Two-phase simulations of the full load surge in Francis turbines

J Wack and S Riedelbauch
Institute of Fluid Mechanics and Hydraulic Machinery, University of Stuttgart, Pfaffenwaldring 10, 70550 Stuttgart, Germany
E-mail: jonas.wack@ihs.uni-stuttgart.de

Abstract. At off-design conditions, Francis turbines experience cavitation which may reduce the power output and can cause severe damage in the machine. Certain conditions can cause self-excited oscillations of the vortex rope in the draft tube at full load operating point. For the presented work, two-phase simulations are carried out at model scale on a domain ranging from the inlet of the spiral case to the outlet of the draft tube. At different locations, wall pressure measurements are available and compared to the simulation results. Furthermore, the dynamics of the cavity volume in the draft tube cone and at the trailing edge of the runner blades are investigated by comparing with high speed visualization. To account for the self-excited behaviour, proper boundary conditions need to be set. In this work, the focus lies on the treatment of the boundary condition at the inlet. In the first step, the dynamic behaviour of the cavity regions is investigated using a constant mass flow. Thereafter, oscillations of the total pressure and mass flow rate are prescribed using various frequencies and amplitudes. This methodology enables to examine the response of the cavity dynamics due to different excitations. It can be observed that setting a constant mass flow boundary condition is not suitable to account for the self-excited behaviour. Prescribing the total pressure has the result that the frequency of the vapour volume oscillation is the same as the frequency of the excitation signal. Contrary to that, for an excitation with a mass flow boundary condition, the response of the system is not equal to the excitation.

1. Introduction
The integration of renewable energies like wind or photovoltaics requires other technologies that can compensate fluctuations in energy generation for the issue of electrical grid stability. In this field hydropower plays a key role as it has a flexible operating range. However, this necessitates operating turbines at off-design conditions which results in complex flow phenomena like cavitation.

Cavitation is an important topic in the design of hydraulic machinery as it can result in severe damage of the turbine and has an impact on the efficiency. When operating at off-design conditions, both, the rotating runner and stationary parts of the machine can be affected by cavitation. The location and occurrence of cavitating flow regimes in the turbine strongly depend on the discharge and head [1].

At full load operating conditions, the flow in the draft tube generates an axisymmetrically shaped vortex rope that may form a cavitation volume depending on the pressure level. The investigated full load operating point is characterized by a self-excited behaviour. Comprehensive measurements that have been carried out by Müller et al. [2] indicate that the self-excited behaviour of the draft tube vortex rope is a result of a change in the flow swirl due to cavitation on the runner blades.
Numerically, several studies have been carried out for self-excited pressure surges at full load conditions. Dörfler et al. [3] made 2D investigations using three different constellations of boundary conditions. Several authors [4, 5] use a 1D-3D coupling method in their simulations. Therein, the penstock is modelled by a 1D-approach while the turbine is simulated 3D.

In the scope of this work, different excitation signals are prescribed at the inlet boundary condition. A similar approach has been used by Decaix et al. [6] on a Venturi nozzle test case. There, the excitation has been prescribed at the outlet boundary condition by specifying a sinus signal for the static pressure. The results show that for the excitation with resonance frequency, the amplitude of the vapour volume oscillation is multiple times larger compared to an excitation at non-resonance conditions.

2. Mathematical model

Cavitation modelling is more complex compared to single phase simulations due to the interaction between the phases. There are several different approaches to simulate cavitation. A detailed review, for instance, is given by Koop [7]. For the current study, the simulation software ANSYS CFX is used applying the homogeneous model [8]. Hereafter, the subscript l denotes the liquid phase, v the vapour phase and m the mixture of the two phases.

2.1. Governing Equations

The homogeneous model is based on the assumptions that the relative motion between the two phases – liquid and vapour – can be neglected and both phases share a common pressure field. This enables to consider the mixture of both phases as a pseudo-fluid. The governing equations that describe the flow of the liquid-vapour-mixture are the continuity equation, the momentum equation and a transport equation for the volume fraction.

\[
\frac{\partial \rho_m}{\partial t} + \frac{\partial (\rho_m U^i)}{\partial x_j} = 0
\]

\[
\frac{\partial (\rho_m U^i)}{\partial t} + \frac{\partial (\rho_m U^i U^j)}{\partial x_j} = \frac{\partial p}{\partial x_i} + \tau_{ij}^{ml} + \rho_m g^i
\]

Due to the fact that the relative motion between the two phases is neglected, both share a common velocity field \(U\). In the transport equation of the volume fraction, \(\Gamma_{lv}\) denotes the mass transfer rate due to cavitation. It has to be modelled by a mass transfer model, also called cavitation model.

