Numerical simulation of air injection in Francis turbine

D Chirkov¹, P Scherbakov¹, V Skorospelov², S Cherny¹, A Zakharov³

¹Institute of Computational Technologies SB RAS, Novosibirsk, Russia
²Sobolev Institute of Mathematics SB RAS, Novosibirsk, Russia
³OJSC "Power Machines" LMZ, St-Petersburg, Russia

Abstract. Operation of Francis turbines in part load and sometimes in full load operating points is associated with increased pressure pulsations. One of the practical ways to reduce these pulsations is to add atmospheric air into the flow. It was shown previously that air injection/admission through the center of the runner cone can significantly reduce the amplitude of pressure pulsations both in part load and in full load operating points. Up to now the effect of flow aeration has been investigated mainly experimentally. In the present paper we performed CFD simulations of this phenomenon. Computations have been carried out in frames of homogeneous three-phase “liquid – vapor – non-condensable gas” mixture model. Air phase has been assumed incompressible. Both part load and full load operating points have been considered with different air flow rates. Computations have shown that even at small flow rate the air changes the structure of the swirling flow downstream the runner and considerably reduces pressure pulsations, caused by vortex rope rotation. The obtained results are in agreement with corresponding experimental data for model turbine.

1. Introduction

It is well known, that operation of Francis turbines in part load and sometimes in full load operating points is associated with increased level of pressure pulsations. One of the most effective practical solution to reduce these pulsations consists in addition of atmospheric air into the turbine flow passage [1]. Different methods of aeration have been suggested in the literature, including aeration through the runner shaft, through blade trailing edges, draft tube (DT) peripheral aeration, and aeration in the vaneless space between the guide vanes and the runner [2, 3]. Air is either injected into the flow passage by a compressor (forced air injection) or sucked naturally through a special valve when the pressure in the flow drops below the specified level (natural air admission). Aeration through the center of the runner shaft proved to be the most effective to suppress the vortex rope pulsations in part load [4]. In [1, 5] it was shown that air injection/admission can also reduce power oscillations and unit vibrations in high load operating points.

Injected air affects the pressure pulsations in different ways. First, the airflow changes pressure and velocity field, increasing the pressure and thus reducing cavitation in the center of draft tube cone. Second, it alters the cavity compliance \( C = -\partial V_c / \partial H \) of the gaseous cavity of volume \( V_c \) including water vapor and air. It should be noted, that \( C \) is one of the key parameters determining the stability and the frequency of the natural oscillations of the power plant [6].

Despite recent progress in development of incompressible and multiphase CFD methods, as well as increased computer power, there are only few papers devoted to numerical simulation of air injection...
in hydraulic turbines [7, 8, 9]. These papers adopted different mathematical models for simulation of the air transport. However all these papers considered 2-phase “water-air” mixture and did not take into account cavitation. It is known that cavitation plays an important role in the development of pressure pulsations, especially in full load operating points.

In [10, 11] for the first time we tried to simulate the effect of air injection on self-excited oscillations in full load. For that we considered a 3-phase “water-vapor-air” mixture model for the turbine domain and attached a 1D hydro-acoustic model of the penstock, as suggested in [12]. Computations were performed for a prototype turbine with gravity acceleration taken into account. It was shown that even small air flow rate of about 0.4% of water flow rate completely eliminates cavitation in the DT, the source of self-excited oscillations, and thus stabilizes the turbine operation. These results are briefly shown in figure 1.

Figure 1. The effect of air injection on full load operation of prototype turbine.

(a) vapor volume fraction with no air injection, (b) axial velocity with air injection \(Q_{air}=0.4\%\),
(c) efficiency and pressure pulsation as function of air flow rate.

In the present paper the above 3-phase model is applied for simulation of forced air injection in a scale model turbine with specific speed \(n_q=73.6\), figure 2. The computed results are compared to experimental measurements, performed in the Laboratory of Hydraulic Turbines, “Power Machines” LMZ (St-Petersburg).

