Transient two-phase CFD simulation of overload operating conditions and load rejection in a prototype sized Francis turbine

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

Abstract.
An increasing shift in operating conditions of hydropower turbines towards peak load operations comes with the necessity for numerical methods to account for such operations. This requires modifications to state-of-the-art CFD simulations. In the first part of this paper a 1D hydroacoustic model to represent the pressure oscillations in the penstock was introduced and coupled with a commercial CFD solver. Based on previous studies, various changes in cavitation and turbulence modeling were done to influence the behavior of a cavitating vortex rope typically occurring at high load conditions of a Francis turbine. In the second part, mesh motion was added to this model to simulate a load rejection starting from full load conditions. It was shown that additional extensions to the 3D CFD model are compulsory to model specific operating conditions as well as transient operations. Thus, accordance with measurement data at overload operation was improved and only small deviations remained. For the load rejection the maximum overspeed was well captured and the comparison of guide vane torques with model test measurements showed a sufficient agreement. With the gained insights, occurring effects which influence the performance and the life-time can be detected and conclusions for the hydraulic design as well as the operating mode can be drawn. Upcoming studies will focus on evaluating the flow field in detail and on reducing the remaining deviations by further extending the mathematical model.

1. Introduction
The increasing number of wind turbines and solar power plants require an extended operation range for hydro-power turbines to stabilize the power grid. This leads to additional challenges given that this operating modes need greater care regarding transient flows such as pressure peaks and fluctuations. CFD methods are state of the art to predict the hydraulic performance of a turbine at steady-state operation, but for more demanding cases like transient operations they still have some shortcomings. For instance if it is necessary to consider the interaction with the hydraulic waterway up-/downstream of the turbine or if due to load changes the guide vane opening angle is varying. So far not many investigations have been made on this field.

Flemming et al. [1] investigated the influence of two-phase modeling for overload surge trying to extract cavitation parameters like mass flow gain factor and cavitation compliance. Ruprecht et al. [2] considered the response of the waterway passage by means of a Method of Characteristic approach (MoC), where Chirkov et al. [3] used an one-dimensional hyperbolic system and a
two-phase approach. The investigations of the latter two research teams were done with in-house CFD solvers, in model size and without a comparison to measurement data. Transient operations considering moving guide vanes were performed by Nicolle et al. [4] simulating a start-up procedure by using a new mesh for every time step. Li et al. [5] were focusing on a load rejection process, modeling the whole plant including the penstock in a 3D manner with a tetrahedral unstructured mesh and dynamic mesh functionality. Similar investigations were carried out by Lui et al. [6] using measured pressure signals as inlet boundary condition to the spiral case. Zhang et al. [7] coupled a MoC approach to the 3D solver, considering a compressible water model and applying a dynamic mesh approach to simulate moving guide vanes during the load rejection. So far no investigations have been published that describe the coupling of a commercial CFD code to finite difference system, which is showing more flexibility in space and time discretization than the MoC approach. The same holds true for a direct comparison between prototype CFD results and measurement site data for the mentioned turbine operations.

The operation conditions discussed in this paper are overload and load rejection of a prototype sized Francis turbine. To accomplish this, a one-dimensional (1D) hydroacoustic model to represent the waterway upstream of the turbine is developed and coupled to a three-dimensional (3D) commercial CFD solver. In a second step an already available moving node approach of the commercial code is extended by an automated remesh procedure to represent moving guide vanes. Results of both operating conditions are compared with measurement site data. Benefits from this work are the possibility to represent conditions which were so far not able to simulate and therefore give the opportunity to get a full insight into the unsteady 3D flow field.

2. Hydroacoustic Modeling

Derived from the mass and momentum equation and after neglecting the convective terms because of high wave speeds at low flow velocities, the dynamic behavior of a pipe element with length $dx$ can be described according to eq (1):

$$\frac{\partial h}{\partial t} + \frac{a^2}{gA} \frac{\partial Q}{\partial x} = 0$$

$$\frac{\partial h}{\partial x} + \frac{1}{gA} \frac{\partial Q}{\partial t} + \frac{\lambda|Q|}{2gdA^2} Q = 0$$

(1)

As discussed in Nicolet [8] and Moessinger et al. [9] the hyperbolic partial differential equation system is discretized in space using a centered scheme on a staggered grid. Temporal differentiating is done using second order implicit Euler to be consistent with the commercial CFD solver. The resulting implicit matrix equation is solved with a Gauss-Seidel method using an additional under-relaxation factor to ensure stability and convergence. Test-cases to validate the 1D code with one-dimensional commercial code (SIMSEN) and 3D commercial CFD solver (ANSYS CFX) have been applied and described in previous studies [9].

