Progress in load prediction for speed-no-load operation in Francis turbines

C Mende, W Weber and U Seidel
Voith Hydro Holding GmbH & Co. KG, Alexanderstraße 11, 89522 Heidenheim, Germany
E-Mail: carsten.mende@voith.com

Abstract. Francis turbines are increasingly required to operate from 0-100% of power output. For the design of these turbines, a sound understanding of formerly “off-design” operating points, e.g. speed-no-load, is necessary. One way to assess loads at these operating points is to apply scale resolving CFD methods in order to account for the broad-band turbulence spectrum. This results in stochastic load patterns, which are characteristic for these off-design points. In this paper, two CFD approaches for scale resolving simulations are applied to a Francis model turbine to resolve the larger anisotropic scales in turbine flow at speed-no-load operation. The first uses a hybrid RANS-LES model presented by Menter & Egorov [2]. Here the resolved scales are adapted continuously by RANS-LES blending based on the actual flow condition. The second approach is a LES with a dynamic model based on Germano et al. [3]. For both approaches, spectra of resulting pressure fluctuations at the draft tube cone wall are presented. Additionally, pressure loads from LES are applied to calculate mean and dynamic stresses. The dynamic stresses are finally compared with measurements in the corresponding prototype turbine.

1. Introduction

The main challenge of emerging renewable energy sources is their intermittent supply. Power infeed, especially from wind and solar radiation, is actually not reliably foreseeable and doesn’t account for the actual demand. Therefore, energy storage capacity is crucial for economic power generation and grid stability. At present, the build-up of capacity is not keeping pace with the growing amount of volatile energy from new renewables. To meet the requirements for electrical grid stabilization, hydroelectric installations are the most efficient way to supply the lacking regulatory power with sustainable energy. Consequently, the operation of Francis turbines changes from traditional base load operation towards stabilization modes, characterized by frequent start-stop cycles and significant operation at low power output or speed-no-load (SNL). The impact of this changed operating strategy on fatigue can be tremendous for availability, reliability and safety. Recent progress with CFD on the simulation of SNL operation of a Francis runner was shown by Seidel et al. [1]. The paper on hand aims to describe the investigated setup and the simulation procedure briefly. Also, more results on the unsteady flow field shall be given. Additionally, the impact of the numerical turbulence handling on resolved pressure fluctuations is shown by direct comparison of pressures for a scale-adaptive simulation (SAS) proposed by Menter and Egorov [2] and a large eddy simulation (LES) applying a dynamic subgrid-scale model based on Germano et al. [3]. Finally, measured stresses in the prototype runner are compared to results from a Finite Element method (FEM) based on unsteady pressure fields simulated by LES.
2. Simulation approach
The objective of this study was to determine dynamic stresses at the blades of a medium high head Francis turbine during SNL. Therefore, a simulation of the unsteady flow field was performed first. In order to resolve the stochastic pressure fluctuations at SNL, which are observed in measurements, different kinds of unsteady simulation approaches were investigated. The presented results refer to a LES with a dynamic model and a hybrid scale-adaptive simulation with a SST turbulence model (SAS-SST). The flow simulation provided the pressure loads on the blades. These loads were applied in a FE model of the prototype turbine, yielding stresses at the measurement positions for every investigated time step.

Furthermore, during flow simulation monitor points for pressure were defined at the draft tube cone wall. Their position is in accordance with pressure sensors in experimental model tests and similar to prototype test positions. Thus, beside stresses, direct comparison of pressure fluctuations between simulation and experiment is presented.

The subsequent sections give more detailed information on general and individual settings of the simulations.

2.1. Unsteady flow simulation
For CFD, water was considered incompressible with only one fluid phase, thus neglecting cavitation. Simulations were performed at model scale in order to better resolve boundary layers. Instead of a sector model, full distributor and runner are modelled to avoid periodic boundary conditions. This allows for non-axisymmetric flow patterns from turbulent structures namely large coherent ones. An overview of numerical schemes is given in table 1. Discretization of advection and time were second order or high resolution, depending on the individual solver. Turbulence was modelled with first order accuracy in case of the SAS-SST simulation.

