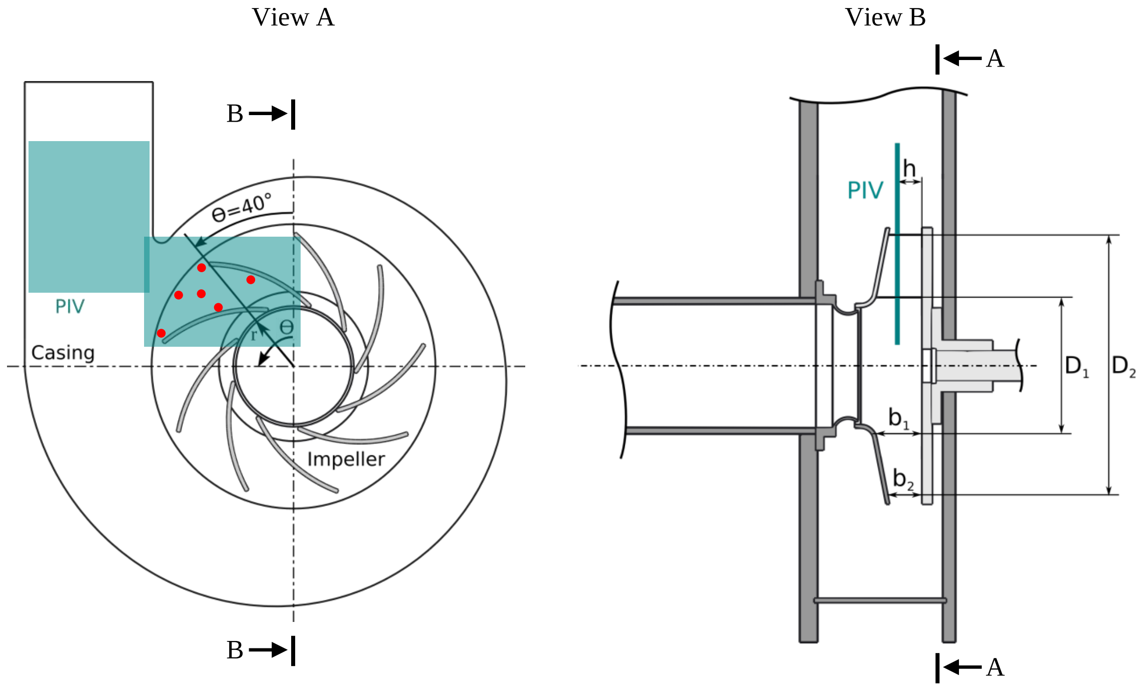

Figure 5: Position of the PIV measurement planes and probe points for the statistical significance (red dots)

In Figure 6, 7 and 8 the measured velocity fields are presented representatively in the operation points M1, M3 and M5, respectively. The contour plot of the absolute velocity c related to the reference velocity c_{ref} is shown. The reference velocity is defined as

$$c_{ref} = \dot{V} / \left(\frac{\pi}{4} \cdot {D_2}^2\right) = \varphi u_2 \,. \qquad (2)$$

