CFD NUMERICAL SIMULATION OF CAVITATING TURBULENT FLOW IN A FRANCIS TURBINE

A. Laouari¹, A. Ghenaiet²

¹Laboratory of Energetics and Engineering, Faculty of Engineering, University of Boumerdes, Boumerdes 35000, Algeria. (AHMEDLAOUARI@GMAIL.COM)

²Laboratory of Energetics and Conversion Systems, Faculty of Mechanical Engineering, University of Sciences and Technology Houari Boumediene, BP32 El-Alia Bab-Ezzouar 16111, Algiers

ahmedlaouari@gmail.com

Abstract: As a numerical method to study the cavitation performance of a Francis turbine, the model for cavity/liquid two-phase flow is adopted in the cavitating turbulent flow analysis together with the k-ω SST turbulence model in the present paper. The cavitating flow in a complete Francis turbine is treated from the steady flow simulation since the feasibility of the cavitation model to the performance prediction of the turbine is the present major concern. First, the steady non-cavitating relative flow is computed in all passage of turbine. Second, the cavitation model is activated. Results for cavity shape and extent, as well as for the pressure distribution on the blade with and without cavity are presented and discussed.

Keywords: Francis Turbine, CFD, CAD, Internal Flow, Hydrodynamic performances, Cavitation

I. INTRODUCTION

Increase in demand for energy along with the necessity of reducing harmful environmental impact has given rise to renewed interest in extracting energy from clean sources like Hydroelectric power. Hydropower is an efficient method which can turn %90 of the available energy to electrical energy. It is cost effective as it has long power plant life: REN21 (2016). Hydroelectric power plants can be classified in several ways. According to the impact types, two main types of the hydraulic turbines are reaction and impulse type turbines. Examples of the impulse turbines are Pelton type, Turgo type and Michell-Ossberger type turbines. Examples of the reaction turbines are Francis. These turbines are applicable to a wide range of head and specific speed values. Their wide range of applicability and easier structural design make Francis turbines advantageous compared to other hydraulic turbines. Main components are spiral case, stay vane, guide vane, runner and draft tube. Spiral case keeps the velocity distribution uniform and converts pressure head into velocity head. Stationary vanes carry pressure loads in the spiral case and they provide the flow to reach the guide vanes without hydraulic losses. Guide vanes are the movable components of a Francis turbine. These are connected to the shafts, to the runner inlet and also control the flow, thus affect the overall the power output of the turbine: Raabe (1985), Dirtina (1996). The flow in a hydro-turbine runner is extremely complex, since it is generally turbulent, unsteady and highly three-dimensional in nature with strong effects from rotation and curvature.
Recently, with the rapid development of computer technology and advanced CFD, it has become routine to directly simulate internal turbulent flow in individual or multiple components of a turbomachine: Sabourin (1996). Examples include sophisticated large eddy simulations (LES) in a Francis turbine: Song (1996), and in a centrifugal pump impeller at design and off-design conditions: Byskov (2003). However, a robust and fully 3D inverse design approach, by which the required flow characteristics and parameters are specified as inputs and the corresponding blade geometry is computed and generated as output, is still not commonly performed: Goto (2002), Goto (2002).

On the other hand, the management of the large and small hydro power plants for achieving maximum efficiency with time is an important factor, but the plant components like turbine show the declining performance after a few years of operation as they get several damages due to many reasons like as erosion due to silt, cavitation, corrosion and fatigue. One of the significant reasons is cavitation. According to Bernoulli’s principles, an increase in velocity in a fluid is accompanied by a decrease in pressure. If at any point liquid flows into a region where the pressure is reduced to vapor pressure, the liquid boils and bubble formation takes place locally and when these bubbles reach to areas of higher pressure, they suddenly a collapse. This process is called cavitation. It produces high pressure pulses, when such collapse takes place adjacent to solid walls continually and at high frequency. The material in that area gets damaged due to pitting of solid surfaces. It causes the problem of noise, vibration in draft tubes and trailing edge of turbine blades and drop in efficiency: Logan (2004).

