Investigation of Francis Turbine Part Load Instabilities using Flow Simulations with a Hybrid RANS-LES Turbulence Model

Timo Krappel¹, Albert Ruprecht¹, Stefan Riedelbauch¹, Roland Jester-Zuerker² and Alexander Jung²

¹Institute of Fluid Mechanics and Hydraulic Machinery, University of Stuttgart, Pfaffenwaldring 10, 70550 Stuttgart, Germany
²Voith Hydro Holding GmbH & Co. KG, Alexanderstraße 11, 89522 Heidenheim, Germany
E-mail: timo.krappel@ihs.uni-stuttgart.de

Abstract. The operation of Francis turbines in part load condition causes high pressure fluctuations and dynamic loads in the turbine as well as high flow losses in the draft tube. Owing to the co-rotating velocity distribution at the runner blade trailing edge a low pressure zone arises in the hub region finally leading to a rotating vortex rope in the draft tube. A better understanding and a more accurate prediction of this phenomenon can help in the design process of a Francis turbine. The goal of this study is to reach a quantitatively better numerical prediction of the flow at part load and to evaluate the necessary numerical depth with respect to effort and benefit. As standard practice, simulation results are obtained for the steady state approach with SST turbulence modelling. Those results are contrasted with transient simulations applying a SST as well as a SAS (Scale Adaptive Simulation) turbulence model. The structure of the SAS model is such, that it is able to resolve the turbulent flow behaviour in more detail. The investigations contain a comparison of the flow losses in different turbine components.

A detailed flow evaluation is done in the cone and the diffuser of the draft tube. The different numerical approaches show a different representation of the vortex rope phenomenon indicating differences in pressure pulsations at different geometric positions in the entire turbine.

Finally, the turbulent flow structures in the draft tube are illustrated with several evaluation methods, such as turbulent eddy viscosity, velocity invariant and turbulent kinetic energy spectra.

1. Introduction
Steady state flow simulation is state of the art in the design process of hydraulic machines. The usage of RANS turbulence models gives quite good results for the design point at best efficiency. In contrast to that Francis turbines have to operate more and more in off-design operating conditions. At such operating points, in particular in part load, it is well known, that with this modelling approach the prediction accuracy of the complex draft tube flow is insufficient.

Therefore, the goal of this study is the adequate prediction of the flow field for a high specific speed Francis turbine in part load. The focal point of the current investigations is set on one part load operating point with a discharge of \( Q = 0.72 Q_{\text{opt}} \) and a head of \( H = 0.9 H_{\text{opt}} \). For this operating point different modelling approaches based on SST turbulence model are investigated.
A comparison of steady state and transient flow simulations is done. Also a more advanced turbulence model than a RANS model, the SAS (Scale Adaptive Simulation) approach, is part of this study using different mesh sizes.

2. Turbulence modelling
The SAS approach enables the unsteady SST RANS turbulence model [5] to operate in SRS (Scale Resolving Simulation) mode [7]. This is achieved by introducing an additional source term $Q_{SAS}$ in the transport equation for the turbulence eddy frequency $\omega$ of the SST model [1, 2, 6], as described in equation (1).

$$\frac{\partial \rho \omega}{\partial t} + \nabla \cdot (\rho U \omega) = \alpha \frac{\omega}{k} P_k - \rho \beta \omega^2 + Q_{SAS} + \nabla \cdot \left[ \left( \mu + \frac{\mu_t}{\sigma_\omega} \right) \nabla \omega \right] + (1 - F_1) \frac{2\rho}{\sigma_\omega \omega} \nabla k \nabla \omega \quad (1)$$

