Influence of the Runner Gap on the Flow Field in the Draft Tube of a Low Head Turbine

Bernd Junginger¹, Stefan Riedelbauch¹

¹Institute of Fluid Mechanics and Hydraulic Machinery, University of Stuttgart, Pfaffenwaldring 10, 70550 Stuttgart, Germany
E-mail: bernd.junginger@ihs.uni-stuttgart.de

Abstract. The gap flow of axial turbines is usually neglected in the design process of hydraulic machines, although it can lead to a stabilization of the draft tube flow. Though, this negligence of the gap can falsify the flow field in the draft tube. Presented in this paper are simulations of an axial propeller turbine operated at \( \Delta \gamma = \Delta \gamma_{BEP} \) with \( Q > Q_{BEP} \). Simulations of four gap sizes, using a mesh with about 15 million elements for the entire machine, are performed. Additionally, two turbulence models are applied, the k-\( \omega \)-SST and the SAS-SST model. At the evaluated operating point a full load vortex develops. Depending on the turbulence model the developing vortex rope can either arise from the hub in a straight shape or in a shape resembling a corkscrew. Integral quantities such as head and torque are compared with experimental model test results performed in the laboratory of the Institute. Flow field simulation results are evaluated for different gap widths. Furthermore, the impact of the gap flow respectively the gap size can be observed in velocity profiles evaluated at different positions downstream the runner until to the end of the draft tube cone. Moreover, the pressure signals recorded at the beginning of the draft tube cone are also affected by the gap flow.

1. Introduction

The Water Framework Directive advises all members of the European Union to achieve a good ecological status for all flowing water [1]. Unused dams and weirs, where the installation of small hydro turbines is suitable, were unprofitable before the introduction of the Water Framework Directive. These spots are back in the focus of national energy providers due to promotions given by the European Union. Owing to the energy revolution in Europe, the amount of renewable energy produced by photovoltaic and wind energy is significantly increasing. Fluctuations of the electric grid induced by those generation types have to be balanced to ensure the power system stability. Hydro power plants are very well suited to balance the fluctuations of the grid due to their good predictability and the fast adjustability of the energy output. However, the operating range of hydraulic machines is getting larger, the machines are operated in off-design conditions and thus, the risk of occurring transient phenomena like vortex ropes in the draft tube, cavitation, etc. is increasing. The share of renewable energy in Germany in the year 2014 was about 25.8% and is increasing to about 30% in 2015 [2]. Hydro power contributes about 3.1% to the total energy mix in Germany in the year 2014.

The overall performance of low head turbines like Kaplan, bulb and propeller turbines strongly depends on the effectiveness of the draft tube. The runner gap has an effect on the draft tube flow field which cannot be neglected for accurate flow field prediction. In the design process
typical geometrical simplifications carried out are the negligence of gaps between runner and shroud and, if occurring, at the trailing edge of the guide vanes. Moreover, the computational costs are usually reduced by numerical simplifications like circumferential averaging of the flow field at the interfaces between stationary and rotating machine parts with the target to obtain steady state computations. The gap flow between runner and shroud, however, can lead to a stabilization of the flow field in the draft tube.

In this paper a propeller turbine with 4 runner blades is investigated. The analyzed operating point has the characteristic values \( n'_{1} = 0.78n'_{1BEP} \) and \( Q'_{1} = 0.89Q'_{1BEP} \). The guide vane opening is \( \Delta \gamma = \Delta \gamma_{BEP} \). For the analysis transient simulations with and without runner gap are performed and compared with results obtained from experimental measurements. A model sized propeller turbine is installed in the closed loop in the laboratory of the Institute of Fluid Mechanics and Hydraulic Machinery at the University of Stuttgart. The experimental measurements follow the IEC 60193 which is the standard for model acceptance tests of hydraulic machines [3]. For the investigated operating point a vortex rope in the draft tube develops in the shape of a full load vortex. A full load vortex can arise in a symmetrical or asymmetrical shape [4]. The vortex rope develops from a low pressure zone at the runner hub and is developing downstream into the cone of the draft tube. In addition to the runner gap the gap at the trailing edge of the guide vanes is also modelled in all investigated computational models. The gap at the guide vane trailing edge, discretized in the numerical model, is equivalent to the gap of the model turbine. The investigated normalized gap sizes for the runner gap are \( \tau = 0, \tau = 2.0, \tau = 3.3 \) and \( \tau = 6.7 \). The normalized gap size \( \tau \) is defined as