3. Francis turbine at overload condition

3.1. Case study

At high load operation, Francis turbines feature a large axisymmetric cavitating vortex rope beneath the runner. Under certain conditions this cavitation volume can act as an internal energy source leading to instabilities. Serious pressure oscillations may occur due to the interaction of the oscillating vortex rope with the hydraulic system. When simulating this operating point it is crucial to account for the water-dynamic upstream, otherwise constant boundary conditions lead to unstable (as seen in figure 1 for constant discharge at the inlet) or unreliable results.

For this investigation a Francis turbine of the upper power range with unbranched bare steel penstock and a draft tube with direct outflow to the tailwater has been chosen. During
commissioning various measurements were carried out and led to a good size database. For comparison with the numerical results an operating point according to OP#1 in Table 1 is selected.

Table 1: operating conditions – overload

| Point | Head [m] | Discharge [m$^3$/s] | $P/P_{rated}$ | Guide vane opening [%] |
|-------|----------|----------------------|----------------|------------------------|
| OP#1  | 132      | 464                  | 1.035          | 100                    |

3.2. Numerical setup

The previously described and validated coupling is applied to model the above introduced power plant. To avoid scaling effects and take to the buoyancy into account, prototype sized modeling is chosen, which allows a direct comparison with measurement data. Considering the axisymmetric appearance of the vortex rope, it is assumed sufficient to model only one passage of stay vane, wicket gate and runner by applying periodical boundary conditions. The draft tube is modeled completely including elbow and piers. Transfer between stationary and rotating frame is realized by circumferentially averaging the flow variables (stage interface, wicket gate to runner) and by transient rotor-stator coupling (runner to draft tube), accounting for all transient flow characteristics. To capture the dynamic interaction between the cavitation volume and discharge variations together with pressure oscillations, a multiphase simulation is carried out. This comes along with the additional advantage that non-physical vorticity peaks in the vortex center can be avoided. The mass transfer between the phases representing evaporation and condensation is described by the Rayleigh-Plesset equation [10] and the Zwart-Gerber-Belamri cavitation model [11]:

\[
\dot{m}_{fg_{cond}} = F_{cond} \frac{3f_g \rho_g}{r_B} \sqrt{\frac{2}{3} \frac{|p_e - p|}{\rho_{fl}}} \text{sgn}(p_e - p) \tag{2}
\]

\[
\dot{m}_{fg_{vap}} = F_{vap} \frac{3f_{nu} (1 - f_g) \rho_g}{r_B} \sqrt{\frac{2}{3} \frac{|p_e - p|}{\rho_{fl}}} \text{sgn}(p_e - p) \tag{3}
\]
where the model constants are as follows:

\[ r_B = 2\mu m \quad F_{\text{cond}} = 0.01 \]
\[ f_{\text{nuc}} = 5e^{-4} \quad F_{\text{vap}} = 50 \]

Turbulent scales are modeled by the SST k-\( \omega \)-model with automatic wall function in previous studies and a scale-resolving SAS-SST model at current simulations. In general, ANSYS CFX 15.0 uses a finite volume based discretization scheme up to second order accuracy for convective fluxes and truly second order accuracy for diffusive fluxes. Time dependent computations are performed with a second order accurate time scheme. The computational grid consists of three domains (tandem cascade, runner and draft tube) with block-structured hexahedral cells. The mesh size of SVWG (865,000), RU (606,000) and DT (11,027,000) was kept constant for all investigations.