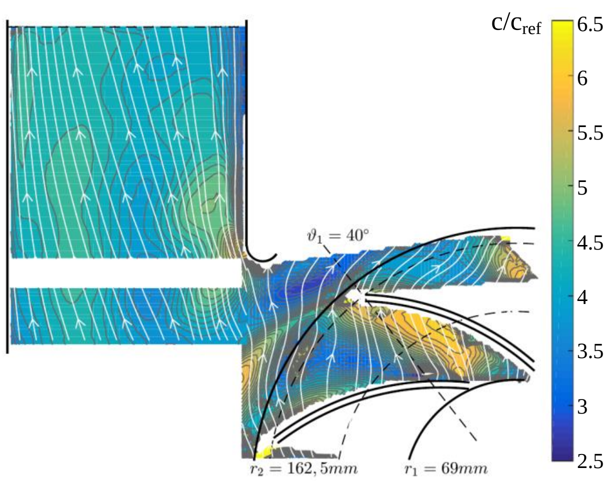

Figure 6: Normalized absolute velocity at M1

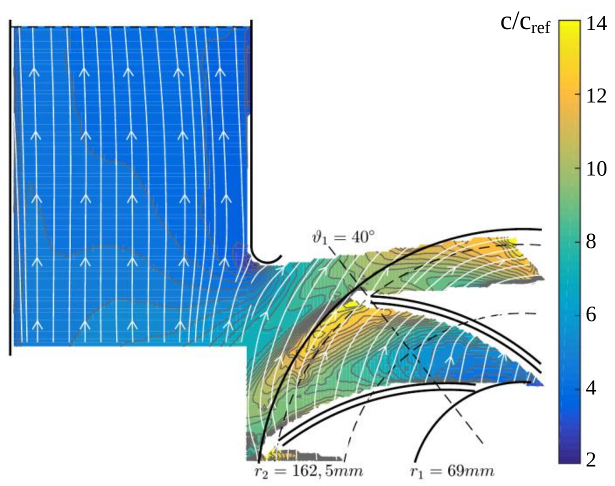

Figure 7: Normalized absolute velocity at M3

Each result was created as a phase average from 1500 image pairs. The statistical significance of the measured data is investigated at different probe points (red dots in Figure 5). Figure 9 shows representative for the operation point M3 the coefficient of variation (CV), which is defined as the ratio of standard deviation σ to the mean value μ, in function of the number of samples (image pairs). In most of the probe positions a converged value of CV is reached already with 100 samples, while it can be stated that CV converged in all positions with 1500 samples (more detail can be found in [10]). Beyond this the high quality of the measurement is demonstrated in the overlapping region where the agreement of the two subsequent measurements is very good.

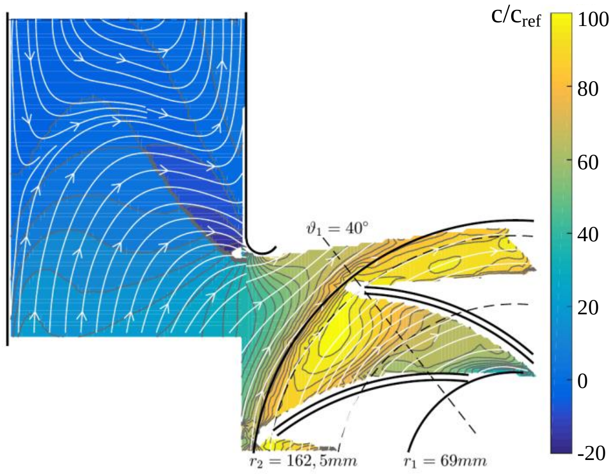

Figure 8: Normalized absolute velocity at M5

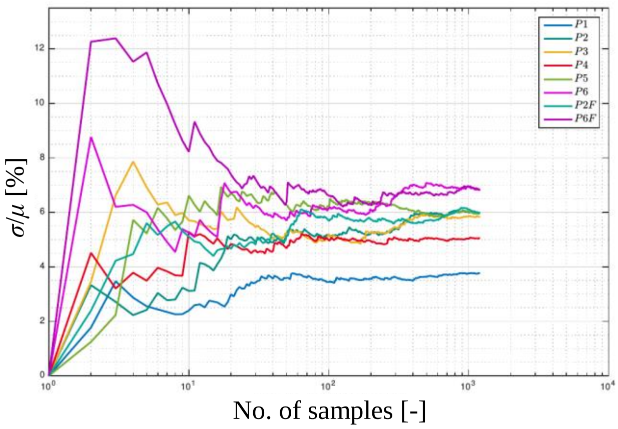

Figure 9: Coefficient of variation (CV) at different probe positions (red dots in Figure 5)

From the measured absolute velocity c the relative velocity w can be calculated via subtraction of the circumferential velocity u in every location. The contour plot of the relative velocity related to the reference velocity c_{ref} is shown exemplarily for the operation point M3 and M1 in Figure 10. Simultaneously the experimental results are compared with numerical simulations by means of velocity vectors. Corresponding to the characteristic curve the agreement between measurement and numerical simulation is much better close to BEP (M3) and more discrepancy can be found far from BEP, in this example, in over-load region (M1).

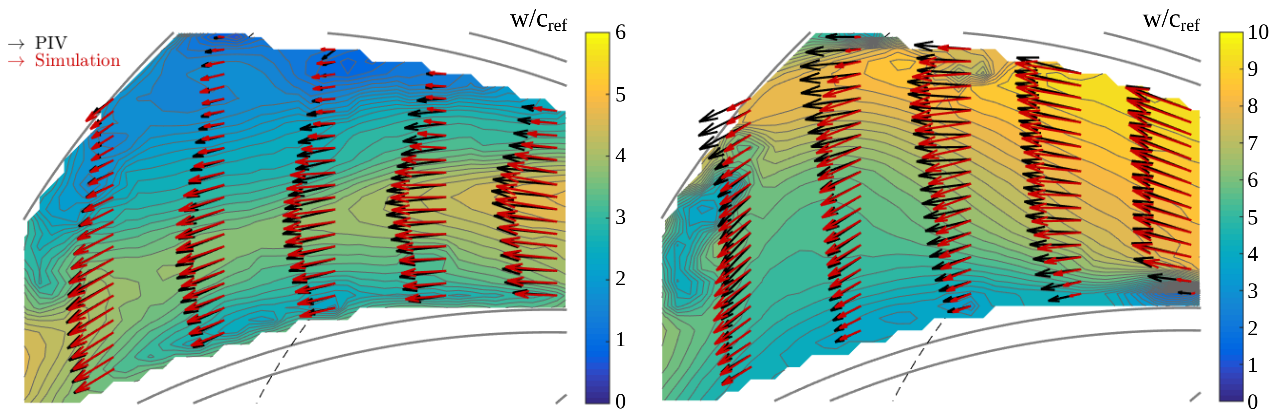

Figure 10: Normalized relative velocity as contour plot and comparison of measurement and numerical simulation (OpenFOAM®) by means of velocity vectors at M3 (left) and M1 (right)