\[
\tau = \frac{sc_{cl}}{D}
\]

with the runner gap size \( s \), the runner diameter \( D \) and a normalizing factor \( c_{cl} \). The diameter of the runner for both the computational model and in the experiment is \( D = 0.3m \) with a Reynolds number of \( Re = 4 \cdot 10^6 \).

2. Numerical Setup
The simulations are carried out using the commercial CFD (Computational Fluid Dynamics) code Ansys CFX Version 16.0. All computational models are representing the original geometry of the model sized turbine except for the simulation without runner gap (\( \tau = 0 \)). A comparison of two turbulence models is carried out for a normalized gap size of \( \tau = 2.0 \). The analyzed turbulence models are the \( k-\omega \)-SST and the SAS-SST model. The first analyzed turbulence model, the \( k-\omega \)-SST model is a standard RANS (Reynolds-Averaged-Navier-Stokes) model. The second investigated model, namely the SAS-SST model, is a hybrid RANS-LES (Large Eddy Simulation) model. The hydraulic contour without the expansion tank is illustrated in Fig. 1.

A mass flow inlet boundary condition set for all simulations originates from the experiment. At the outlet a pressure boundary condition is set. The evaluation spots for the velocity profiles and pressure measurements in the draft tube cone are shown as straight lines. The evaluation lines for the velocity profiles are located downstream of the runner trailing edge at \( \frac{D}{4} \) (L1=yellow line), \( \frac{D}{2} \) (L2=green line), \( D \) (L3=black line) and \( 2D \) (L4=red line). Wall pressure signals are analyzed at \( D \) downstream the runner at the beginning of the draft tube cone.
Table 1: Number of elements for all turbine components for a gap of \( \tau = 2.0 \)

| Turbine part                               | Number of elements |
|--------------------------------------------|--------------------|
| Guide Vanes                                | 3.3M               |
| Runner                                     | 5.9M               |
| Draft Tube with expansion tank             | 4.9M               |
| Total                                      | 14.1M              |

Number of nodes in runner gap 20

For the RANS turbulence model a high resolution schemes is used for the advection term, whereas a bounded central differencing scheme (BCD) is applied for the advection term for the hydrid RANS-LES turbulence model [5], [6]. The temporal discretization is performed by a second order Euler backward scheme. For the SAS-SST model a BCD scheme is used for the spatial discretization [7]. The temporal discretization of the turbulence quantities is performed by a bounded second order Euler backward scheme while a first order scheme is used for the spatial discretization of the turbulence quantities [6]. A detailed listing of the number of nodes for the investigated mesh is plotted in Tab. 1. Additionally, the number of nodes in the gap is listed in Tab. 1. The mesh in the gap is crucial to resolve gap flow and its influence on the draft tube flow field. The discretization of the runner gap at the leading and trailing edge is illustrated in Fig. 2.

3. Turbulence Modeling

Two turbulence models are applied for the analysis. The k-\( \omega \)-SST model is a RANS model which is the standard turbulence model used for design activities of hydraulic machines. Additionally, the SAS-SST model, a hybrid RANS-LES turbulence model is applied which is able to switch between RANS and LES like behavior as a function of various influencing variables.