2.2. Mass transfer model

The mass transfer from one phase to the other is depending on the local pressure level and has to be modelled by a cavitation model. There are several mass transfer models that can be used for cavitation. In the present study the Zwart model [9] is used which is based on the Rayleigh-Plesset equation [8].

\[
\Gamma_{lv} = F_v \frac{3a_{nuc}(1-\alpha_v)\rho_v}{R_{nuc}} \sqrt{\frac{2p_{vap}p}{3\rho_l}} \quad \text{if} \quad p < p_{vap}
\]

\[
\Gamma_{lv} = F_c \frac{3a_{nuc}\rho_v}{R_{nuc}} \sqrt{\frac{2p p_{vap}}{3\rho_l}} \quad \text{if} \quad p > p_{vap}
\]

In the model, the evaporation process is activated when the pressure falls below the vapour pressure \(p_{vap}\). Condensation takes place when the vapour pressure is exceeded. Furthermore, there exist four model parameters. \(F_v\) and \(F_c\) are empirical calibration coefficients for the evaporation and condensation process. \(R_{nuc}\) and \(\alpha_{nuc}\) denote the radius and volume fraction of the nucleation site. The
following recommended model parameters are used in this study: $F_v=50$, $F_c=0.01$, $R_{\text{nucl}}=1 \, \mu\text{m}$ and $\alpha_{\text{nucl}}=5\cdot10^{-4}$ [9]. Investigations of Morgut et al. [10] on model scale propellers indicate that the usage of other mass transfer models results in similar results of the cavitation behaviour.

3. Numerical setup

The numerical simulations are carried out on a Francis turbine at model scale. The flow domain, which is displayed in figure 1, ranges from the inlet of the spiral case to the outlet of the draft tube. Figure 1 contains the pressure measuring points C1N and D1. In the following investigations, a structured mesh of 11 million nodes is used and a transient rotor stator approach is applied at the interfaces between stationary parts and the rotating runner. All simulations are performed with the SST turbulence model. For spatial discretisation a high resolution scheme is applied and for temporal discretisation a second order backward Euler scheme. In the following investigations, different boundary conditions at the inlet are investigated while the boundary condition at the outlet is set to a constant pressure in all cases.

For two-phase simulations the pressure level plays a significant role as the mass transfer from one phase to the other is depending on the vapour pressure $p_{\text{vap}}$. For experiments, the pressure level typically is specified by the cavitation number $\sigma$ which is defined as the ratio of net positive suction head NPSH and the head $H$. For the numerical simulation of a Francis turbine it is more feasible to choose a definition that sets the correct pressure level in the region where cavitation occurs. Thus, for the following investigations the pressure level is set according to pressure measurements in the draft tube cone.

Two-phase simulations are numerically more complex. Especially when it comes to a collapse of the vapour region, the occurring pressure peak may cause the simulation to diverge. Increasing the number of coefficient loops during the cavity collapse has a stabilizing effect on the simulation. For the investigated operating point, a total number of 15 coefficient loops per time step is applied to overcome the cavity collapse.

4. Numerical results

In the scope of this work, different boundary conditions at the inlet are investigated. The goal is to examine, which boundary condition is suitable to capture the phenomena of the pressure surge. Furthermore, the system response to different excitations frequencies is investigated.

The characteristics of the investigated full load operating point are displayed in table 1. The discharge is 30% above the discharge of the best efficiency point (BEP).

| $Q_{\text{ED}}$ [-] | $Q/Q_{\text{BEP}}$ [-] | $n_{\text{ED}}$ [-] | $\sigma$ [-] |
|---------------------|------------------------|---------------------|--------------|
| 0.26                | 1.3                    | 0.288               | 0.11         |

Figure 1. Simulation domain and pressure measuring points.
4.1. Constant mass flow
First, a constant mass flow at the inlet is prescribed. For the initialisation of the simulation a single phase simulation at high pressure level is carried out. After switching to a two-phase simulation, the pressure level at the outlet is reduced until the required sigma number is reached.

![Figure 2](image1.png)  ![Figure 3](image2.png)

**Figure 2.** Static pressure at the outlet and at measuring point C1N and vapour volume. **Figure 3.** Static pressure at measuring point C1N – constant mass flow.