2. Governing equations

Cavitating flow in turbine flow passage is described as isothermal homogeneous three-phase mixture, consisting of water, vapour, and non-condensable gas (air). It is assumed that all the phases share common pressure and velocity fields. Distribution of liquid volume fraction \(\alpha_L\) is governed by transport equation with source terms, responsible for evaporation and condensation. The convection of a gas volume fraction \(\alpha_G\) is governed by the same transport equation with zero source term [13]:

\[
\frac{\partial \rho}{\partial t} + \text{div}(\rho \mathbf{v}) = 0, \quad (1)
\]

\[
\frac{\partial \rho \mathbf{v}}{\partial t} + \text{div}(\rho \mathbf{v} \otimes \mathbf{v}) + \nabla \mathbf{p} = \text{div}(\mathbf{\tau}) + \rho \mathbf{f}, \quad (2)
\]

\[
\frac{\partial \alpha_L}{\partial t} + \text{div}(\alpha_L \mathbf{v}) = \frac{1}{\rho_L}(m^+ + m^-). \quad (3)
\]

\[
\frac{\partial \alpha_G}{\partial t} + \text{div}(\alpha_G \mathbf{v}) = 0. \quad (4)
\]

Here \(\rho = \alpha_L \rho_L + \alpha_G \rho_G + (1 - \alpha_L - \alpha_G) \rho\) is the mixture density; \(\mathbf{v}\) is the velocity vector; \(\mathbf{p} = p + 2\rho k/3\); \(p\) is the static pressure; \(k\) is the turbulence kinetic energy. Absolute reference frame is used for static components of turbine flow passage, while rotating reference frame is used for the
runner, rotating with angular velocity \( \omega \) around the \( Oz \) axis. Thus for runner sub-domain \( f = (x_1 \omega^2 + 2u_1 \omega, x_2 \omega^2 - 2u_1 \omega, g) \). Gravity acceleration \( g \) is necessary to account the buoyancy of the air phase. In (2) \( \tau \) is the tensor of viscous stresses. Dynamic viscosity of the three phase mixture is computed as
\[
\mu = \alpha_L \mu_L + \alpha_G \mu_G + (1 - \alpha_L - \alpha_G) \mu_v,
\]
where \( \mu_L, \mu_G, \mu_v \) are the dynamic viscosity coefficients of liquid, gas (air) and vapour, respectively. Kim-Chen \( k-\varepsilon \) [14] and SST \( k-\omega \) turbulence models with log-law wall functions were used to evaluate \( k \) and turbulent viscosity \( \mu_T \) needed to close the mean flow equations (1)-(4).

Condensation (\( m^+ \)) and evaporation (\( m^- \)) terms in (3) are evaluated using the model of Zwart, Gerber, and Belamri [15]:
\[
m^+ = C_{prod} (1 - \alpha_L - \alpha_G) \rho_v \sqrt{\frac{2}{3} \max[0, p - p_v]}, \quad m^- = -C_{dest} \alpha_L \rho_v \sqrt{\frac{2}{3} \max[0, p_l - p]},
\]
where \( C_{prod} = 3 \cdot 10^4 \), \( C_{dest} = 7.5 \cdot 10^4 \).

According to (4) air is treated as incompressible fluid with constant density \( \rho_0 < \rho_L \). It should be noted that since all phases share the same velocity, the present model is unable to describe the effect of separation of premixed phases.

Boundary conditions are total specific flow energy fixed in the inlet cross section and total specific flow energy kept in the DT outlet. With these inlet/outlet conditions discharge \( Q \) is not known a priori and is found iteratively in the process of computation.

In case of air injection, constant volumetric air flow rate is specified in the DT inlet section, in cells where \( R < R_v \), see figure 3.

3. Numerical method

Governing equations of the above three-phase model are solved numerically using our in-house developed CADRUN solver, based on finite volume artificial compressibility approach. Dual time stepping is used for unsteady calculations. In pseudotime equations are marched using implicit finite volume scheme. Third order accurate MUSCL scheme is used for discretization of inviscid fluxes through cell faces. In order to prevent numerical oscillations, artificial dissipation is added on liquid-gas interfaces, as suggested in [13]. Second order backward scheme is applied for physical time derivatives. Linearized system of discrete equations is solved using LU-SGS iterations. All equations (1)-(4) are solved for \( (p, v, \alpha_L, \alpha_G) \) in a coupled manner. Details of the solver can be found in [11].