$Q_{SAS}$ is defined as:

$$Q_{SAS} = \max \left[ \rho \kappa S^2 \left( \frac{L}{L_{vK}} \right)^2 - C \frac{2\rho_k}{\sigma_{\Phi}} \max \left( \frac{\left| \nabla \omega \right|^2}{\omega^2}, \frac{\left| \nabla k \right|^2}{k^2} \right), 0 \right], \quad (2)$$

containing the turbulent length scale $L$ and the von Karman length scale $L_{vK}$. Based on the theory of Rotta [10], $L_{vK}$ describes the second derivative of the velocity field:

$$L_{vK} = \kappa \left| \frac{U'}{U} \right| \frac{U''}{U}, \quad U' = S = \sqrt{2 S_{ij} S_{ij}} \quad S_{ij} = \frac{1}{2} \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \quad (3)$$

This procedure reduces the turbulent viscosity and therefore enables the resolution of unsteady structures in the simulation. Compared to the RANS-SST model, smaller structures can be generated and the turbulence cascade goes down to grid limit. As the SAS model does not provide sufficient damping of the smallest turbulent scales at grid limit, a limiter related to the Smagorinsky LES model [1] is applied to the turbulent eddy viscosity which ensures a proper dissipation of turbulence. One advantage of the SAS model is that it has the ability to operate in RANS mode, if grid resolution and time step is not fine enough. There are other hybrid RANS-LES turbulence models, e.g. the DES-type models [11]. These models, however, need to have a sufficiently fine mesh and time step.

3. Numerical model
For the flow simulation of the Francis turbine the commercial CFD code Ansys CFX version 14.5 is used. As the vortex rope is an unsteady three-dimensional phenomenon transient analysis has been chosen for all relevant components. These are spiral case, stay and guide vanes, runner blades and draft tube with expansion tank (see Fig. 1). An expansion tank has been chosen as draft tube extension to replicate the test rig which is ready to operate in the near future. Therefore the simulation model has “test rig size”.

In contrast to other similar publications, no negligence of turbine components was done for saving computational time (see e.g. [4, 9]), as the upstream effect of the vortex rope is also investigated in this work. Gaps between the runner and non-rotating components are not modelled. At the inlet of spiral casing steady boundary conditions are applied with a mass flow specification of the part load operating point.

For temporal discretisation a second order backward Euler scheme and for spatial discretisation a bounded second order central differencing scheme [3] is used. For the turbulence quantities a bounded second order backward Euler scheme is applied for the temporal discretisation and a first order scheme for the spatial discretisation [8].
Figure 1. Hydraulic contour of Francis machine used for the simulations

Table 1. Description of grid size for the coarser mesh (16M) and the finer mesh (40M) for different turbine parts in number of elements and averaged $y^+$-values

| Turbine part                  | 16M elements | 16M $y^+$ | 40M elements | 40M $y^+$ |
|-------------------------------|--------------|-----------|--------------|-----------|
| Spiral case                   | 1019k        | 28.0      | 4643k        | 21.0      |
| Stay/guide vanes              | 3708k        | 17.5      | 8561k        | 12.4      |
| Runner                        | 3778k        | 22.2      | 9582k        | 16.3      |
| Draft tube (with exp. tank)   | 8091k        | 15.6      | 18204k       | 8.9       |
| **Total**                     | **16.2M**    | **16.2**  | **40.1M**    |           |

Two different grids of entirely hexahedral type are evaluated for the simulation of the Francis turbine: one with approximately 16 million elements (16M) and a refined one with around 40 million elements (40M) (see Table 1). Although the grid size is quite large, wall resolution is still away from resolving the boundary layer into the laminar region. The majority of the mesh points, about 50%, are concentrated in the draft tube to obtain a good resolution of the complex vortex flow. As it is essential to have a Courant-number smaller than one to resolve turbulent structures to small scales, the time steps size corresponds to $2^\circ$ of runner revolution for the coarser (16M) respectively $0.5^\circ$ for the finer mesh (40M).

4. Results

A comparison of flow field results obtained with different modelling approaches is presented here. Focus is put on global quantities, like hydraulic losses and efficiency and detailed flow evaluation in draft tube cone and the diffuser. Also, pressure pulsations at different positions in the machine are depicted. Finally, the vortex rope shape and turbulent quantities are visualised.