Figure 2 shows the course of the static pressure at the outlet and the measuring point C1N as well as the vapour volume. After one runner revolution the simulation is switched from single phase to two-phase and after three runner revolutions the pressure level at the outlet is reduced. Thus, the vapour volume is starting to increase. However, due to the unstable behaviour of the vortex rope, it comes to a sudden collapse of the cavity volume which results in a severe pressure peak at the measuring point. Thereafter, a bigger cavity volume forms that shows an oscillating behaviour but does only once reproduce the cavity collapse from the measurements after approximately 13 runner revolutions. A comparison of the cavity size with high speed visualisation images from the measurements [2] indicates that a constant mass flow is not suitable as inlet boundary condition. With this boundary condition the size of vapour volume is predicted too small by the simulation. Furthermore, the dynamics of the self-excited pressure surge cannot be reproduced correctly as displayed in figure 3. Thus, other boundary conditions have to be applied for this operating point.

4.2. Total pressure signal from measurements
In the next step, it is investigated whether the applied modelling approach is suitable to capture the phenomena of this operating point. Thus, a signal for the total pressure is applied at the inlet boundary condition that is estimated by using pressure measurements at a point close to the spiral case inlet. It must be emphasised that this signal is based on the average pressure surge frequency of the measurement.

Figure 4 shows the simulation and measurement results of the static pressure at the location C1N. It can be stated that the simulation meets the low pressure level plateau well because the pressure level at the outlet is set according to this. The pressure surge due to the cavity collapse indicates differences. The simulation slightly overestimates this region. However, it has to be stated that the general course of the measurement is met well. Evaluating the frequency of the pressure surge in figure 4, it can be noted that the simulation does not perfectly match the frequency of the measurement. This can be explained by the fact, that the occurrence of the cavity collapse in the measurement is in a certain frequency range while the simulation is based on the average pressure surge frequency of the measurement.
The results for measuring point D1 are presented in figure 5. The pressure level of the simulation is in good agreement to the measurement. That leads to the conclusion that the losses in the draft tube are well reproduced by the simulation. At runner revolution 1 and 5, the simulation and the measurement show a small peak, which can be explained by the cavity collapse in the runner as observed with a high speed video. The measurement shows small oscillations in the pressure signal with approximately twice the runner frequency that are not present in the simulation. This might come from the constant pressure that is prescribed at the outlet boundary condition. Due to the location of the measuring point close to the outlet boundary condition, these oscillations might not be present in the simulation.

For the investigation of the cavity volume dynamics, the course of the vapour volume in runner and draft tube over time is displayed in figure 6. Both, runner and draft tube experience an oscillating vapour volume. The maximum cavity volume in the draft tube appears earlier compared to the runner. Furthermore, the cavity collapse in the runner is earlier than in the draft tube. Both of these characteristics have also been detected in the measurements (see figure 7) [2].

At three different instants of the pressure oscillation (see figure 6), the cavitation volume at the runner blades is shown in figure 7. The cavity volume of the simulation is displayed with an isosurface of the volume fraction of the liquid phase. For the vortex rope, a reasonable visual agreement can be stated between simulation and measurement during one cycle of pressure surge. Just the collapse of the vortex rope does not occur in the simulation so that a small vortex rope remains in the draft tube. The cavity volume in the runner is well met for the instants b and c. At instant a, the simulation overestimates the vapour volume. This might result from the fact that the pressure surge in the simulation is forced from the boundary condition at the inlet.
The results indicate that the used modelling approach is suitable for the simulation of this operating point. However, the usage of the investigated signal at the inlet boundary condition has the disadvantage of being dependent on experimental data. Thus, it is investigated in the following how the pressure surge is affected by different excitation signals.

4.3. Sinus signal – total pressure
A sinus signal is prescribed at the inlet boundary condition for the total pressure. The excitation is varied in two different ways – change in amplitude and variation of frequency. All following simulation results start with the same initialisation. This initialisation is based on the setup f100, which has as excitation frequency the average pressure surge frequency of the measurement. For the different amplitudes and frequencies at the beginning of every simulation a cross-fading to the particular case takes place.