3.1. k-\( \omega \)-SST

The k-\( \omega \)-SST model is a two equation turbulence model based on the Boussinesq hypothesis for solving the turbulent quantities using the Boussinesq eddy assumption [8]. Using the Boussinesq assumption, which implies that the Reynolds stress tensor \( \tau_{ij} \) is proportional to the main shear strain rate tensor \( S_{ij} \), it can be written as:
\[-\rho u_i' u_j' = \mu_t \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} - \frac{2}{3} \frac{\partial U_k}{\partial x_k} \delta_{ij} \right) - \frac{2}{3} \rho k \delta_{ij} \] (2)

Hence, closure to the set of equations can be achieved. A great advantage of the k-ω-SST against other RANS models is the combination of advantages of the k-ε and the k-ω model. In near wall regions no additional damping function has to be introduced to the k-ω formulation. A blend function switches from the k-ω to the k-ε model so the advantages of the k-ε formulation can be used for the core flow to avoid the known problems of the k-ω formulation of being too sensitive to inlet-free-stream turbulence properties.

### 3.2. SAS-SST

The SAS-SST model is a hybrid turbulence model which can switch between RANS and LES. For smaller turbulence scales the model can switch to SRS (Scale Resolving Simulation) mode [9]. An additional source term \( Q_{SAS} \) has been introduced in the transport equation of the turbulence eddy frequency \( \omega \) into the RANS model leading to following equation [10], [11], [12]:

\[
\frac{\partial \rho \omega}{\partial t} + \frac{\partial}{\partial x_j} (\rho U_j \omega) = \alpha \omega \frac{\partial \omega}{\partial x_j} + \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_j} \right] + (1 - F_1) \frac{2 \rho \omega}{\sigma_{\omega^2}} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} + \frac{Q_{SAS}}{\rho \sigma_{\omega^2}} \left( \frac{\partial^2 U_i}{\partial x_k^2} \frac{\partial^2 U_j}{\partial x_l^2} \right) - \rho \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \] (3)

The source term \( Q_{SAS} \) is defined as:

\[
Q_{SAS} = \max \left[ \rho \kappa \rho S^2 \left( \frac{L}{L_{uK}} \right)^2 - \frac{2 \rho k}{\sigma_{\Phi}} \right] \left( \frac{1}{\omega^2 \frac{\partial \omega}{\partial x_j} \frac{\partial \omega}{\partial x_j}} \frac{1}{k^2 \frac{\partial k}{\partial x_j} \frac{\partial k}{\partial x_j}} \right), 0 \] (4)

containing the turbulent length scale \( L \) and the von Karman length scale \( L_{uK} \) which is defined as:

\[
L_{uK} = \kappa \left| \frac{\overrightarrow{U'}}{U'} \right|, \quad U' = \sqrt{\frac{\partial^2 U_i}{\partial x_k^2} \frac{\partial^2 U_j}{\partial x_l^2}} \quad \overrightarrow{U'} = S = \sqrt{2 S_{ij} S_{ij}}, \quad S_{ij} = \frac{1}{2} \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \] (5)

The von Karman length scale is an essential quantity for the SAS-SST model to switch to SRS mode and hence an LES like behavior [13]. A reduction of the turbulent eddy viscosity leads to smaller turbulent structures down to the grid limit. The grid cell size \( \Delta \) is a limiter for the turbulent eddy viscosity to control the damping of the smallest revolved fluctuations [6]. DES (Detached Eddy Simulation) models do not have the major advantage of the SAS-SST to operate in RANS mode if the grid density and time step are too coarse.

### 4. Results

In total four gap sizes and two turbulence models are analyzed in this paper. A comparison of the integral quantities head and torque is carried out. The turbine head is compared to experimental results performed in the lab of the Institute. The main focus is placed on the evaluation of the influence of the gap flow and moreover the effects of the analyzed variables on the overall draft tube flow field. The velocity components are evaluated at four spots downstream the runner marked in Fig. 1. An analysis of the evaluated turbulence models and the runner gap size on the shape of the vortex rope and the turbulence structures is carried out. Additionally, the pressure signal for the different gap widths is evaluated in the draft tube cone. All analyzed simulations are time averaged over 50 runner revolutions.

The effect of the gap widths on the cavitation at the leading edge of the runner blade is not evaluated in detail. Leading edge cavitation develops on the suction side of the runner blade for
the investigated operating point. With an increasing runner gap size the cavitation volume at the leading edge is reduced, however cavitation still exists. The hybrid RANS-LES turbulence model predicts a higher cavitation volume than the RANS turbulence model for the same gap size.