Leading edge cavitation, traveling bubble cavitation, von Karman vortex cavitation and draft tube swirl are the main forms of cavitation that can arise in Francis turbines. The flow structure in the hydro turbines is highly complex. Hence reaching an adequate level of precision in CFD simulations depends on the use of very fine meshes and complex turbulence models, which require enormous computing resources and large simulation times. Consequently, the unsteady CFD approaches would not be fast and robust enough for calculating global performance characteristics of hydro turbines in design mode. In contrast, the steady CFD is now able to predict hydro turbine characteristics in shorter computational time with adequate level of accuracy and less computational resources: Hosseinimanesh (2014). Vu (2002) used steady-state stage computations for accurate prediction of efficiency characteristics of a Francis turbine near its best efficiency point. He also showed steady-state simulations to be a highly effective methodology for comparing global draft tube performance for nearby design operating points: Vu (2010). However, to the authors knowledge, the capacity of steady state RANS to accurately assess turbine runaway speed over a range of operating conditions has not been evaluated. In recent years, cavitation in hydraulic machines has been extensively studied: Ji (2012), and Wei (2012). Most studies mainly focused on the numerical methods of cavitation prediction: Ji (2012), and performance improvement through the application of cavitating flow simulation: Luo (2008). These contributions have prompted the need for explanations regarding cavitation in hydraulic machines. Studies show that turbulent cavitating flow simulation predicts the cavitation development in flow passages, and the flow upstream of the impeller inlet is crucial for cavitation performance: Luo (2010).

This paper presents a 3D numerical simulation of turbulent flow in Francis turbine at different operating conditions, different rotations speed of runner and the openings of distributor were changed to study the performances of four openings using ANSYS
CFD software and compared with the experimental datum. In addition, the velocity and pressure distribution through the whole flow passage of the model Francis turbine was attained. The energy property of the model turbine is then predicted. Also, numerical simulations with cavitation model through ANSYS CFX were performed and compared with simulation without cavitation model to verify the design and the influence of cavitation model on behavior of flow at different operating conditions.

II. CAVITATION MODEL

The Rayleigh-Plesset model is implemented in the multiphase frame-work as an inter-phase mass transfer model in CFX code. For cavitating flow, typically the homogeneous multiphase model is used. The cavitation model based on the equation of Rayleigh-Plesset is applied to cavitation rate estimation. The Rayleigh-Plesset equation describing the growth of a gas bubble in a liquid is given by:

\[ R_B = \frac{d^2 R_B}{dt^2} + \frac{3}{2} \left( \frac{dR_B}{dt} \right)^2 + \frac{2\sigma}{\rho_f R_B} = \frac{P_v - P}{\rho_f} \]  

(1)

where \( R_B \) represents the bubble radius, \( \rho_f \) is the liquid density and \( S \) is the surface tension coefficient between the liquid and vapor. This is derived from a mechanical balance, assuming no thermal barriers to bubble growth. Neglecting the second order terms (which is appropriate for low oscillation frequencies) and the surface tension, the final expression has been derived assuming bubble growth (vaporization). It can be generalized to include condensation as follows in equation (2):

\[ m_{cav} = \begin{cases} 
-F_c \frac{3r_{nuc}(1-\alpha)\rho_v}{R_B} \sqrt{\frac{2}{3} \frac{P_v - P}{\rho_l}} & \text{if } P \leq P_v \\
F_c \frac{3\alpha\rho_v}{R_B} \sqrt{\frac{2}{3} \frac{P - P_v}{\rho_l}} & \text{if } P \geq P_v 
\end{cases} \]

(2)

where \( P_v \) is the liquid vapor pressure, \( \rho_{nuc} \) is a nucleation site volume fraction, \( R_B \) is the radius of a nucleation site, \( F \) is an empirical factor depend on condensation and vaporization designed for different rates. A detailed description of multiphase models and modeling of cavitation could be found in ANSYS CFX solver theory: ANSYS Inc (2010). The Rayleigh-Plesset cavitation model validation is used as presented by various researchers: Wang (1994) and Bakir (2004).

III. NUMERICAL SIMULATION METHOD

The computational region of Francis turbine consisted of three components: stationary domain 1 (casing, and 06 guide vanes); rotating domain (runner with 10 blades); stationary domain-2 (draft tube). The mesh of Francis turbine was provided in Gambit and turbo-Grid software meshing, the total number of mesh was 3.2 million nodes. The numerical simulations of Francis turbine at three operating points have achieved.
Table 1 shows the parameters of calculated operating points. Table 2 shows the solution parameters used for performing the numerical simulations.