4.3.1. Frequency variation. For the frequency variation, setup f100 and three additional excitation frequencies are investigated. One excitation frequency is reduced by 25 % (f075) compared to the basic configuration. The other setups are characterized by an increased excitation frequency by 25 % (f125) and 50 % (f150). The results for the vapour volume in runner and draft tube and the prescribed signal of the total pressure at the inlet boundary condition are presented in figure 8.

A comparison of the different setups shows that the frequency of the vapour volume oscillation is identical with the excitation frequency. The minimum of vapour volume in the draft tube occurs around the time when the total pressure at the inlet reaches its maximum and vice versa. This agrees with the expectations, as a low total pressure indicates a low level of static pressure which results in an increased vapour volume.
Regarding the resulting amplitude of the vapour volume oscillation in runner and draft tube, only minor differences can be observed. For both – maximum and minimum value of the vapour volume – all simulations are in the same range. The only noticeable difference that can be observed is the course of the vapour volume in the runner. For case f075, the vapour volume stays longer at a low level compared to the other cases. Case f100 shows a short time of stagnation during the period of increasing vapour volume. The other two cases show none of these characteristics. This behaviour might be a result of the differences in the total pressure gradient between two extrema for different frequencies. For high frequencies the gradient is steeper which causes the vapour volume to grow and shrink faster.

4.3.2. Amplitude variation. For the amplitude variation, as standard configuration the setup f100 from the frequency investigations is used and the amplitude of the excitation signal is varied. For the two other configurations the amplitude of the total pressure signal is reduced (f100a-) or increased (f100a+) by 50 % which corresponds to approximately 1 % of the head. The signal at the inlet boundary condition of the three investigated cases and the simulation results for the vapour volume in draft tube and runner are displayed in figure 9.

As expected, the highest amplitude of excitation causes the highest amplitude of vapour volume oscillation and the smallest oscillation is forced by setup f100a-. Furthermore, it can be observed for most of the oscillation cycles that the maximum vapour volume in draft tube and runner occurs earlier with decreasing amplitude. An investigation of the vapour volume oscillation in the runner shows that
for f100 and f100a+ all cycles show a similar behaviour. At the end of every cycle the vapour disappears and all maxima of vapour volume are in the same range.

For the reduced amplitude a different behaviour is observable. The first three cycles from runner revolution 8 to around 19 are characterised by a damping of the vapour volume oscillation in the runner. This might be caused by the fact that the oscillation is still affected by the initial conditions. In the third cycle (runner revolution 15 to 19) both – maximum and minimum vapour volume – occur significantly earlier compared to the other cycles. As a result, this phase shift might interact with the excitation at the inlet boundary condition and cause the significant increase of amplitude in the vapour volume oscillation for the next cycle. A similar behaviour can be found in the draft tube. There, the phase change of the third cycle and the increased amplitude of vapour volume in the fourth cycle are also apparent. However, the damping of vapour volume in the first three cycles is not observed in the draft tube.

Figure 9. Oscillation of the vapour volume in draft tube (DT) and runner (RU) for different excitation amplitudes.

4.4. Sinus signal – mass flow
In the following, a sinus signal of the mass flow is prescribed at the inlet boundary condition. Two different excitation frequencies (f125m and f150m) are investigated that are equal to the frequencies of the configurations f125 or f150, respectively. The simulation results for the vapour volume oscillations in draft tube and runner are displayed in figure 10.

The results observed with the mass flow boundary condition show a different behaviour compared to the simulations excited with a sinus signal for the total pressure. The frequency of the vapour volume oscillation is not identical with the excitation frequency. For case f125m no clear response frequency can be observed. However, as the vapour volume oscillation does not match with the excitation frequency, over a wide range of cycles the vapour volume oscillation in the draft tube is characterized by a damping. Nevertheless, at certain conditions the oscillation gets excited again. A comparison of the excitation signal with the vapour volume oscillation indicates that when the minimum of vapour volume occurs at the same time when a maximum in the mass flow appears (at runner revolutions 19.5 and 34.5), the oscillation gets excited. If the minimum vapour volume coincides with a minimum in mass flow (runner revolution 15 and 30) then the oscillation experiences a strong damping.

For case f150m it can be observed that the frequency of the vapour volume oscillation is in the range of the average frequency of the pressure surge observed in the measurements. Regarding the vapour volume oscillation in the draft tube it can be observed that cycles with high amplitudes and cycles with low amplitudes are alternating. Comparing the oscillation with the excitation signal supports the facts observed in case f125m. Thus, every second cycle experiences strong excitation
(e.g. after runner revolution 29.5) while the other cycles experience strong damping (e.g. after runner revolution 33).

