Francis-99: Transient CFD simulation of load changes and turbine shutdown in a model sized high-head Francis turbine

Peter Mössinger, Roland Jester-Zürker, Alexander Jung
Voith Hydro Holding GmbH & Co KG, St Pölten Straße 43, 89522 Heidenheim, Germany
E-mail: peter.moessinger@voith.com

Abstract. With increasing requirements for hydropower plant operation due to intermittent renewable energy sources like wind and solar, numerical simulations of transient operations in hydraulic turbo machines become more important. As a continuation of the work performed for the first workshop which covered three steady operating conditions, in the present paper load changes and a shutdown procedure are investigated. The findings of previous studies are used to create a 360° model and compare measurements with simulation results for the operating points part load, high load and best efficiency. A mesh motion procedure is introduced, allowing to represent moving guide vanes for load changes from best efficiency to part load and high load. Additionally an automated re-mesh procedure is added for turbine shutdown to ensure reliable mesh quality during guide vane closing. All three transient operations are compared to PIV velocity measurements in the draft tube and pressure signals in the vaneless space. Simulation results of axial velocity distributions for all three steady operation points, during both load changes and for the shutdown correlated well with the measurement. An offset at vaneless space pressure is found to be a result of guide vane corrections for the simulation to ensure similar velocity fields. Short-time Fourier transformation indicating increasing amplitudes and frequencies at speed-no load conditions. Further studies will discuss the already measured start-up procedure and investigate the necessity to consider the hydraulic system dynamics upstream of the turbine by means of a 1D3D coupling between the 3D flow field and a 1D system model.

1. Introduction
An increasing demand for clean peak power energy leads to new requirements for hydraulic turbines such as a rising number of load changes or start-stop procedures. So far, numerical simulations are mostly based on the investigation of steady operating points. In recent years the focus shifted towards extreme operation ranges like deep part load and overload as well as load variations, start-stop conditions or total load rejection with the goal to better understand occurring local flow phenomena. This leads to new challenges for computational fluid dynamics (CFD) methods like the representation of moving guide vanes or to take account of hydrodynamics up- and downstream of the turbine. First studies were done by Nicolle et al. [1, 2], investigating a startup procedure or Li et al. [3] and Mössinger et al. [4] simulating a load rejection for a prototype Francis turbine. In case of a load rejection either the domain upstream the guide vanes has to be fully modeled or with the aid of a 1D Code considered in a simplified way. In the first of three upcoming workshops different numerical solver, mesh densities and simulation approaches were used to investigate three different operating points [5]. In the current, second workshop load changes and shutdown/startup procedures are investigated.
2. Experimental setup

2.1. Model machine

The test rig used for this measurement campaign is located at the Water Power Laboratory, Norwegian University of Science and Technology, Trondheim, Norway. The turbine, operating with an open loop circuit, is scaled down from prototype 1:5.1 [6]. Besides the runner with a total of 15 splitters and 15 full length blades the model includes 14 stay vanes inside the volute casing, 28 guide vanes and an elbow draft tube. Figure 1 shows the top view (left) and the meridional section (right) of the model machine. Line 1 and 2 are indicating the particle image velocimetry (PIV) measurement lines and DT5 the piezoelectric dynamic pressure sensor.

![Figure 1: model machine, location of the pressure sensors and PIV measurement sections [7]](image)

2.2. Measurement

Similar to the first workshop three different operating points are investigated. In contrast to previous investigations the part load (PL) and high load (HL) operation point are modified such that they have now a constant speed factor $n_{ed}$ (cf. figure 2). Based on these conditions, load changes are initiated going from best efficiency point (BEP) to part load and high load. Axial and radial velocity measurements in the draft tube as well as pressure measurements in the spiral case, vaneless space and draft tube are performed. Table 1 summarizes the conditions for all three operating points.