4.1. Integral Quantities
The turbine head is validated with experimental results. Hence, all numerical results are normalized with the result measured at the model turbine. Due to the lack of experimental data the torque of the runner is normalized to the results of the gap size $\tau=2.0$ using the $k-\omega$-SST turbulence model. The normalized results are listed in Tab. 2.

| $\tau$=0 SST | $\tau$=2.0 SST | $\tau$=2.0 SAS | $\tau$=3.3 SST | $\tau$=6.7 SST |
|-------------|----------------|----------------|----------------|----------------|
| $h_{\text{CFD}} - h_{\text{Exp}}$ [%] | 4.2 | 2.5 | 3.0 | 0.9 | -1.9 |
| $T_{\text{CFD}} - T_{\text{CFD \_ref}}$ [%] | 2.4 | - | 0.5 | -1.6 | -4.8 |

Up to a gap width of $\tau=3.3$ the head is overestimated by all simulations. Only the largest investigated gap size of $\tau=6.7$ has a head lower than measured in the experiment. The scale resolving turbulence model, which is applied on a gap size of $\tau=2.0$, leads to a larger deviation of the head than the standard RANS model. The large deviation between the two investigated turbulence models originate from the head loss in the draft tube and the different flow fields downstream the runner. The losses in the other machine parts are on the same level. The torque is decreasing with an increasing gap width due to the gap flow. Fluid passes the runner section without generating torque. The deviations between the analyzed gap sizes originate from the gap flow and the different turbulence models, since the wall resolution of the runner is equal for the meshes. Additional measurements of the torque are planned to verify the numerical results.

4.2. Gap Flow
In Fig. 3 time averaged axial, circumferential and radial velocity profiles at four evaluation lines, illustrated in Fig. 1, are shown. First, the influence of the gap size is analyzed applied the k-ω-SST turbulence model is discussed. After this a comparison of the influence of the turbulence model on the velocity profiles is carried out.

At evaluation line L1, which is positioned at $D_4$ behind the runner the influence of the gap size can be seen in all three velocity profiles when applying the k-ω-SST turbulence model. The axial velocity is almost constant for a runner without gap when the radius is smaller than $0.8 \frac{R}{R_{\text{ref}}}$. The gap flow results in a reduction of the axial component close to the shroud at a smaller radius. This effect increases with an increasing gap size. In the circumferential velocity profiles of L1 the runner gap leads to a reduction of the magnitude of the circumferential velocity close to the shroud. With an increasing gap width a shift of the local minimum to a smaller radius can be observed. Moreover, a reduction of the magnitude of the circumferential velocity for the entire circumferential velocity profile is developing when the gap size is increasing. The gap size can also be seen in the radial velocity components close to the shroud. The velocity profiles close to the shroud possess a point of inflection for all gap sizes. The slope close to the point of inflection is decreasing with an increasing gap size.
At the second evaluation line L2, at $\frac{D}{2}$ downstream the runner, which is just behind the hub of the runner, the characteristics of the axial and circumferential component have in principle the same shape at the shroud. The effects of the gap flow respectively the gap size are still clearly visible in these velocity profiles. The differences in the axial and circumferential velocity are about the same as at L1. Additionally a stagnation region behind the outlet hood arises with the approximate diameter of the runner hub. A back flow in the axial component can be observed in the center of the draft tube. The circumferential component is reaching the maximum magnitude at the boundary of the stagnation region. In the radial velocity component two effects can be observed, the influence of the gap flow close to the shroud and the influence of the runner hub on

Figure 3: Velocity profiles at the evaluation spots L1, L2, L3 and L4
the velocity profile in the center of the draft tube. Furthermore, a change of the effect of the gap flow on the radial velocity components can be observed. The change concerns the magnitude of the velocity. With an increasing gap size the magnitude of the radial velocity close to the shroud is increases. An additional peak of the radial velocity develops close to the rotation axis caused by the end of the runner hub.