Flow simulations, the choice of turbulence models can be essential depending on the problem under investigation. Two turbulence models were applied (detailed description is - for the sake of brevity - omitted here and can be found in [2-7]). The dynamic LES model proposed by [3] extends the classical model of Smagorinsky by introduction of a time and space dependent constant to replace the Smagorinsky constant $C_s$ in expression (2.1) for the eddy viscosity $\nu_t$ ($\Delta$ is the filter width related to the grid spacing and $|\vec{S}|$ the magnitude of large-scale strain rate tensor $\vec{S}_{ij}$).

$$\nu_t = (C_s \Delta)^2 |\vec{S}|$$ \hspace{1cm} (2.1)

$C(x,y,z,t)$ is computed by the help of the so called Germano identity, see [4], which relates the resolved turbulent stresses $\mathcal{L}_{ij}$ to modelled subgrid-scale stresses $\tau_{ij}$ and $T_{ij}$ obtained from a filter at grid spacing level $\Delta$ (results in $\tau_{ij}$) and a test filter $\tilde{\Delta}$ at a coarser filter level (determines $T_{ij}$). Note, that $\tilde{\tau}_{ij}$ is obtained by filtering $\tau_{ij}$ with $\tilde{\Delta}$.

$$\mathcal{L}_{ij} = T_{ij} - \tilde{\tau}_{ij}$$ \hspace{1cm} (2.2)

The SAS-SST hybrid turbulence model consists of the SST model used for RANS mode and the SAS model for LES mode. The SAS model is integrated in the SST model as an additional production term on the right hand side in the $\omega$-equation (cf. [2,7]). The ratio of turbulent length scale $L$ proposed by Rotta [10] and von Karman length $L_{vk}$ (see eq. (2.3), therein $\kappa$ denotes the von Karman constant) is used as non-dimensional indicator for inhomogeneous shear flows. It triggers the switching from SST model to SAS in unsteady situations.

$$L_{vk} = \kappa \frac{|U'|}{|U''|}$$ \hspace{1cm} (2.3)

Since $L_{vk}$ depends on the local flow unsteadiness, regions with necessity for scale resolution are detected automatically. The dependency of $L_{vk}$ on local unsteadiness is due to the second velocity derivative.
2.1.1. Application of SAS-SST turbulence model. The geometry used along with the SAS-SST turbulence model in ANSYS CFX® consisted of stay vanes, guide vanes, runner, and draft tube as shown in figure 1 (left). The inlet is given by the distributor entry, where mass flow and direction is set (cf. table 1). Outlet pressure and direction is prescribed at the end of the draft tube.

![Figure 1](image)

Figure 1. Clipped view of flow domain of investigated Francis turbine with SAS-SST model (left) and with dynamic subgrid-scale model (right)

2.1.2. Application of dynamic subgrid-scale model. For LES the dynamic subgrid-scale model of [3] was applied in another commercial CFD solver (AcuSolve®) based on finite elements. To achieve high spatial resolution at reasonable computational cost, the flow domain had to be reduced. As LES results from Guo et al. [9] for a Francis turbine at part load operation indicated, velocity distributions below the runner are in better agreement to experimental data, when both runner and draft tube are modelled. To meet this requirement at affordable computational costs, the spatial resolution in the draft tube had to be decreased for LES also. Consequently, the runner, a small stationary ring at the inlet and the draft tube (see figure 1, right) are modelled. At the entry to the annular inlet domain, a time averaged velocity profile from a previous URANS simulation is prescribed (cf. table 1). Thus, unidirectional interaction of wicket gate wakes with rotating blades is ensured without need to resolve the distributor itself.