All transient simulations are initialised with a result from a simulation with less numerical accuracy. Afterwards, approximately 15 runner revolutions have to be simulated to reach a periodic flow behaviour. To get statistical meaningful results further simulation time is necessary. For the calculation of global machine data, ten runner revolutions were time-averaged. The velocity profile data at the draft tube cone and draft tube diffuser are time-averaged with 35 runner revolutions.

4.1. Global machine data

The steady state simulation approach with SST turbulence model overestimates the total losses for the complete turbine, especially in the runner and draft tube (see Fig. 2), by more than
4% of total head compared with the reference simulation SAS-SST 40M. By using a URANS approach, the deviation is reduced to about 1%. As the 16M mesh already has a quite good resolution for predicting the large structures of the vortex rope, the losses are quite similar compared to the reference. Generally, the largest deviations of losses can be found in the runner and draft tube domain, where transient phenomena are most dominant.

4.2. Flow analysis
A flow analysis is done on an evaluation line half way down the cone for all velocity vector components (see Fig. 3). The steady state simulations show large deviations from the transient simulations for all velocity components (see Fig. 4). For transient simulations the differences between RANS-SST and SAS-SST turbulence model are moderate for the coarser mesh (16M). However, the result for the finer mesh (40M) with SAS-SST model shows a larger axial velocity component without any back flow in the core region. The tangential component is co-rotating along the entire evaluation plane except a tiny region close to the rotation axis. This might be due to a better resolution of the vortex rope phenomenon at the runner outlet. For the radial component only minor differences are visible.

Velocity vector components are also displayed on an evaluation line in the right channel of the diffuser (see Fig. 3). In the diffuser a similar tendency is visible for the steady state simulation like in the cone (see Fig. 5). The meridional velocity component of the steady state simulation has the largest relative deviation from the other simulations. The transient RANS simulation also shows some deviations, whereas the SAS simulations with both meshes produce similar
results. For the other velocity components in transversal direction similar tendencies are visible.

4.3. Vortex rope
The vortex rope phenomenon is generated at the runner outlet due to a velocity distribution with a high transport and tangential portion at the runner band, similar to Fig. 4. This velocity distribution causes a low pressure zone downstream of the runner in the center region. The vortex rope is rotating around this low pressure zone and propagates into the draft tube. It has a helical shape and rotates with about 30% of the runner rotational frequency.

The result of the transient RANS-SST simulation shows a comparably short vortex rope with a quite clear elliptical shape (see Fig. 6). The vortex rope obtained by SAS-SST simulation with the coarser 16M mesh propagates further into the cone towards elbow. The result with the
Figure 5. Velocity components in the draft tube diffuser on a horizontal line in the right channel in streamwise direction

Figure 6. Visualisation of vortex rope shape by isosurfaces of pressure

finer mesh (40M) has almost the same extension. With larger mesh resolution the elliptic shape of the vortex rope is less pronounced. The result shows more details of turbulent structures, both on half height where the vortex rope is rotating around its own axis and at the end in the elbow where it decays to turbulence. These turbulent structures are well known from LES simulations which indicates a LES-like resolution of the vortex rope for the fine resolution mesh (see Fig. 6(c)).
Figure 7. Pressure amplitude at different locations in the Francis turbine; the positions for (b) can be found in Fig. 3; the results for Fig. 7(a) and for Fig. 7(b) position 4 have the same location.