Evaluation line L3 is placed at \( D \) behind the runner in the draft tube cone. The diameter of the stagnation region in the center of the draft tube, in which the vortex rope arises has about doubled. This can be seen in all three velocity components. The shape of the axial components is expect for the size of the stagnation region comparable to the shape of the axial component at L2, including a small region in the center of the draft tube where back flow is occurring. The effects of the gap size are also comparable to L1 and L2. The influence of the different sizes on the circumferential velocity components in the stagnation region is amplified, whereas the effects close to the shroud are equal to evaluation lines L1 and L2. The radial velocity profiles are smoothed out between the different gap widths except in the region close to the shroud. The shape of the velocity profiles close to the shroud remain almost unchanged, while the differences of the velocity profiles induced by the gap sizes reduces compared to L1 and L2.

The fourth evaluation line L4 is located at \( 2D \) behind the runner at the end of the draft tube cone shortly before the draft tube is transitioning from a round contour to a rectangular contour. The reduction of the velocity, caused by the area enlargement in the draft tube, leads to a smoothing of the peaks of the velocity profiles by increasing distance to the runner. In the stagnation region in the center of the draft tube, which is clearly visible in L2 and L3, the flow field is getting progressively balanced. However, the effects induced by the gap flow can still be observed at \( 2D \) downstream the runner, in the time averaged velocity profiles. The same conclusion as for the axial components can be drawn for the circumferential velocity components. For the radial velocity profiles rather large deviations between the different gap sizes occur. Additionally an unsorted rank of the velocity profiles can be noticed compared to the previous analyzed evaluation lines L1, L2 and L3. An explanation for this could be a too short averaging period, which can lead to such phenomena, as a result of the unstable flow and occurring separations in the second part of the draft tube for the investigated operating point.

For a gap width of \( \tau = 2.0 \) a comparison of the two investigated turbulence models is performed. At evaluation line L1 the differences between the turbulence models are rather small. The biggest differences can be observed in the three velocity components close to the shroud. In total a good agreement of the velocity profiles between the two turbulence models at L1 can be noted. The first major derivations of the velocity can be seen at L2, where the stagnation region in the center of the draft tube is developing to a larger radius. At L3 the radius of the stagnation region, using the SAS-SST model, is almost double the size compared to the k-\( \omega \)-SST model. It can be stated, when comparing the two turbulence models, that in particular in the first part of the draft tube the overall flow field is significantly different. At L4 the differences between the turbulence models still exist, but due to the decreasing velocity component the velocity profiles are more and more smoothed out. A comparison of the standard deviation shows that the fluctuations of the velocities are significantly larger for the hybrid turbulence model than for the standard RANS model. The fluctuations of the velocity profiles, quantified with the standard deviation, reduce with an increasing distance to the runner. Laser-Doppler-Anemometry measurements are planned at specified spots in the draft tube in order to validate the numerical results.

4.3. Vortex Rope

The developing vortex rope in the draft tube for a runner gap of \( \tau = 2.0 \) for the two investigated turbulence models is illustrated in Fig. 4. Significantly differences of the size of the stagnation region can be observed in Fig. 3. The stagnation region when using the SAS-SST model has about twice the radius than the stagnation region when applying the k-\( \omega \)-SST turbulence model.
When using the RANS turbulence model the vortex rope develops a straight shape, which is the typical shape of a full load vortex. The shape of the vortex rope arising, when using the SAS-SST model, reassembles a corkscrew and, hence, looks like a part load vortex. In contrary to a part load vortex the stagnation region is only in the very center of the draft tube. In part load points the stagnation point respectively the vortex rope reduces the area where the fluid can flow significantly. The influence of the runner gap on the shape of the straight vortex rope, developing when using the RANS turbulence model is negligible. Vectors plotted on a cutting plane positioned at the same axial coordinate as evaluation line L3 are illustrated in Fig. 4. The shape of the vortex rope developing using the k-ω-SST turbulence model is round with the vortex core in the center of the draft tube. For the simulation with the SAS-SST model a vortex rope with an oval shape is arising. The vortex core is no longer positioned in the center of the draft tube. Due to the asymmetric shape of the vortex the flow field in the cutting plane is not as smooth as the flow field of the simulation using the k-ω-SST model.