The entire runner domain was meshed uniformly to resolve large coherent structures. Smaller scales of turbulence with more homogeneous and isotropic distribution are left to be captured by the subgrid-scale model. The boundary layer in the runner is resolved (in general $y^+ < 5$). Non-conformal grid interfaces between inlet ring, runner, and draft tube are used. The interface position between runner and draft tube is shifted downstream (in respect to typical positions) to maintain the spatial resolution of the runner also some distance below. The adjacent draft tube mesh featured lower grid density to limit computational effort.

2.2. Finite element material stress simulation

Since low frequency components which are much lower than natural frequencies are dominating the fluctuating pressure, a series of quasi-static stress calculations using FE-method was performed to represent the dynamic stresses. Time steps every 1.92° were selected and the simulated pressure field

| Table 1. Discretization & boundary settings of simulations with SAS-SST and dynamic model |
|---------------------------------|-----------------|-----------------|
|                                 | SAS-SST model   | LES with dyn. subgrid-scale model |
| Number of cells                 | $13 \times 10^6$ | $105 \times 10^6$ |
| Discretization of advection     | High resolution | Second order     |
| Discretization of time          | Second order    | Second order     |
| Discretization of turbulence    | First order     | -                |
| Time step size                  | $0.2^\circ$     | $0.048^\circ$    |
| Inlet boundary                  | Mass flow & direction | Time averaged velocity field |
| Outlet boundary                 | Pressure & direction | Pressure |
on all wetted components extracted. Coordinates of pressure values and pressure values itself were scaled to fit prototype scale. For stress calculation the pressure was applied to a sector model of the runner containing one blade. Periodic boundary conditions were prescribed at the cutting planes of the sector. At the connection to the shaft common boundary conditions are prescribed. For the pressure in the runner side chambers that is not derived from CFD, common distribution between inlet and outlet considering seal location is prescribed. The mesh of the finite element model was refined towards crown and band at trailing edge in order to obtain converged stress results. For the material, linear elastic behaviour is assumed.

The simulations were performed using commercial software ANSYS® mechanical. From the resulting stress history maximum and minimum stress value at each node were determined. Therewith, the dynamic stress range as well as the static mean stress is estimated.

3. Results

3.1. Global flow field and local fluctuations

Since the global flow field doesn’t differ significantly between SAS-SST simulation and LES, it is convenient to present in this section only LES results. For Francis turbines, flow fields at low load and speed-no-load are marked by a large, stable recirculation zone in the draft tube, which has its origin within the runner. The recirculation results from the radial pressure increase in the runner due to centrifugal forces [8]. This leads to a characteristic pressure distribution at suction side of runner blades shown in [1] together with surface streamlines, indicating upstream flow near crown at the blade’s suction side. Further evidence of the recirculation zone is given by figure 2, showing vectors of velocity for a distinct time step. The vectors are coloured by the time averaged velocity component w (in direction of the runner axis). Please note that in the figure, w is related to the area averaged velocity magnitude at the runner inlet \( v_{\text{mag,in}} \). From the figure the recirculation zone is distinguishable. It also becomes obvious, that the recirculation’s downstream extent is influenced by the elbow. At the inner elbow wall, the flow is deviated later towards the runner compared to the outer elbow wall. This is indicated by the black dashed ellipses.

![Figure 2. Velocity vectors on a cut-plane through turbine axis coloured by time averaged w-velocity.](image)

Furthermore, the vectors indicate that main turbine discharge is accomplished near runner band and draft tube walls. In contrast, the central draft tube part below the runner and the hub region are dominated by upwards coming flow. Because of the reversed flow direction and also due to
continuous reduction of swirl (indicated by streamlines in figure 3), a cylindrical very turbulent shear layer is created between downwards and upwards flows.

Especially the downwards going flow exhibits high fluctuations of velocity and pressure. Figure 4 shows fluctuations of pressure for a cut-plane containing the turbine axis (left). On the right-hand side pressure fluctuations on the runner blade’s suction side are displayed. Fluctuations are determined from actual pressure values diminished by running averages according to: \[ p_{fluc} = p_{act} - p_{avg}. \]

Pressures amplitudes on the blade’s suction side amount to approx. 1% of area-averaged inlet pressure.