Table 2. Mass flow fluctuation at different positions in % of total mass flow

| Position                  | RANS-SST 16M | SAS-SST 16M | SAS-SST 40M |
|---------------------------|--------------|-------------|-------------|
| Stay vane inlet           | 0.0183       | 0.0181      | 0.0159      |
| Vaneless space            | 0.0509       | 0.0470      | 0.0404      |
| Runner outlet              | 0.1058       | 0.1010      | 0.0942      |

Pressure fluctuation amplitude in the draft tube cone is highest for transient RANS simulation and decreases with increasing modelling effort (see Fig. 7(a)). Also the frequency is somewhat shifted towards higher frequencies for the SAS-SST 40M simulation. As the vortex rope passes the cone wall with a clear and compact shape, a local high pressure peak is generated. In contrast, when the elliptic shape is less pronounced due to the resolution of more details of turbulence, this peak is reduced. A reason for the frequency shift of the SAS-SST 40M simulation could be the higher tangential velocity (see Fig. 4(a)).

Pressure fluctuation amplitudes are also evaluated at other positions in the machine (see Fig. 7(b)) as well as upstream and downstream of the runner. In the whole draft tube the trend of increasing pressure amplitudes with decreasing numerical modelling approaches persists. Further upstream the same trend as in the draft tube is visible for the spiral case (position 1) and the stay vanes (position 2).

In the vaneless space between guide vanes and runner another trend is visible: The results obtained with a more advanced modelling approach show higher pressure amplitudes.

The rotating vortex rope is leading to some kind of blockage effect in the runner, stay vanes and guide vanes. The blockage of a channel of the correspondent turbine part is \( \sim 0.1\% \) of total mass flow at the runner outlet, \( \sim 0.05\% \) in the vaneless space and \( \sim 0.018\% \) at the inlet of the stay vanes (see Table 2). For the more exact modelling approach not only the pressure pulsations in the draft tube are lower, but also the fluctuation of mass flow.

4.4. Turbulence evaluation

The turbulent eddy viscosity is a measure of the amount of LES-content being used by the SAS turbulence model. For RANS-SST turbulence model the eddy viscosity in the draft tube is about
Table 3. Turbulent eddy viscosity ratio (turbulent eddy viscosity/dynamic viscosity) in the draft tube

| Simulation    | Average | Maximum |
|---------------|---------|---------|
| RANS-SST 16M | 915.05  | 3011.63 |
| SAS-SST 16M  | 67.10   | 422.57  |
| SAS-SST 40M  | 52.30   | 401.27  |

Figure 8. Cutting planes in the draft tube, coloured by viscosity ratio (turbulent eddy viscosity/dynamic viscosity) 0-1500 for (a) RANS-SST and 0-150 for (b) and (c) SAS-SST

One order higher than for the SAS-SST model (see Table 3 and Fig. 8). Higher viscosity values lead to a higher damping of flow structures. The difference of maximum and average turbulent eddy viscosity values between both mesh resolutions is moderate for the SAS-SST model. This is an indication that the 16M mesh is already quite appropriate to resolve the turbulent flow structures. On the other hand, if the grid filter of the SAS model is reached, a reduction of turbulent eddy viscosity would require a considerably finer mesh.

The result of the RANS-SST model shows relatively large flow structures with high eddy viscosity values compared with the results using SAS-SST model (see Fig. 8 and Fig. 9). Therefore, the SAS model is able to resolve smaller turbulent structures. There is still some mesh dependency using the SAS model, which can be seen in the size of the flow structures visualised by the velocity invariant $Q$ ($Q = 0.5(\Omega^2 - S^2)$) (see Fig. 9). Especially in the draft tube cone and elbow, where larger structures of the vortex rope decay to smaller turbulent structures. This decay is indicated by the higher values of turbulent eddy viscosity. In the diffuser the difference...
Figure 9. Isosurfaces of velocity invariant $Q = 1$ in the draft tube, coloured by viscosity ratio 0-150

Figure 10. Turbulent kinetic energy spectra of each one point in draft tube cone and diffuser is not as obvious as upstream (at least for this $Q$-isosurface).