CONCLUSIONS

The development of a test rig for a low pressure centrifugal fan with spiral casing is started to generate comprehensive data set for validations of numerical simulations. The first design of the impeller has nine blades to get lower slip, thus the flow in the impeller is more stable. Consequently a relatively good agreement between measurement and numerical simulations is found. However, the predicted slope of the characteristic curves is still slightly different and vector plots show the same tendency. As expected the discrepancies are larger in part-load and over-load, respectively.

The next design of the impeller will have seven blades, thus more instability of the flow and larger discrepancy between measurement and numerical simulation is expected. Beyond that less blades increase the availability for illumination at the leading edge.

After various improvements of the test facility a large series of data extractions at numerous axial positions in the rotor vanes is currently in progress.

REFERENCES

[1] J. Konrad – *Aktuelle Entwicklungen im Bereich Validierung der Simulationen von Strömungsmaschine (A review of recent progress in validation of turbomachine simulations)*. Bachelor thesis, Institute of Fluid Machinery, Karlsruhe Institute of Technology, **2016**

[2] S. Fortuna, K. Sobczak – *Numerical and experimental investigation of the flow in the radial fan.* Mechanics, Vol. 27, No. 4, pp. 138–143, **2008**

[3] S. Fortuna, J. Górski, T. Siwek – *Thermoanemometrical study of flow structure through a centrifugal fan*. Mechanics and Control, Vol. 32 No. 3, **2013**

[4] T. Siwek, J. Górski, S. Fortuna – *Numerical and experimental study of centrifugal fan flow structures and their relationship with machine efficiency*. Pol. J. Environ. Stud., Vol. 23, No. 6, pp. 2359-2364, **2014**

[5] A. Lucius, A. Lehwald, G. Brenner, D. Thevenin – *Investigation of unsteady flows in a centrifugal fan using high-speed PIV and numerical simulations*. Proceedings of Fan 2012, International Conference of Fan Noise, Technology and Numerical Methods, Senlis, France, **2012**

[6] A. Lucius, A. Lehwald, D. Thevenin, G. Brenner - *Experimental and numerical analysis of flow instabilities in a radial fan*. Proceedings of ASME Turbo Expo 2014: Turbine Technical Conference and Exposition, Düsseldorf, Germany, **2014**

[7] ISO 5801 – Industrial fans – Performance testing using standardized airways, **2007**

[8] L. Bommes – *Ventilatoren*. Vulkan-Verlag GmbH, **2003**

[9] H. Ratter – *Auslegung und Optimierung gehäuseloser Radialventilatoren*. PhD thesis, Institute of Fluid Machinery, Karlsruhe Institute of Technology, Germany, **2013**

[10] B. Driedger – *Experimentelle Untersuchung eines Radialventilator-Modells (Experimental investigation of a generic radial fan model)*. Master thesis, Institute of Fluid Machinery, Karlsruhe Institute of Technology, Germany, **2016**

[11] Y. Xia, S. Caglar, H. Ratter, M. Gabi – *Maximum achievable efficiency of centrifugal fans without housing*. Proceedings of Fan 2012, International Conference of Fan Noise, Technology and Numerical Methods, Senlis, France, **2012**

[12] V. Krämer – *Numerische Untersuchung eines generischen Radialventilator-Modells (Numerical investigation of a generic radial fan model)*. Master thesis, Institute of Fluid Machinery, Karlsruhe Institute of Technology, **2016**

[13] D. Sauerer – *Konstruktive Überarbeitung eines Prüfstandes für einen radialen Ventilator (Design improvement for a test rig of a centrifugal fan)*. Master thesis, Institute of Fluid Machinery, Karlsruhe Institute of Technology, **2017**

[14] T. Rohde – *Numerische Untersuchung eines generischen Radialventilator-Modells mit komerzieller und freier Software (Numerical investigation of a generic radial fan model by means of commercial and free solvers)*. Master thesis, Institute of Fluid Machinery, Karlsruhe Institute of Technology, Germany, **2016**

[15] J. Walter, F. Trimborn, B. Pritz, V. Krämer, M. Gabi – *Numerical investigation of a centrifugal fan*. In Proceedings of ISAIF-13: International Symposium on Experimental and Computational Aerothermodynamics of Internal Flows, Okinawa, Japan, **2017**