4.4. Turbulence Quantities

For the quantification of the resolved turbulence scales the velocity invariant \( Q = \frac{1}{2}(\Omega^2 - S^2) \) is used with the absolute value of the strain rate \( S \) and the absolute value of the vorticity \( \Omega \) [14], [6]. A comparison of the turbulence quantities for a runner with a runner gap of \( \tau = 2.0 \) is shown in Fig. 5. The SAS-SST model is able to resolve finer turbulence structures than the k-ω-SST model. In particular in the draft tube cone a good resolution of the turbulence structures is achieved even though a relatively coarse grid density is used. The k-ω-SST model is only capable to resolve large flow structures in the entire draft tube. A good indicator of the LES content is the eddy viscosity ratio which has been plotted as variable on the isosurface in Fig. 5. A high viscosity ratio can be observed for the hybrid RANS-LES model in regions of the vortex breakdown and at the end of the draft tube. The resolved turbulence structures in these regions are relatively large. Compared to the hybrid RANS-LES model the viscosity ratio in the draft tube is very high, in particular in the second part of the draft tube. This is due to the occurring separations caused by the draft tube shape.

4.5. Pressure Gauge

Evaluations of dynamic pressure at L3 show that the runner gap size also has an influence on the amplitude of the pressure signal. The amplitudes of pressure signals are listed for different gap
widths for a normalized frequency of $3.98f_{\text{Runner}}$ in Tab. 3. The pressure amplitude is normalized using the head measured in the experiment. The highest pressure fluctuation amplitude can be observed for the SAS-SST model. An increase of the amplitude linked to an increasing gap size can be observed. For computations without runner gap ($\tau=0$) the effects on the pressure amplitudes are very small. However, starting from the actual model size gap $\tau=2.0$ the effects of the gap on the pressure amplitude may have to be taken into account for operation points where pressure fluctuations in the draft tube cannot be neglected. Additional experimental measurements of the pressure signal are needed for further validation.

Table 3: Pressure amplitudes at evaluation line L3

| $\tau$ | $\frac{\Delta p}{\rho g H_{\text{Exp}}} [%]$ |
|--------|------------------------------------------|
| $\tau=0$ SST | 0.32 |
| $\tau=2.0$ SST | 1.01 |
| $\tau=2.0$ SAS | 3.01 |
| $\tau=3.3$ SST | 1.13 |
| $\tau=6.7$ SST | 2.50 |

5. Summary and Conclusion
Simulations of an operation point with the characteristic values and $n'_1 = 0.78 n'_{1\text{BEP}}$, $Q'_1 = 0.89 Q'_{1\text{BEP}}$ for a guide vane opening of $\Delta \gamma = \Delta \gamma_{\text{BEP}}$ for a 4-bladed axial propeller turbine, varying the runner gap size, are performed. The entire machine is discretized with about 15 million elements. A comparison of two turbulence models, namely the k-\omega-SST and the SAS-SST model is carried out for a normalized runner gap of $\tau=2.0$.

The integral quantities, head and torque, are an indicator of the runner gap width. With increasing gap size both analyzed quantities reduce. For the largest investigated gap width of $\tau=6.7$ the torque and head are no longer overestimated compared to the normalization quantities. For the same gap size the SAS-SST model has a higher torque and head than the standard RANS model. Since almost all numerical simulations are overestimating the head losses a finer mesh may help to increase the accuracy of the simulation results. Velocity profiles are evaluated at four different locations downstream the runner. The effects of the runner gap size respectively the gap flow can be observed even at the end of the draft tube cone which is at $2D$ behind the runner. At the shroud an increased runner gap size leads to higher axial velocities, while the circumferential and radial velocities are reduced. The strongest effects can be noticed close
behind the runner at the first evaluation line L1. The influence of the runner gap on the velocity profiles is smoothed out with increasing distance of the evaluation line to the runner. Differences of the size of the stagnation region, where the vortex rope develops, can be observed between the two applied turbulence models. A straight vortex rope in the center of the draft tube develops for the k-ω-SST model. For the SAS-SST model a vortex rope in the shape of a corkscrew arises from the runner hub leading to a stagnation region with about twice the radius compared to the k-ω-SST model. The hybrid turbulence model is capable of resolving smaller turbulence scales than the standard RANS model in the entire draft tube. Influences of the gap size respectively the gap flow can also be noted in the pressure signals at the beginning of the draft tube cone. An increase of the gap width leads to higher pressure amplitudes.