![Figure 2: hill chart of the high head Francis turbine model [5]](image)
Table 1: operating points and experimental data for pressure and velocity measurement

| Parameter             | symbol | unit   | PL    | BEP   | HL   |
|-----------------------|--------|--------|-------|-------|------|
| Net head H m          |        |        | 11.87 | 11.94 | 11.88|
| Flow rate Q m³/s      |        |        | 0.14  | 0.20  | 0.24 |
| Runner angular speed n| 1/min  |        | 332   | 332   | 332  |
| Guide vane angle α    |        | °      | 6.72  | 9.84  | 12.43|
| Efficiency ηh %       |        | %      | 90.13 | 92.39 | 91.71|
| Hydraulic torque T Nm |        |        | 420.8 | 620.6 | 744.4|
| Speed factor n ed     |        |        | 0.179 | 0.178 | 0.179|
| Discharge factor Q ed  |        |        | 0.110 | 0.152 | 0.184|

3. Numerical setting

3.1. Solution setup

All simulations are performed with the software ANSYS CFX v16.0. CFX is an established and well validated fluid solver and allows very high accuracy in the prediction of complex turbulent flows [8]. It uses a finite volume based discretization scheme up to second order accuracy for convective fluxes and truly second order accuracy for diffusive fluxes. Time step for all simulations is Δt = 2 · 10⁻³ s which is equivalent to a runner rotation of ΔRU = 4° per time step. Convergence is achieved with root mean square residuals of pressure and mass-momentum of r ≤ 2 · 10⁻⁵ with minimum seven iterations per time-step. Turbulent scales were modeled by the SST k-ε-model with automatic wall function.

3.2. Mesh and boundary conditions

Based on the outcome of the first workshop an appropriated mesh density is used [9]. All simulations were done with a 360° domain, including the volute casing (SC), all 14 stay vanes (SV), 28 guide vanes (GV), 30 runner blades (RU) and the elbow draft tube (DT). Table 2 summarizes the resulting mesh and quality parameters. Non matching grid interfaces are modeled with the general grid interface (GGI) where the transition from stationary to rotating domain and vice versa is handled with a transient rotor stator interface, accounting for all transient flow characteristics. Meshing is done with support of an in-house tool, using scaled geometry and block-structured hexahedral cells, ensuring constant mesh conditions when varying the guide vane angle and furthermore allowing a rapid change of mesh properties or interface positions. The stay vanes in the present geometry extend far in the spiral case which prevents a separate modeling as individual domains. Therefore the spiral case and the stay vanes are

Table 2: used grid densities and resulting quality parameters

|                  | SC+SV | GV   | RU   | DT   |
|------------------|-------|------|------|------|
| Nodes (×10⁶)     | 4,13  | 14,46| 9,57 | 1,63 |
| Total nodes (×10⁶)| 29,8  |      |      |      |
| Min. element angle| 13.3°| 15.0°| 15.1°| 57.3°|
| Max. aspect ratio | 280  | 148  | 255  | 55   |
| Max. expansion factor | 39   | 84   | 49   | 3    |
| Mean y⁺ (BEP)    | 50    | 23   | 60   | 49   |
meshed manually as a single domain. Since the discharge and inlet pressure measurements show some time delay, a linear discharge variation based on the end points of the transitions is imposed at the inlet. The domain outlet is defined with a zero static pressure condition allowing backflow by switching to total pressure based on the normal component of velocity.

### 3.3. Guide vane rotation

Moving guide vanes during load changes and shutdown are realized with a combination of node motion and automated remeshing. ANSYS CFX already comes with the ability to deform meshes [10]. While the boundary movement is specified, all remaining nodes shift according to a displacement diffusion by solving the equation:

\[
\nabla (\Gamma_{\text{disp}} \nabla \delta) = 0
\]

Mesh stiffness \( \Gamma_{\text{disp}} \) increases as the control volume size decreases:

\[
\Gamma_{\text{disp}} = \left( \frac{V_{\text{ref}}}{V} \right)^{C_{\text{stiff}}}
\]

For this case, larger control volumes absorb more mesh motion. An increase of the stiffness model exponent \( C_{\text{stiff}} \) from 2 (default) to 15 yields to a much more abrupt stiffness variation, ensuring a desirable mesh quality along the guide vane movement. As the guide vane movement is not a linear function, measurement values were taken for the simulation. As shown in figure 3 a moving mesh approach is sufficient for both load changes where the minimum mesh angles are kept in an acceptable range.