For the upstream 1D penstock inlet boundary condition, total energy is prescribed. Based on a measured pressure at the spiral case, pipe losses and the expected discharge according to the hill chart, the fixed total specific energy follows as:

\[
E_1 = \frac{p_{\text{statSC}}}{\rho g} + \frac{1}{2g} \left( \frac{Q}{A} \right)^2 + \frac{\Delta p_{\text{pipe}}}{\rho g}
\]  
(4)

At the 1D-3D interface an averaged pressure extracted from the 3D stay vane inlet is set as a 1D pressure boundary condition. A discharge return value of the 1D model in addition with an appropriate inflow angle serves as the 3D domain inlet condition. As buoyancy is considered the hydrostatic pressure profile at the draft tube outlet is set in accordance to the measured tail water level.

3.3. Flow visualization

Figure 2 shows the computational domain in a 3D view with the resulting vortex rope visualized by means of an isosurface corresponding to a volume fraction of \( \gamma = 0.5 \) in gray. The coloured
iso-surface represents the second invariant of the velocity gradient tensor [12] (Q criteria):

\[ Q = \frac{1}{2} \left( ||\Omega||^2 - ||S||^2 \right) \]  

(5)

where the color visualizes the ratio between the turbulent eddy viscosity \( \nu_t \) and the laminar molecular dynamic viscosity \( \nu \):

\[ \nu_{ratio} = \frac{\nu_t}{\nu} \]  

(6)

As the color range is in a logarithmic scale it is obvious, that for RANS simulations especially in the necking area of the vortex rope large viscosity ratios indicate a strong influence of the turbulence model. For the SAS-SST model, the viscosity ratio was significantly reduced. Additionally, downstream of the cavitation volume a vortex core of helical shape is visible. This phenomenon has been found as well by applying a SST-SAS turbulence model to Kaplan turbines [13].

3.4. Transient simulation
3.4.1. Previous results
In previous studies [9] the pressure oscillations were damped out when applying the 1D3D coupling (cf. figure 3) as a consequence of the underestimated rope frequency in the 3D-CFD simulation. To investigate the influence of the system eigenfrequency, three different test cases with three different pipe eigenfrequencies, reduced towards the rope frequency (\( f_{pipe}/f_{rope} = 2.1/1.4/1.1 \)), showed damped, stable and unstable amplitudes. This led to the conclusion that more effort has to be spent to improve the vortex rope frequency of the 3D simulation.

3.4.2. Current results
According to Brekke [14] the cavitating draft tube natural rope frequency can be described by the analytical equation:

\[ f_{rope} = \left( \frac{1}{2\pi} \right) \sqrt{\frac{gA}{LA_{eq}}} \]  

(7)
where the equivalent area states:

\[ A_{eq} = \frac{V_0}{\kappa H_0} \]  \hspace{1cm} (8)

According to eq. (8) cavitation volume and the associated pressure level affect the frequency. Various modifications regarding the cavitation and turbulence modeling have been made to influence the vortex rope behavior. Figure 4 shows the FFT of the dynamic pressure at draft tube location for the measurement and 1D3D coupling of previous and current results. Compared to the measurement, the modifications substantially improve the rope frequency. After an initialization phase, the amplitudes reach a stable level with some underlying low frequency. Figure 5 shows a smaller time interval of the simulated and measured pressure oscillations. The slightly underestimated amplitudes could be a consequence of the remaining discrepancies in the frequency range. Further studies will include different cavitation models and take the compressibility of both phases into account.
4. Francis turbine at load rejection

4.1. Case study
Load rejection occurs when a turbine in operation is suddenly disconnected from the grid. To limit the resulting overspeed of the runner, guide vanes have to close at the maximum feasible speed and prevent undesired pressure peaks at the same time. As shown in figure 6, during guide vane closure the turbine accelerates to an overspeed of 150% of the nominal runner speed. Starting at overload, the vortex rope in the draft tube collapses, the pressure suddenly drops and rises with further closure of the guide vanes to a stable level as the guide vanes are fully closed.

![Figure 6: measured values for load rejection process](image)

4.2. Numerical setup
After disconnecting from the grid, runner speed is no longer constant but depends on the runner torque and runner/generator inertia. This can be expressed in an explicit equation which will be solved after every time step.

\[ I \frac{d\omega}{dt} = M_R \] (9)

The flow field during the load rejection is highly instationary and asymmetric. Therefore it is necessary to take all stay vanes, guide vanes and runner passages as well as the spiral case into account. This sums up in a total mesh size for the block structured hexahedral grid of \(25.2 \cdot 10^6\) cells. Transition between the domains are set to general grid interfaces (GGI), where the transfer between stationary and rotating domain is handled by transient rotor stator coupling, accounting for all transient flow characteristics. To ensure stability, turbulence scales are modeled by the SST \(k-\omega\)-model with automatic wall functions and a time step which corresponds to 1/72 of the initial runner revolution.