Figure 10. Oscillation of the vapour volume in runner and draft tube for different excitation frequencies at the inlet using a mass flow boundary condition.

5. Conclusion and outlook
In the current study, two phase simulations are performed on a Francis turbine at model scale for a full load operating point, which is characterized by self-excited behaviour. Different boundary conditions with and without excitation are applied at the inlet. Setting a constant mass flow at the inlet of the spiral case is not suitable to capture the oscillations of the vortex rope. With a prescribed signal of total pressure derived from measurements, the simulation results show a good agreement with the measurements. Thus, it can be stated that the used modelling approach can capture the existing phenomena. However, this method is only advisable for basic modelling checks as it depends on the measurement results.

To overcome this problem, different excitation signals are applied. The frequency investigation using a sinus signal for the total pressure at the inlet boundary condition shows that the excitation of the system leads to a direct response of the system with the same frequency. For this analysis, the amplitudes of the vapour volume oscillation in runner and draft tube show only minor differences. The amplitude variation for constant frequency indicates that with reduced amplitude the appearance of the maximum vapour volume during one cycle is earlier. Furthermore, the expected behaviour can be observed that with increasing amplitude of the excitation the amplitude of the vapour volume oscillation is increasing.

Exciting the system with a mass flow boundary condition shows a different behaviour compared to an excitation with the total pressure. With the mass flow boundary condition, the system does not respond directly to the excitation. Thus, the pressure surge of the simulation can have a different frequency compared to the excitation. However, this leads to different oscillation states that damp or excite the oscillation of the vortex rope.

Further investigations are necessary especially for different mass flow excitation signals to better understand the response of the pressure surge to different excitations. A 1D-3D coupling approach might be necessary to examine the interaction between the test rig and the turbine. Furthermore, the investigation of this operating point at different sigma levels might allow for a prediction of the operating range where self-excited behaviour occurs.
Acknowledgments

The research leading to the results published in this paper is part of the HYPERBOLE research project, granted by the European Commission (ERC/FP7-ENERGY-2013-1-Grant 608532). Special thanks to EPFL, in particular Andres Müller, Keita Yamamoto, Arthur Favrel, Christian Landry and François Avellan, for providing all experimental data enabling comparisons presented here.

All the computations in this paper were performed on the Cray XC40 in High Performance Computing Center of Stuttgart (HLRS). HLRS is gratefully acknowledged for their support for this research.

References

[1] Avellan F 2004 Introduction to cavitation in hydraulic machinery The 6th Int. Conf. on Hydraulic Machinery and Hydrodynamics, Timisoara, Romania
[2] Müller A, Favrel A, Landry C, Yamamoto K and Avellan F 2014 On the physical mechanisms governing self-excited pressure surge in Francis turbines IOP Conf. Series: Earth and Environ. Sci. vol 22 (IOP Publishing) p 032034
[3] Dörfler P, Keller M and Braun O 2010 Francis full-load surge mechanism identified by unsteady 2-phase CFD IOP Conf. Series: Earth and Environ. Sci. vol 12 (IOP Publishing) p 012026
[4] Chirkov D, Panov L, Cherny S and Pylev I 2014 Numerical simulation of full load surge in Francis turbines based on three-dimensional cavitating flow model IOP Conf. Series: Earth and Environ. Sci. vol 22 (IOP Publishing) p 032036
[5] 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 IOP Conf. Series: Earth and Environ. Sci. vol 22 (IOP Publishing) p 032033
[6] Decaix J, Alligné S, Nicolet C, Avellan F and Münch C 2015 Identification of the wave speed and the second viscosity of cavitation flows with 2D RANS computations - part I J. Phys.: Conf. Series vol 656 (IOP Publishing) p 012060
[7] Koop A H 2008 Numerical simulation of unsteady three-dimensional sheet cavitation Ph.D. Thesis University of Twente, Enschede
[8] ANSYS CFX 2013 ANSYS CFX - Solver theory guide (release 15.0)
[9] Zwart P J, Gerber A G and Belamri T 2004 A two-phase flow model for predicting cavitation dynamics Proc. of the 5th International Conf. on Multiphase Flow, Yokohama, Japan
[10] Morgut M and Nobile E 2012 Numerical predictions of cavitating flow around model scale propellers by CFD and advanced model calibration Int. J. of Rotating Machinery 2012