![Figure 3: guide vane angle and minimum cell angle for load changes from best efficiency to part load (left) and high load (right)](image)

However for a shutdown process the mesh quality successively decreases as the guide vane angle goes to zero and thus a mesh adaption is required. Therefore an automated remesh routine is used [4]. Depending on some logical expression such as maximum runtime, minimum cell angle, maximum aspect ratio or any combination, the simulation is paused and with the aid of an in-house mesher, depending on the current guide vane angle, a new mesh is generated. After loading the mesh into CFX and interpolating the old results, the simulation proceeds. This procedure is valid up to small openings \( (\alpha < 2^\circ) \) where manually generated meshes are necessary. Figure 4 shows the guide vane movement, corresponding minimum cell angles and maximum aspect ratios. At a run-time of 5.6 s and 7.4 s new meshes were generated.
4. Results

4.1. Operating points

In a first step three operating points that serve as start and end points of the load changes are investigated. Figure 5 shows the comparison between measurement, averaged over 5 s and simulation results which are averaged over 3 s. As in the first workshop the guide vane angle is corrected to ensure the similarity between the velocity fields up- and downstream of the runner for the experiment and the simulation. However in contrast to previous investigations the total guide vane angle correction set at best efficiency is kept constant for all operating points to ensure identical GV rotation. All investigated values are within an acceptable range of less than 4 % deviation, except the pressure measurement at the vaneless space. As indicated in the small sketch the pressure position is at the suction side near the trailing edge. A positive correction of the guide vanes will decrease the pressure and therefore lead to an underestimation compared to the measurement.

Figure 5: comparison between numerical and experimental values for all three operating points

Axial velocities of the simulation results at the top evaluation plane in the draft tube are in good agreement to the measurement for all three operating points (cf. figure 6), where positive values are pointing in flow direction. At best efficiency, velocities in the draft tube center are slightly underestimated where at part load the simulation shows some backflow which is also an underestimation compared to the measurement. The averaged velocity profile for part load condition after the load change from best efficiency was added to the left diagram of figure 6. Slight deviations between the measured steady operating point and the velocity profile after the load change are visible. This was only recognizable for part load conditions.
4.2. Load changes

4.2.1. Best efficiency to part load  The first load change starts from best efficiency and moves towards part load. There are only minor differences between the top and bottom measurement plane, therefore a representation is focused on the top plane. In figure 8a a comparison of the axial velocity at four different time instances during the load change is shown, where in figure 8b four different positions in radial direction are displayed. The reduced flow velocity in the draft tube center is visible, where the simulation slightly underestimates the velocity in the main area. Also over time a decreasing axial velocity is visible and in good agreement to the measurement.

In figure 7 time and spacial domain are combined in one contour plot. Along the abscissa the axial velocity at one time instance is plotted, where the ordinate shows the velocity at one measurement point along time. Qualitative, a good agreement between measurement and simulation is visible. At the beginning higher velocities are present near the draft tube cone and during the load change to part load the velocity in the draft tube center decreases.

Figure 6: averaged axial velocity distribution at top evaluation plane for part load (left), best efficiency (center) and high load (right)

Figure 7: axial velocity distribution at top evaluation plane of measurement (left) and simulation (right) for load change from best efficiency to part load
Figure 8: Axial velocity distribution at top evaluation plane for load change from best efficiency to part load at four different times and four different radial positions.
4.2.2. Best efficiency to high load

For the load change from best efficiency to high load the fluctuations of measured velocities are increasing, making a direct comparison of different points in the draft tube more difficult. But again a contour plot of the axial velocity can be plotted (cf. figure 9). Both contour plots showing again a good conformity. Towards high load the velocities sidewards of the draft tube cone are increasing, where in the center a stagnation region is visible.

![Figure 9: axial velocity distribution at top evaluation plane of measurement (left) and simulation (right) for load change from best efficiency to high load](image)

4.3. Turbine shutdown

In a last step a shutdown procedure is initiated. While the runner speed is kept constant the guide vanes are closing to speed-no-load conditions at $\alpha = 0,9^\circ$. Similar to previous investigations at least one second of simulation time before and after the guide vane movement is added to reach steady conditions. For the shutdown this results in a total simulation time of 9 s. Figure 10 shows again a comparison between measurement and simulation results for the axial velocity at four different time instances (figure 10a) and radial positions (figure 10b). Again the temporal and spacial agreement between both is satisfying. Increasing velocity fluctuations at small openings are visible and captured well by the simulation. Close to speed-no-load condition the axial velocity is in general slightly overestimated as seen for $t=7,5$ s in figure 10a and in the contour plot of figure 11.