For the instantaneous pressure field on blade suction side, the stable pressure distribution originating from centrifugal force (cf. figure 5 in [1], left) is still observable as figure 5 shows. The fluctuations displayed in figure 4 alter the symmetrical shape of the stagnation regions (coloured in red) in figure 5 only slightly. In contrast, the flow near band is governed by more stochastic pressure oscillations.

Figure 3. Instantaneous streamlines coloured by w-velocity: downstream (blue) and upstream (red).

Figure 4. Field of pressure fluctuations: cut-plane through rotation axis (left) and on blade suction side viewed from below runner (right).
3.2. Spectral content of pressure fluctuations

Since monitor point positions are identical to pressure transducer locations in model test, comparison of characteristic pressure amplitudes is possible. As already given in [1], good agreement was found for LES. Similar good agreement was found for SAS-SST model. Beside model test data, also prototype pressure fluctuations are available - although more downstream the draft tube wall (0.97*D₂ᵃ instead of 0.71*D₂ᵃ below distributor centerline). The characteristic amplitude of 1.9% is in the same range as simulated values. Yet, it suggests that for an identical position measured values at model scale are smaller than in prototype. Thus, further improvement of the simulation accuracy is necessary.

SNL is governed by pressure pulsations, covering a broad band of low frequencies. For prediction of fatigue at this operating condition, the spectral distribution of pressure pulsations must be captured. The spectrum on the left of figure 6 is determined from short signals (1.29 runner revolutions) and shows the ability of SAS-SST and dynamic subgrid-scale model to predict the typical spectral distribution. In comparison to LES, amplitudes with SAS-SST tend to be predicted higher at very low frequencies and lower at high frequencies. Yet, it cannot be concluded, that LES gives a more broad-banded spectrum, since grid densities differed for both simulations.

![Figure 5](image)

**Figure 5.** Instantaneous pressure field: cut-plane through rotation axis (left) and on blade suction side viewed from below runner (right).

![Figure 6](image)

**Figure 6.** Spectra of relative pressure fluctuations at draft tube wall: short signals for turbine model (left) and comparison for long signals (right) of simulated model and measured prototype.

Observed amplitudes in the left spectrum of figure 6 are very high - a direct consequence of the short signals used for analysis, leading to low spectral density. Applying a longer signal (11 revolutions) therefore reduces maximum amplitudes significantly (see figure 6, left). Because of the computational cost of LES, prolonging simulation time is avoided. Consequently, the comparison with the measured prototype spectrum is performed for SAS-SST simulation results. Figure 6 shows the related spectra on the right-hand side. As expected, good agreement is found for the shape of the
spectrum, with high amplitudes around 2 times rotating frequency \( (f_n) \) for measurement and simulation. Remembering the sensor position at prototype, measured amplitudes are expected to be lower than simulated ones. Also, missing damping due to negligence of cavitation fosters over-prediction of amplitudes. Remarkable is the under-prediction of pressure fluctuations for higher frequencies. It suggests that LES with higher amplitudes for a broader frequency band would be more suited to represent measured prototype data – in case computational costs would not be of major concern. In summary, SAS-SST model provides a sufficient mean to capture the general shape of the pressure pulsation spectrum.

3.3. **Comparison of simulated stresses with prototype measurements**

Finally, mechanical stress analysis is performed for CFD results of LES simulation. Therewith, an integral value of the pressure field of CFD calculation is obtained that is compared with scaled strain data from prototype measurements. The strain gauge locations at pressure side of the blade for the comparison are sketched in figure 7.

![Figure 7. Sketch of strain gauge positions on pressure and suction side.](image)

Since the pressure distribution is obtained for 15 blades from the CFD simulation the mechanical simulation procedure that is described in section 2.2 is successively applied for the pressure fields of all blades. Therewith, different characters of dynamic pressure fields at the different blades are considered in the evaluation.

