Numerical simulations of the cavitation phenomena in a Francis turbine at deep part load conditions

J Wack and S Riedelbauch

Institute of Fluid Mechanics and Hydraulic Machinery, University of Stuttgart, Pfaffenwaldring 10, 70555 Stuttgart, Germany

E-mail: jonas.wack@ihs.uni-stuttgart.de

Abstract. In recent years, the operating range of hydraulic machines has been more and more extended. As a consequence, the turbines are facing off-design conditions with highly complex flow phenomena like cavitation. In the present study, the occurrences of cavitating inter blade vortices at deep part load conditions in a Francis turbine are investigated using two-phase flow simulations. The numerical simulations require small time steps and fine meshes to reproduce the required flow characteristics and resolve the minimum pressure in the vortex core. Furthermore, the treatment of the outlet boundary condition is important, as this operating point is facing severe backflow in one diffusor channel in the draft tube. The simulation results indicate that the inter blade vortices can be reproduced.

1. Introduction

Cavitation is a key factor in the hydraulic design of turbines. For a Francis turbine different types of cavitation can occur, depending on the head and discharge [1]. In the present study, the focus is on the inter blade vortices that occur at deep part load conditions. Deep part load conditions are characterized by a low discharge. The small guide vane opening causes a misaligned flow to the runner tip which results in diverse unsteady flow phenomena like inter blade vortices. These vortices can cause severe mechanical vibrations over the whole hydro power plant. Penstock vibrations that can be traced back to this cavitation phenomenon are reported by Dörfler et al. [2].

Only little research has been performed on deep part load in the past. Investigations performing single-phase numerical simulations show that deep part load conditions are accompanied by severe pressure pulsations with possibly higher amplitudes compared to full load and part load conditions [3]. First predictions of the inter blade vortex limits with numerical simulations have been carried out by Zuo et al. [4]. There, two-phase simulations showed a better accuracy compared to single-phase simulations. An experimental study has been carried out by Yamamoto et al. [5]. It is reported that the speed factor n_{ED} has a strong impact on the occurrence of the inter blade vortices.

2. Modelling

Several different concepts are applicable for two-phase modeling [6]. In the present study, the simulations are performed based on the homogenous model [7]. It allows considering the mixture as a pseudo-fluid. Thus, the governing equations consist of the mass conservation, one set of momentum equations and a transport equation for the volume fraction.

The phase change due to cavitation is considered as source term in the transport equation for the volume fraction and has to be modeled by a cavitation model. In the present study, the Zwart model is applied. It contains four parameters that are set according to the recommendation by Zwart et al. [8].

3. Numerical setup

The numerical simulations are performed on a reduced scale model Francis turbine with the flow domain ranging from the inlet of the spiral case to the outlet of the draft tube as displayed in figure 1. For the investigation of the treatment at the outlet boundary condition, one additional setup contains a tank that is attached to the draft tube outlet. In the present study, structured meshes are used and three different mesh sizes are investigated. The smallest mesh consists of approximately 11 million (11M) elements. The 26M mesh is refined in all domains but with the focus on the runner and the draft tube to resolve regions of strong vortex flow and cavitation occurrence. The 35M mesh has a strong refinement in the runner to account for resolving the core of the inter blade vortices.

All simulations are carried out with Ansys CFX. The calculations are performed with a transient rotor stator approach. At the inlet boundary condition, a constant mass flow is specified according to the investigated operating point. At the outlet, the static pressure is specified and thus the cavitation number σ is set. For spatial discretization a high resolution scheme and for temporal discretization a second order backward Euler scheme is used. The SST model is applied as RANS turbulence model.

4. Numerical results

In the present study, an operating point at deep part load conditions, which is specified in table 1, is investigated. It is characterized by a discharge below 30% of the discharge at best efficiency point Q_{BEP}. At these operating conditions, the experiments observed that no vortex rope is present any longer and the flow is dominated by a huge stagnant region in the center of the draft tube cone [5].

Table 1. Specification of the investigated operating point

Q_{ED} (-)	Q/Q_{BEP} (-)	n_{ED} (-)	n (rpm)	σ (-)
0.055	0.275	0.317	880	0.09

First, an appropriate time step is selected. Single-phase simulations with the 11M mesh are performed for four different time steps. For the torque, the deviation to the measurement results is investigated. The results are listed in table 2. For huge time steps, the torque significantly depends on the time step size. Regarding the results for a time step of 0.26° and 1° shows only minor differences and thus indicate a time step independency. As a consequence, for the following investigations, a time step according to 1° of runner revolution is selected.