4.3. Adaptive grid method
Moving guide vanes during load rejection are realized with a combination of node motion and automated remeshing. ANSYS CFX already comes with the ability to deform meshes [15]. While the boundary movement is specified, all remaining nodes shift according to a displacement diffusion by solving the equation:

\[ \nabla (\Gamma_{disp} \nabla \delta) = 0 \] (10)

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

\[ \Gamma_{disp} = \left( \frac{v_{ref}}{\nabla} \right)^{C_{stiff}} \] (11)
For this case, larger control volumes absorb more mesh motion. An increase of the stiffness model exponent $C_{stiff}$ from 2 (default) to 15 yields to a much more abrupt stiffness variation, ensuring a desirable mesh quality along the guide vane closure.

However during the load rejection process, starting from overload, total guide opening angle varies over $32^\circ$. This angular rotation is too large to ensure a good mesh quality along the complete guide vane closure using only the mesh deformation method. Therefore an automated remesh process is introduced, allowing a remesh procedure during runtime (cf. figure 7). Depending on an arbitrary logical condition (maximum runtime, minimum cell angle) 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.

![Figure 7: remesh condition](image)

### 4.4. Transient simulation

In preliminary investigations a reduced model domain and simulation complexity was chosen. But as seen in figure 8, the use of a single flow passage as done in the studies in chapter 3 or even a full $360^\circ$ consideration of the domain, with a constant total pressure as inlet boundary condition,

![Figure 8: speed curve for different simulation approaches](image)
lead to a massive overprediction of the runner speed. By coupling the 1D code to represent the pressure rise due to the closing wicket gates a substantial improvement of the speed prediction is achieved. A pure 1D approach (SIMSEN) is also able to predict the maximum overspeed well, but shows some discrepancies when the runner speed is decreasing. For a final statement on how the predicted runner speed of the CFD simulation will develop, more simulation time is necessary. Figure 9 shows a comparison between model test measurement and prototype CFD of the average torque of four wicket gates. From the beginning the torque is slightly overpredicted with increasing deviations during the closure process. However the development from opening to closing tendency is captured sufficient well. Steady state simulations of different openings will follow and give an indication if these deviations are related to the transient closing procedure. Furthermore, time step size variation and mesh adaption are other starting points to improve the simulation. Thereafter a more detailed comparison of additional measurement points will be possible.

5. Conclusion
A previously developed and validated 1D-3D coupling method was further analyzed. When applied to a prototype sized Francis turbine at overload conditions, accordance with measurement data was improved by adjustments regarding cavitation and turbulence modeling. Upcoming studies about different cavitation models and the consideration of compressibility for both phases are aiming at further reducing the remaining deviations.

In a second application, full condition load rejection was investigated. The guide vane closure was taken into account by moving mesh motion. The runner speed was modeled as a function of runner torque and runner/generator inertia. Simplified models led to a significant overprediction of maximum runner torque. By considering a full 360° computational domain and including the penstock by the already introduced 1D code, a better accordance with measurement data and commercial 1D solver (SIMSEN) was achieved. Also a comparison of guide vane torque showed at the beginning good agreement, with increasing deviations during the load rejection process. After a detailed evaluation of the simulation more effort has to be put in the computational model to ensure stable convergence and reasonable results along the whole closing procedure.
Nomenclature