Even though the SAS-SST model is capable of resolving smaller turbulence scales there are also some limitations [6]. For moderate flow instabilities the SAS model may stay in RANS mode which can lead to a wrong overall flow field which might be the case in the result using the SAS-SST model [15]. Other turbulence models like DES or SBES (Stress Blended Eddy Simulation), do not require as large flow instabilities as the SAS-SST model, and are hence possibly better suited for the simulation of operation points where a full load vortex in the draft tube appears.

Acknowledgment
The authors would like to thank the HLRS, Stuttgart for providing the computational resources in the framework of a Bundesprojekt

References
[1] Commision of the European Communities (editor) Towards sustainable water management in the European Union first stage in the implementation of the Water Framework Directive 2000 L327 1–72
[2] Federal Ministry for Economic Affairs and Energy Accessed: 2016-01-10 Erneuerbare Energien auf einen Blick URL http://www.bmwi.de/DE/Themen/Energie/Erneuerbare-Energien/erneuerbare-energien-auf-einen-blick.html
[3] International Electrical Commision (editor) International standard IEC 60193 Second Edition 1999-11: Hydraulic turbines. storage pumps and pump-turbines Model Acceptance Tests 1999
[4] Kirschner O 2011 Experimentelle Untersuchung des Wirbelzopfes im geraden Saugrohr einer Modell-Pumpturbine Ph.D. thesis University of Stuttgart
[5] Strelets M Detached Eddy Simulation of Massively Separated Flows 2001 39th Aerospace Sciences Meeting and Exhibit
[6] Menter F R Best Practice: Scale-Resolving Simulations in ANSYS CFD Version 2.0 2015
[7] Jasak H, Weller H G and Gosman A D High resolution NVD differencing scheme for arbitrarily unstructured meshes 1999 International Journal Numerical Methods in Fluid 31 432–449
[8] Menter F R and Egorov Y Two-equation eddy-viscosity turbulence models for engineering applications 1994 AIAA-Journa 32(8) 269–289
[9] Menter F R, Schütze J and Gritskевич M Global vs. Zonal Approaches in Hybrid RANS-LES Turbulence Modelling 2012 Progress in Hybrid RANS-LES Modeling Notes on Numerical Fluid Mechanics and Multidisciplinary Design 117 15–28
[10] Menter F R and Egorov Y The Scale-Adaptive Simulation Method for Unsteady Turbulent Flow Predictions. Part 1: Theory and Model Description 2010 Flow, Turbulence and Combustion 85 113–138
[11] Egorov Y, Menter F R, Lechner R and Cokljat D The scale-adaptive simulation method for unsteady turbulent flow predictions. part 2: Application to complex flows 2010 Journal Flow, Turbulence and Combustion 85(1) 139–165
[12] Egorov Y and Menter F R Development and Application of SST-SAS Turbulence Model the DESIDER Project 2008 Advances in Hybrid RANS-LES Modelling, Notes on Numerical Fluid Mechanics and Multidisciplinary Design 97 261–270
[13] Rotta J C 1972 Turbulente Strömungen (Teubner)
[14] Joeng J and Hussain F On the identification of a vortex 1995 Journal of Fluid Mechanics 285 69–94
[15] Junginger B and Riedelbauch S Investigation of the effects of runner gap width on the flow field in the draft tube 2016 16th International Symposium on Transport Phenomena and Dynamics of Rotating Machinery