Table 1, Parameters of calculated operating points

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Symbol</th>
<th>Part load</th>
<th>BEP</th>
<th>Full load</th>
</tr>
</thead>
<tbody>
<tr>
<td>Net head [m]</td>
<td>H</td>
<td>12</td>
<td>12</td>
<td>12</td>
</tr>
<tr>
<td>Flow rate [l/min]</td>
<td>Q</td>
<td>95</td>
<td>165</td>
<td>215</td>
</tr>
<tr>
<td>Runner velocity</td>
<td>rpm</td>
<td>1100</td>
<td>1900</td>
<td>2360</td>
</tr>
</tbody>
</table>

Table 2, Boundary physics and solution parameters used in the numerical simulations

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Analysis type</td>
<td>Steady State</td>
</tr>
<tr>
<td>Interfaces</td>
<td>Frozen Rotor; discretization type-GGI and Stage</td>
</tr>
<tr>
<td>Fluid</td>
<td>Water properties updated with actual density, viscosity</td>
</tr>
<tr>
<td>Boundary conditions</td>
<td>Mass flow inlet and atmospheric static pressure outlet with Reference pressure: 0 kPa</td>
</tr>
<tr>
<td>Discretization and solution controls</td>
<td>Advection scheme: high resolution and Turbulence numeric: high resolution</td>
</tr>
<tr>
<td>Turbulence model</td>
<td>k-ω SST</td>
</tr>
<tr>
<td>Convergence control</td>
<td>RMS of pressure, mass-momentum, and turbulent parameters &lt;10E-6</td>
</tr>
</tbody>
</table>

IV. RESULTS AND DISCUSSION

IV.1 Performances prediction and analysis

3D numerical simulations of turbulent steady flow through the complete Francis turbine at different operating conditions, in terms of variations of flow rate four distributor openings and rotational speed of runner. The first step present the results of global performance parameters that allowed to evaluate the characteristics of the turbine in addition to various factors affecting the performance at partial load, optimal, and over load conditions. In the second presents a description with visualization of internal flows through the components of this turbine. Figure 1 presents a comparison between CFD and experimental results for the curve of power-volume flow. It can be readily observed that for optimum loading the accuracy of the CFD predictions is very satisfactory, and even for moderately partial loads or over volume flow, the CFD results are close to experimental results.
The evolution of the produced power by the runner of Francis turbine at different rotational speed and flow rate was illustrated in Figure 2. For all velocities the produced power increases with the volume flow rate but at different scales. For velocity of 2360 rpm, the power sweeping the whole operation range is 470 W. As against, the rotational speed 1100 rpm the gain is 124 W. This comparison gives us an idea on the partial flow functioning which is generally the case.

Figure 1. Comparison of numerical simulation and experimental results

Figure 2. Produced power of numerical simulation

Figure 3. Efficiency hill chart

Figure 3 present the efficiency at different rotational velocity and volume flow rate. As seen, the rotational velocity 1900 rpm and flow rate of 165 l/min are considered as the optimum operating point for this turbine, because the efficiency is reaching a maximum of 79.18%. The specific speed \( \omega_s \) is a dimensionless parameter defined with coefficients of energy \( \psi \) and discharge \( \Phi \) which do not depend on the runner outlet diameter \( D \).
On the optimal operating point, the corresponding to the best efficiency operating point, specific speed is a classification criterion for turbomachinery as we can see in Figure 4 that shows a hill chart of Head depending on the specific speed at four openings of distributor for a constant rotational speed 1900 rpm that validated by the theoretical of the Francis turbines that have a specific speed $\omega_s$ between 0.14 and 0.65: Tridona (2010).

![Hill Chart of Head vs Specific Speed](image)

Figure 4. Head depending on the specific speed

\[
\Phi = \frac{8Q}{(\pi D^2 \omega)} \tag{3}
\]

\[
\psi = \frac{8gH}{(D^2 \omega^3)} \tag{4}
\]

\[
\omega_s = \frac{\Phi^{1/2}}{\psi^{3/4}} = \frac{\omega \sqrt{\frac{\pi}{2gH}}}{(2gH)^{3/4}} \tag{5}
\]

Streamlines flows for part load, optimal load, and over load are shown in figure 5. Its indicating maximum turbulence in the runner and draft tube for all three regimes, which is converted into head loss. Runner is the major component of turbine for energy conversion, therfore runner part plays critical role for deciding the efficiency of turbine. This figure describe velocity profil inside the turbine assembly indicates that runner and draft tube domain has smooth velocity profile whereas as soon as water enters draft tube domain velocity starts decreasing and profile becomes non uniform. At part load, the 3D flow streamlines along the wall of the draft tube are streamlined and uniform but the flow field in inner region is disturbed with a very low velocity creating a dead water zone, resulting flow recirculation and genesis of vortex shedding.

Operation of hydraulic turbines in some off-design conditions exhibits local pressure pulsation caused by rotor-stator interaction and draft tube vortex precession that propagate along the whole water conduit: Chirkov (2012).
Figure 5. 3D velocity streamlines pattern across whole domain

Figure 6 shows the pressure distribution in the wall of the turbine at different loads. Much of the turbine’s performance depends on the flow behavior in the draft tube and the flow condition in this component depends on point of operation.