| Term                  | Symbol | Unit  | Term                  | Symbol | Unit  |
|-----------------------|--------|-------|-----------------------|--------|-------|
| Water wave speed      | a      | m/s   | Volume fraction       | γ      | -     |
| Pipe cross-section    | A      | m²    | Mesh stiffness        | Γ      | -     |
| Distributer pad height| b₀     | m     | Displacement          | δ      | m     |
| Pipe diameter         | d      | m     | Isentropic exponent   | κ      | -     |
| Total specific energy | E      | m     | Darcy friction factor | λ      | -     |
| Frequency             | f      | Hz    | Viscosity             | ν      | m²/s  |
| Gravity acceleration  | g      | m/s²  | Angular frequency     | ω      | rad/s |
| Piezometric pressure  | h      | m     | Vorticity tensor      | Ω      | -     |
| Hydraulic head        | H      | m     | Nabla operator        | ∇      | -     |
| Inertia               | I      | kg m² | Control volume size   | ∀      | m³    |
| Runner torque         | Mₚ     | Nm    | Boundary condition    | BC     |       |
| Static pressure       | p      | Pa    | Draft tube            | DT     |       |
| Power                 | P      | MW    | Runner                | RU     |       |
| Discharge             | Q      | m³/s  | Scale adaptive simulation | SAS |       |
| Strain-rate tensor    | S      | -     | Shear-Stress-Transport | SST |       |
| Time                  | t      | s     | Stay vane / wicket gate | SVWG |       |
| Guide vane pitch      | t₀     | m     |                        |        |       |
| Cavitation volume     | V₈     | m³    |                        |        |       |
| Length along pipe     | x      | m     |                        |        |       |

Abbreviations

| Boundary condition   | BC     |
| Draft tube           | DT     |
| Runner               | RU     |
| Scale adaptive simulation | SAS |       |
| Shear-Stress-Transport | SST |       |
| Stay vane / wicket gate | SVWG |       |

References

[1] Flemming F, Foust J, Koutnik J and Fischer R 2009 Overload surge investigation using CFD data Int. J. of Fluid Machinery and Systems 2 315-23
[2] Ruprecht A, Helmarich T, Aschenbrenner T and Scherer T 2002 Simulation of vortex rope in a turbine draft tube Proc. of the Hydraulic Machinery and Systems 21st IAHR Symposium Lausanne
[3] Chirkov D, Avdyushenko A, Panov L, Bannikov D, Cherny S, Skorospelov V and Pylev I 2012 CFD simulation of pressure and discharge surge in Francis turbines at off-design conditions Proc. of the Hydraulic Machinery and Systems 26th IAHR Symposium Beijing
[4] Nicolle J, Morissette J and Giroux A 2012 Transient CFD simulation of a Francis turbine startup IOP Conference Series: Earth and Environmental Science 15(6)
[5] Li J, Yu J and Wu J 2010 3D unsteady turbulent simulations of transients of the Francis turbine Proc. of the Hydraulic Machinery and Systems 25th IAHR Symposium Timisoara
[6] Liu J, Liu S, Sun Y, Wu Y and Wang L 2013 Three dimensional flow simulation of load rejection of a prototype pump-turbine Engineering with Computers 29 417-26
[7] Zhang X and Cheng Y 2014 Simulation of the load rejection transient process of a Francis turbine using a 1-D-3-D coupling approach J. of Hydrodynamics 26 715-24
[8] Nicolet C 2006 Hydroacoustic Modelling: Application to Francis turbines [Dissertation] École Polytechnique Fédérale des Lausanne
[9] Mössinger P, Conrad P and Jung A 2014 Transient two-phase CFD simulation of overload pressure pulsation in a prototype sized Francis turbine considering the waterway dynamics Proc. of the Hydraulic Machinery and Systems 27th IAHR Symposium Montreal
[10] Brennen C 1995 Cavitation and bubble dynamics New York: Oxford University Press
[11] Zwart P, Gerber A and Belamri T 2004 A two-phase model for predicting cavitation dynamics ICMF Int. Conf. on Multiphase Flow Yokohama
[12] Hunt J, Wray A and Moin P 1988 Eddies, streams, and convergence zones in turbulent flows Center for Turbulence Research, Proceedings of the Summer Program
[13] Junginger B and Riedelbauch S 2015 Numerical investigation of influence of runner gap for a low head turbine using hybrid RANS-LES turbulence model Proc. of the Hydraulic Machinery and Systems 25th IAHR WG Symposium Ljubljana
[14] Brekke H 2010 A review on work on oscillatory problems in Francis turbines New Trends in Technologies: Devices, Computer, Communication and Industrial Systems
[15] ANSYS Inc. 2013 ANSYS CFX-Solver Theory Guide, Release 15.0