As a last comparison the vaneless space pressure is investigated (cf. figure 12). Due to the adopted guide vane opening the pressure signal is normalized with best efficiency conditions. As expected the simulations show pressure surges at beginning and end of the guide vane movement/discharge variation. This cannot be seen in the measurement. Also the simulation is overestimating (load change to part load) or underestimating (load change to high load) the final pressure level. The assumption of a linear discharge variation may be a modeling error in the current simulations. An improvement could be made by defining an inlet total pressure according to the measurement and let the discharge a result of the head loss due to the current guide vane opening, or even consider the upstream hydrodynamics with a full 3D model or a simplified 1D3D coupling method.

Contrary to the first workshop the draft tube pressure sensors are of piezo electric-dynamic type and thus capture pressure fluctuations more accurately. To compare the pressure signals a procedure to obtain the instantaneous mean of the DT5 pressure fluctuations adapted from
Figure 10: axial velocity distribution at top evaluation plane for turbine shutdown at four different times and four different radial positions.
related investigations are used [12]. Instead of the Savitzky-Golay filter a median filter is used to preserve instant changes and lower frequencies. To capture fluctuations of the pressure-time signal the instantaneous mean pressure ($\bar{p}$) was then subtracted from the raw signal ($p$) and normalized with the energy at best efficiency condition.

$$\tilde{\rho} = p - \bar{p}$$

$$\tilde{\rho}_{E} = \frac{\tilde{\rho}}{(\rho E)_{BEP}}$$

A limitation of the fast Fourier transformation (FFT) is to provide simultaneous time and frequency localization and therefore not very useful for analyzing time-variant, non stationary signals. To overcome this problem, the signal is segmented into narrow time intervals and take the Fourier transformation of each segment. For each FT segment simultaneous spectral informations of time and frequency are available, called short time Fourier transformation (STFT). However this method comes with the drawback of the so called uncertainty principle. Time resolution and frequency resolution can not be made arbitrary small at the same time.

Figure 12: Normalized pressure at vaneless space for load changes from best efficiency to part load (left), to high load (center) and turbine shutdown (right)
and a compromise between both has to be chosen [13]. Figure 13 shows the amplitude pressure signal of the sensor DT5 in the draft tube extracted according to eq (3) and (4) as well as the resulting short time Fourier transformation for measurement and simulation. 

For the measurement at 166 Hz (30·fn) the rotor frequency is visible, where higher amplitudes at 40 Hz may be related to system exciting frequency [14]. At small openings perturbations at the draft tube measurement positions are increasing in intensity and frequency. Similar observations can be made for the simulation. As already seen in the vaneless space pressure signal, start and stop of the discharge variation are also introducing a pressure surge with oscillations, clearly visible at 1 s and 8.1 s. Increasing pressure amplitudes at 5–7 s and to a certain range at speed-no-load condition (>8 s) are captured by the simulation. The absence of higher frequencies after 5.6 s seems to be mesh related since the first re-mesh process was made at this position. Small mesh adaption was necessary to maintain a good mesh quality. This has to be investigated in further studies.

![Figure 13: pressure amplitude and STFT at draft tube pressure location DT5 of turbine shutdown for measurement (left) and simulation (right)](image)

5. Discussion

In the second stage of the Francis-99 workshop campaign numerical simulations of two load changes and a turbine shutdown were investigated. Based on the first workshop the most promising configuration was used to set up a 360° mesh including volute casing and draft tube. A moving mesh procedure with an automated remesh process was introduced to represent moving guide vanes up to small openings. Due to some low time response of discharge and pressure inlet measurement a linear discharge variation together with moving guide vanes were set as boundary conditions. First three steady operating points, best efficiency, part load and high load, were investigated and showed good agreement compared to the measurement. Change of axial velocity profiles during load changes starting from best efficiency going to part load and high load are captured well by the simulation. Also for the shutdown process the agreement was sufficient, where for small openings the velocities in general are slightly overestimated. Pressure signals of the simulation in the vaneless space and the draft tube are showing pressure surges at beginning and end of the discharge variation. This could not be seen in the measurement. Further studies have to improve the boundary conditions, to let the discharge variation be a result of the simulation. This can be achieved with a head variation according to the measurement or a constant upper reservoir pressure were the geometry upstream the turbine have to be fully modeled or considered with a simplified 1D model and coupled to the 3D domain. Also mesh dependency was seen for the frequency domain which has to be further investigated. As a last step starting from the end point of the shutdown process an already measured startup procedure will be modeled.
Nomenclature