Table 2. Deviation of the torque to experiments depending on the time step

Time step size (° of runner revolution)	10.56	6.60	1.056	0.264
Torque – Deviation to experiment (%)	-3.9	-0.5	2.4	2.5

In the second step, a suitable treatment of the boundary condition at the outlet of the draft tube is chosen that takes into account the present flow regime with low discharge. Three different single-phase simulation setups are investigated. The outlet boundary type permits outflow and prohibits inflow while the opening boundary condition allows both, outflow and inflow. In the third setup, a tank which is connected to the draft tube outlet is added. It corresponds to the setup of the test rig. The simulation results for the three setups are presented in table 3.

A comparison of the torque shows similar results for all boundary treatments, whereas the evaluation of the head indicates differences. When assessing the head deviations, it has to be taken into account that the calculation for the setups outlet and opening might be affected by setting the outlet boundary condition just at this location. In terms of accuracy, a difference of 0.7% to the tank simulation seems to be acceptable as the accurate prediction of head is not a priority objective of this study. The analysis of the discharge in one channel of the draft tube diffusor indicates that the investigated operating point is characterized by strong backflow in one channel. This cannot be modeled by the outlet treatment. The results for the opening and tank treatment are in the same range. All in all, the setups opening and tank are suitable for further analyses. Due to the fact that the opening treatment needs approximately 20% less mesh elements, it is selected for the following analysis.

Table 3. Comparison of head, torque and discharge in one draft tube diffusor channel for three different treatments at the outlet boundary condition

Boundary treatment	Outlet	Opening	Tank
Torque – Deviation to experiment (%)	2.42	2.36	2.37
Head – Deviation to experiment (%)	-7.6	-6.4	-6.9
$Q_{DT,Channel1}/Q_{SC,In}$ (%)	0.1	-41.8	-39.1

The results of two-phase simulations with the 11M mesh indicate that the core of the inter blade vortices is not resolved properly, as no cavitation of the vortices occurs. Only a small cavity volume arises that is located at the trailing edge of the runner close to the hub. Thus, further investigations are carried out with the 26M and 35M mesh. Figure 2 displays the course of the vapor volume over time for different meshes. It is noticeable that in terms of vapor volume a mesh dependency is still present. The 35M mesh resolves the channel vortex better and thus the best resolution of the minimum pressure in the vortex core is obtained. Consequently, the biggest vapor volume is achieved there. Furthermore, it is conspicuous that the vapor volume is oscillating and the corresponding frequency is changing with the mesh refinement. The oscillation might be a result of differences in the discharge through the different guide vane channels. A comparison of head and torque shows a deviation of less than 0.1% between the 26M and 35M simulation. Thus, it can be stated that a mesh dependency is still present but only for resolving the vortex core. This is in good agreement with the investigations carried out by Stein et al. [9]. For entire mesh independence, a finer mesh is necessary which results in huge computational efforts.