The flow accelerates in the spiral casing as it glides through the vanes and enters the runner. After the runner transfers the energy into torque by the pressure difference between its blades, the flow decelerates while moving toward the draft tube outlet (Figure 6).

Figure 6. Velocity contours at mid-section
The swirl rate appearing behind the runner at part-load and over load greatly affects the flow condition in the draft tube. Swirl appears at runner downstream due to the radial component of velocity at runner exit where flow is supposed to be purely axial. The incorporated elbow type draft tube with a single outflow channel has a conical diffuser prior to the elbow followed by a cone diffuser toward the outlet.

Velocity gradient in the cone diffuser shows that most of the excess kinetic energy of outlet stream is converted into static pressure thereby delimiting the outlet-bound velocity to minimal. Also, a symmetrical flow distribution in the entire draft tube, free of flow separation, can be observed at full load where the streamlines are uniform and evenly distributed.

The 3D static pressure distribution in Figure 7 is in coherence with the uniform flow that is distributed evenly around the turbine. For all three loads, the static pressure decrease from the inlet of spiral case into the outlet of draft tube.

With the static pressure at the exit of runner being less than the atmospheric pressure, water fills up all the passages of the runner blades. As water glides along the blades of runner, it produces a pressure difference between the either sides of the blade to produce torque and imparts a rotational motion to the turbine. This explains to why the pressure gradient in the band is not gradually decreasing unlike in the blade surface. This phenomenon facilitates to improve the performance of the turbine, where in the runner and draft tube prevents the cavitation inception and the erosion.

![3D Static Pressure Distribution](image)

**Figure 7. 3D-Static pressure**

**IV.2 Numerical simulation with cavitation model**

In the Cavitating flow simulation in a Francis turbine, the following assumptions were used by the work of Shuhong (2008).

Bubble radius: $0.5 \times 10^{-6}$ m
Isothermal temperature: 198 K  
Nuclei volume fraction: $0.5 \times 10^{-5}$

Table 3 shows the produced power prediction by a runner of Francis turbine for simulations with cavitation model and without it at three different operating conditions.

We observed that a value of power for two models at optimal and full load is almost the same. That explain a cavitation number is sufficiently accepted. By against, at part load the difference between values of power is noticeable and a cavitation can had can take place in this conditions.

Table 3. Prediction of produced power

<table>
<thead>
<tr>
<th>Operating conditions</th>
<th>Without cavitation model</th>
<th>With cavitation model</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Produced power (W)</td>
<td>Produced power (W)</td>
</tr>
<tr>
<td>Part load</td>
<td>98.315</td>
<td>94.654</td>
</tr>
<tr>
<td>Optimal load</td>
<td>189.963</td>
<td>189.971</td>
</tr>
<tr>
<td>Full load</td>
<td>434.114</td>
<td>434.130</td>
</tr>
</tbody>
</table>

Figures. 8 and 9 shows a comparison between the volume fraction of water vapor distribution and static pressure distribution for 3D runner blades. It is clearly seen that the volume fraction of water vapor at the top of the blade on the front and bottom as well as the most of the blade on the back is higher than others.

Figure 8. Water vapour volume fraction at part load with cavitation model  
Figure 9. Static pressure distribution for runner blades at part load

Compared with the pressure distribution, the maximum of volume fraction of water vapor is not at same location with the maximum of static pressure, but at the location away from there for some distance along the flow direction. This is the characteristics of the cavitation model, indicating that cavitation is not instantaneous as described by Zhang (2012). It also reflects the mechanism of cavitating flow. If the flow from the runner has strong swirl, the cavitated vortex rope is observed in the draft tube at part load. Using the steady flow analysis, the rope at part load is investigated. To see the configuration of vortex rope in the inlet cone of the draft tube, it is visualized from the numerical results.
V. CONCLUSION

The paper presents a numerical investigation of the 3D cavitating flow in a Francis turbine, using the homogeneous two-phase flow model. After a single phase (liquid) relative steady flow is computed in the turbine, the cavitation model is switched on. Numerical results presented here show the ability of the cavitation model, implemented in ANSYS CFX code to correctly reproduce the cavity location and extent. The analysis of the pressure field without and with cavitation using model of Rayleigh-Plesset to simulate two phase flow through Francis turbine to predict and analyse the cavitating turbulent flow and the vortex rope in draft tube clearly demonstrates that for advanced cavitation stages one needs to compute a two-phase flow.

VI. REFERENCES

18. ANSYS Inc. ANSYS CFX-Solver theory guide. 2010.