| Term                          | Symbol | Unit  | Indices |
|-------------------------------|--------|-------|---------|
| Stiffness model exponent      | $C_{stiff}$ | -     | 0       |
| Velocity, axial               | $c_{ax}$ | m/s   | E       |
| Runner angular speed          | $n$    | 1/min |         |
| Speed factor                  | $n_{ed}$ | -     | BEP     |
| Pressure                      | $p$    | Pa    | DT      |
| Inst. mean pressure           | $p'$   | Pa    | FT      |
| Pressure amplitude            | $\tilde{p}$ | Pa | FFT     |
| Discharge                     | $Q$    | m$^3$/s |         |
| Discharge factor              | $Q_{ed}$ | -     | HL      |
| Dimensionless wall distance   | $y^+$  | -     | SC      |
| Guide vane angle              | $\alpha$ | °     | RU      |
| Displacement                  | $\delta$ | m     |        |
| Hydraulic efficiency          | $\eta_h$ | %    | SST     |
| Mesh stiffness                | $\Gamma_{disp}$ | -  | STFT    |
| Control volume size           | $\forall$ | m$^3$ | SV      |

Abbreviations

| Abbreviation | Description |
|--------------|-------------|
| BEP          | Best efficiency |
| DT           | Draft tube |
| FT           | Fourier transformation |
| FFT          | Fast Fourier transformation |
| GV           | Guide vanes |
| HL           | High load |
| PL           | Part load |
| RU           | Runner |
| SC           | Spiral case |
| SST          | Shear stress turbulence model |
| STFT         | Short time FT |
| SV           | Stay vanes |

References

[1] Nicolle J, Morissette J and Giroux, A 2012 Transient CFD simulation of a Francis turbine startup *IOP Conference Series: Earth and Environmental Science*

[2] Nicolle J, Giroux A and Morissette, J 2014 CFD configuration for hydraulic turbine startup *Proc. of the Hydraulic Machinery and Systems 27th IAHR Symposium Montreal*

[3] Li J, Yu J and Wu Y 2010 3D unsteady turbulent simulations of transients of the Francis turbine *Proc. of the Hydraulic Machinery and Systems 25th IAHR Symposium Montreal*

[4] Mössinger P and Jung A 2016 Transient two-phase CFD simulation of overload operating conditions and load rejection in a prototype sized Francis turbine *Proc. of the Hydraulic Machinery and Systems 28th IAHR Symposium Grenoble*

[5] Trivedi C, Cervantes M and Dahlhaug O 2016 Numerical techniques applied to hydraulic turbines: a perspective review *Applied Mechanics Review* **68**

[6] Trivedi C, Cervantes M, Ghandi B and Dahlhaug O 2013 Experimental and numerical studies for a high head Francis turbine at several operating points *J. of Fluid Engineering* **135**

[7] Cervantes M Second Francis-99 2016 Workshop *Norwegian University of Science and Technology, http://www.ntnu.edu/noks/second-workshop* accessed August 22, 2016

[8] Aschenbrenner T, Otto A and Moser W 2006 Classification of vortex and cavitation phenomena and assessment of CFD prediction capabilities *Proc. of the Hydraulic Machinery and Systems 23rd IAHR Symposium Yokohama*

[9] Mössinger P, Jester-Zürker R and Jung A 2015 Investigation of different simulation approaches on a high head Francis turbine and comparison with model test data: Francis-99 *J. of Physics: Conference Series* **579**

[10] ANSYS Inc: *ANSYS CFX-Solver Theory Guide, release 16.0*, 2015

[11] Trivedi C, Cervantes M, Ghandi B and Dahlhaug O 2014 Transient pressure measurements on a high head Francis turbine during emergency shutdown, total load rejection and runaway *J. of Fluid Engineering* **136**

[12] Trivedi C, Cervantes M, Dahlhaug O and Ghandi B 2015 Experimental investigation of a high head Francis turbine during spin-no-load conditions *J. of Fluid Engineering* **137**

[13] Kiymak K, Güler I, Dizibiyük A, and Akm M 2005 Comparison of STFT and wavelet transform methods in determining epileptic seizure activity in EEG signals for real-time application *Computers in Biology and Medicine* **35**

[14] Trivedi C, Cervantes J.M, Ghandi B.K and Dahlhaug O.G 2014 Pressure measurement on high-head Francis turbine during load acceptance and rejection *J. of Hydraulic Research* **52** 283-97