In figure 8 the deviations of the dynamic stresses of the calculated stresses to the measured ones are illustrated. In detail, the maximum deviations obtained from the 15 blades are shown. A positive value corresponds to a higher value in the measurements than in the simulation. If the deviation is negative simulated stresses are higher than measured ones.

![Figure 8. Deviation of calculated cyclic peak-peak-stresses to measured cyclic peak-peak-stresses.](image)
At the crown the calculated dynamic stresses are higher than the measured stresses. The calculated dynamic peak-to-peak stresses at the crown are in good agreement with the measurements. Here the deviation is only 5%.

At the band the dynamic peak-to-peak stresses are obtained approximately 17% too low. Here, the evaluated stress history might be too short that the whole stress range is captured by the analysis.

4. Conclusion
Due to emergence of new renewables, Francis turbines are growingly employed to supply regulatory power for grid stabilization. This operating mode necessitates frequent start-stop cycles and significant operation at deep-part-load and no-load conditions. For these conditions, dynamic loading is stochastic in nature with comparatively large amplitudes. This in turn imposes high demands on fatigue resistance of the turbines and motivates continuous design improvement of Francis turbines operated in the whole power output range. To capture the stochastic nature of dynamic loads at deep-part-load and SNL, it is necessary to enhance understanding and simulation capability of these operating conditions. Within this work, scale resolving simulations using SAS-SST model and dynamic LES were conducted for SNL of a medium high-head Francis turbine. The resulting flow field, featuring a large recirculation zone extending from runner into draft tube elbow, was shown. Also, the region near runner band and draft tube wall was found to have remarkable pressure fluctuations. Frequency analysis of simulation data revealed broad-banded spectra typical in shape for SNL operation. A comparison with measured prototype pressures suggest that LES is superior to the SAS-SST model in predicting absolute values of frequency dependent amplitudes – assumed that simulated time is long enough for comparable spectral densities of the analyzed signals.

Pressure fields from LES were applied to calculate stresses in the runner blades. Dynamic stresses were in good agreement compared to strain gauge data from prototype. In summary, the presented concept of scale resolving flow simulation followed by quasi-static stress analyses proved suitable for determination of stress amplitudes. It also evidenced scale resolving simulation a promising mean to raise knowledge on SNL operation in Francis and other hydraulic reaction turbines.

References
[1] Seidel U, Mende C, Hübner B, Weber W and Otto A 2014 Dynamic loads in francis runners and their impact on fatigue life IOP Conference Series: Earth and Environmental Science 22 3
[2] Menter F R and Egorov Y 2010 The scale-adaptive simulation method for unsteady flow predictions. part 1: theory and model description J. Flow Turbulence and Combustion 85 1 113-138
[3] Germano M, Piomelli U, Moin P and Cabot W H 1991 A dynamic subgrid-scale eddy viscosity model Physics of Fluids A 3 7 1760–65
[4] Sagaut P 2006 Large eddy simulation for incompressible flows (Berlin: Springer)
[5] Ferziger J H and Perić M 2002 Computational methods for fluid dynamics (Berlin: Springer)
[6] Pope S B 2000 Turbulent flows (Cambridge: Cambridge University Press)
[7] Davidson L 2006 Evaluation of the SST-SAS model: channel flow, asymmetric diffuser and axi-symmetric hill Proc. of the European Conference on Computational Fluid Dynamics ed Wesseling P, Oñate E et al (Egmond aan Zee: Delft University of Technology)
[8] Pfleiderer C and Petermann H 1964 Strömungsmaschinen (Berlin: Springer)
[9] Guo Y, Kato C and Miyagawa K 2007 Large-eddy simulation of non-cavitating and cavitating flows in the draft tube of a francis turbine Seisan Kenkyu 59 1 83-88
[10] Rotta J C 1972 Turbulente Strömungen (Stuttgart: BG Teubner)