To quantify the amount of resolved turbulence in the flow simulation the turbulent kinetic energy spectra are compared (see Fig. 10). As expected the RANS simulation does not represent the inertial range of turbulence as the turbulent content of the flow is modelled. In contrast to that, the simulations with the SAS model show a well represented inertial range of turbulence. The difference between the two meshes is not as pronounced as was expected.
5. Conclusion
A study of different numerical approaches for the flow simulation of a Francis turbine part load operating point has been performed. The steady state simulation type is not able to predict the draft tube flow correctly. It predicts unrealistic high loss values, especially in draft tube and runner resulting a diminished efficiency prediction. Also the flow field in the draft tube has large deviations compared to more accurate numerical approaches. Hence, transient flow simulation is necessary to predict the unsteady flow phenomena in the draft tube, like the vortex rope. Also for transient flow simulations there are differences in losses and efficiency between RANS and SAS turbulence model using 16 million and 40 million element meshes. For those simulation approaches differences in the velocity components in draft tube cone and diffuser are visible.

Using a more exact numerical approach the vortex rope shape is predicted in more detail. As expected from a hybrid RANS-LES turbulence model more detailed turbulent structures are visible, especially for the fine mesh. The different velocity profiles in the draft tube cone and the different vortex rope shapes obtained by the simulation with SAS turbulence model with fine mesh lead to a reduction of simulated pressure pulsations.

Using the SAS turbulence model with fine mesh reduces the turbulent eddy viscosity. Compared to RANS turbulence models, the SAS approach is able to predict turbulent flow structures in more details, especially in the elbow where the vortex rope decays to small turbulent structures and in the diffuser.

Further work should address a substantial increase of the overall number of mesh elements to assure that the grid filter of the SAS turbulence model does not significantly influence the simulation result.

References
[1] Egorov Y and Menter F R 2008 Development and Application of SST-SAS Turbulence Model in the DESIDER Project Advances in Hybrid RANS-LES Modelling, Notes on Numerical Fluid Mechanics and Multidisciplinary Design Volume 97 pp 261-270
[2] Egorov Y, Menter FR and Cokljat D 2010 The Scale-Adaptive Simulation Method for Unsteady Turbulent Flow Predictions. Part 2: Application to Aerodynamic Flows Journal Flow Turbulence and Combustion 85(1) pp 139-165
[3] Jasak H, Weller H G and Gosman AD 1999 High resolution NVD differencing scheme for arbitrarily unstructured meshes Int. J. Numer. Meth. Fluids 31 431-449
[4] Jošt D, Škerlavaj A and LIpej A 2012 Numerical flow simulation and efficiency prediction for axial turbines by advanced turbulence models 26th IAHR Symposium on Hydraulic Machinery and Systems Beijing China
[5] Menter F R 1994 Two-equation eddy-viscosity turbulence models for engineering applications AIAA-Journal 32(8) pp 269-289
[6] Menter F R and Egorov Y 2010 The Scale-Adaptive Simulation Method for Unsteady Turbulent Flow Predictions. Part 1: Theory and Model Description Journal Flow Turbulence and Combustion 85(1) pp 113-138
[7] Menter F R, Schütze J and Gritskevich M 2012 Global vs. Zonal Approaches in Hybrid RANS-LES Turbulence Modelling Progress in Hybrid RANS-LES Modelling Notes on Numerical Fluid Mechanics and Multidisciplinary Design Volume 117 pp 15-28
[8] Menter F R 2012 Best Practice: Scale-Resolving Simulations in ANSYS CFD Version 1.0 ANSYS Germany GmbH April 2012
[9] Neto A D, Jester-Zuerker R, Jung A and Maiwald M 2012 Evaluation of a Francis turbine draft tube flow at part load using hybrid RANS-LES turbulence modelling 26th IAHR Symposium on Hydraulic Machinery and Systems Beijing China
[10] Rotta J C 1972 Turbulente Strömungen BG Teubner Stuttgart
[11] Shur M L, Spalart P R, Strelets M K and Travin A K 2008 A hybrid RANS-LES approach with delayed-DES and wall-modelled LES capabilities International Journal of Heat and Fluid Flow 29 1638-1649