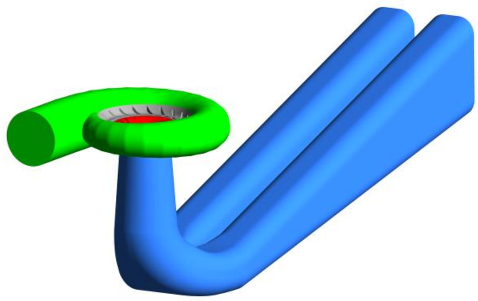

Figure 1. Simulation domain

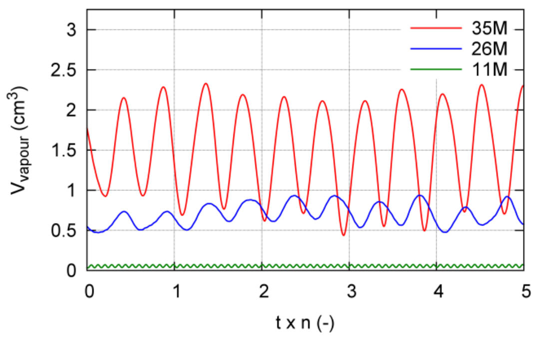

Figure 2. Vapor volume for different mesh sizes

Figure 3 shows a visualization of the inter blade vortices. The simulation results are displayed for the 35M mesh. In figure 3(a) the vortices are visualized with an isosurface of the velocity invariant. The coloring is related to the static pressure, blue being the lowest pressure. The simulation can reproduce the inter blade vortices which can also be visualized with an isosurface of the volume fraction of the liquid phase (see figure 3(b)). Both ways of visualization indicate that the vortices differ in strength in the different runner channels. In some runner channels the simulation does not even show cavitating vortices. High speed visualization recording of the experiments also shows differences in the cavity size in different runner channels (see figure 3(c)). However, the measurements indicate that the inter blade vortices are cavitating in every channel of the runner. As the simulated vapor volume is lower compared to the experiments, a deviation in the pressure level in the runner might be present between simulation and measurements. Further experimental and numerical analyses are required and will be carried out in the future for a better understanding of the processes at these extreme off-design conditions.

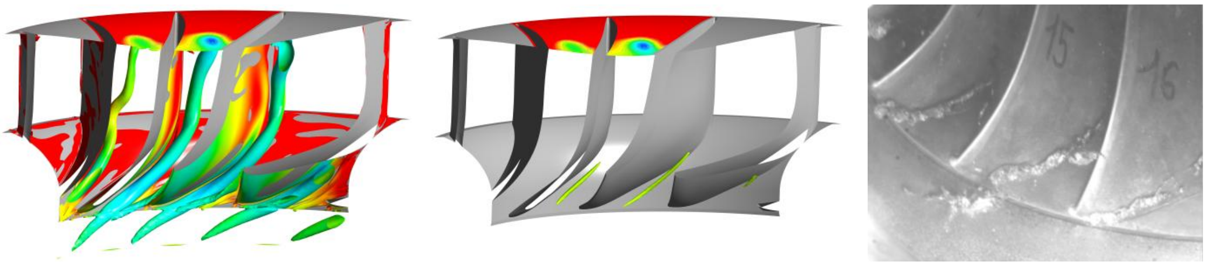

Figure 3. Visualization of the inter blade vortices – (a) Isosurface of the velocity invariant – (b) Isosurface of the volume fraction of the liquid phase – (c) Experiment [5]

5. Conclusion and outlook

In the current study, cavitation simulations are carried out on a Francis turbine at model scale for an operating point at deep part load. Preliminary investigations with single-phase simulations indicate that the time step size should be at most 1° of runner revolution and that the treatment of the boundary condition at the outlet has an impact on the flow characteristics.

The results of the two-phase simulations highlight the necessity of fine grids in the runner to resolve the vortex core of the inter blade vortices. An optically comparison with high speed visualization from experiments shows the existence of inter blade vortices in both, simulation and measurement. Nevertheless, the simulation seems to underestimate the occurrence of cavitation.

In future work, further investigations have to be carried out to better understand the formation of inter blade vortices. An additional measurement campaign will focus on the pressure on the runner blades. That will make it possible to compare the pressure level between simulation and measurement at a location close to the cavity region. Furthermore, existing pressure fluctuations can be analysed in detail. As a main objective, the simulation results will be used as input for a finite element analysis to predict the stresses in the runner. This enables an assessment of the structural integrity for operating turbines at deep part load conditions.

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 Keita Yamamoto, Andres Müller, Arthur Favrel, Christian Landry and François Avellan, for providing all experimental data enabling comparisons presented here.

References

[1] Avellan F 2004 *6th Int. Conf. on Hydraulic Machinery and Hydrodynamics* Timisoara
[2] Dörfler P, Sick M and Coutu A 2013 *Flow Induced Pulsation and Vibration in Hydroelectric Machinery* (London: Springer)
[3] Magnoli M V 2014 *Ph.D. Thesis Technische Universität München*
[4] Zuo Z, Liu S, Liu D, Qin D, Wu Y 2013 *Adv. Mech. Eng.* **2013** Article ID 397583
[5] Yamamoto K, Müller A, Favrel A, Landry C and Avellan F 2014 *IOP Conf. Series: Earth and Env. Sci.* **22** issue 3, p. 022011
[6] Koop A H 2008 *Ph.D. Thesis University of Twente*
[7] Bouziad Y A 2006 *Ph.D. Thesis EPFL Lausanne*
[8] Zwart P J, Gerber A G, Belamri T 2004 *Int. Conf. on Multiphase Flow* Yokohama paper no.152
[9] Stein P, Sick M, Dörfler P, White P, Braune A 2006 *Proc. 23rd IAHR Symp. on Hydraulic Machinery and Systems* Yokohama paper F228