Periodic stage approach is used for turbine flow analysis, requiring computations only in one wicket gate (WG) channel, one runner channel, and the whole draft tube. Mixing plane boundary condition is applied on “WG – runner” and “runner – DT” interfaces with circumferential averaging of flow variables \( (p, Cr, Cu, Cz, \alpha_L, \alpha_G, k, \varepsilon) \), where \( Cr, Cu, Cz \) are the radial, circumferential and axial components of the velocity vector, respectively.

4. Simulation results

Computations were performed for scale model turbine (runner diameter \( D_1 = 0.46 \text{m} \)), operated at \( H = 21.077 \text{m} \), unit speed \( n_1 = 71.32 \). Computational domain is shown in figure 3. It consists of one guide vane channel, one runner channel, and the whole draft tube. Basic block structured mesh consists of 260'000 cells, see figure 3. The value of \( y^+ \) on the solid walls is in the range 15 to 230. The time step \( \Delta t \) is taken equivalent to 7.5 degrees of runner rotation. Total flow energy in the draft tube outlet is set corresponding to Thoma number of the scale model tests.
4.1. Part load operating point

Part load operating point corresponds to guide vane opening (GVO) $a_0=20$mm. First, two-phase cavitation liquid-vapor computations were performed with no air injection. Figure 4 shows pressure pulsations in the DT points D3 and D4, obtained using Kim-Chen turbulence model. It can be seen that the peak-to-peak pressure pulsation in point D4 is higher than that in D3. Figure 5 shows that both Kim-Chen and SST turbulence models gave similar frequency and amplitude of pressure pulsations, that are in good agreement with experimental data. However in case of SST model no cavitation was observed in the core of the vortex rope, figure 6.

Then a series of computations was carried out for different volumetric air flow rates, ranging from $Q_{air}=0.25\%$ to $Q_{air}=2\%$ of nominal turbine discharge. Figure 7 compares frequency and amplitude of pressure pulsations in point D3, obtained in computation and experiment for the case $Q_{air}=0.5\%$ (see also figure 9 for peak-to-peak values). Experimental data clearly indicate that air injection with $Q_{air}=0.5\%$ reduces the amplitude of pressure pulsations. At the same time the frequency of pulsations $f/f_n$ shifts from 0.22 (no air) to approximately 0.3 in case of air injection. As for numerical simulation, the result, obtained in the basic mesh, depends on the turbulence model used. In case of Kim-Chen model, the helical vortex rope almost completely disappeared. In case of SST simulation with air injection the vortex rope remained. However, the amplitude and frequency of the pulsations did not alter significantly. Figure 8 shows that air flow rate has a small influence on pressure pulsations. Figure 9, left shows the aerated vortex rope, obtained using SST model, visualized as iso-surface of the air volume fraction $\alpha_g=0.1$. Figure 9, right shows the evolution of air flow rate in the DT inlet and elbow outlet cross sections. The frequency of air flow rate pulsations corresponds to vortex rotation frequency.

Figure 4. Pressure pulsations in points D3 and D4. No air injection. Kim-Chen model.
Figure 5. Spectrum of pressure pulsations in point D3. No air injection.

Figure 6. Vortex rope in the DT. No air injection.

Figure 7. Spectrum of pressure pulsations in point D3. Air flow rate $Q_{\text{air}}=0.5\%$. 
Figure 8. The influence of air flow rate on pressure pulsation in point D3. SST model.

Figure 9. Air flow rate $Q_{air}=0.5\%$. SST turbulence model. Left: iso-surface of air volume fraction $\alpha_a = 0.1$. Right: air flow rate in DT inlet and elbow outlet cross sections.

Figure 10. Peak-to-peak pressure pulsations in point D3 as function of GVO.

4.2. Full load operating point
Figure 11 shows the computed pressure pulsations in the DT (point D3) for full load operating point (GVO $a_0=48$mm). Both “no air” and “air injection” cases are shown. It can be seen that Kim-Chen and SST models give similar results for both cases. In case of air injection the amplitude of pressure pulsations decreased more than twice in agreement with experimental data, see figure 10. The reason of this reduction is the elimination of cavitation in the DT with aeration. In case of no aeration a cavitation cavity is observed in the centre of the drat tube just below the runner (figure 12, left). This cavity is proved to be the source of synchronous pressure pulsations of frequency $0.08f_n$, shown in figure 11 (left), by analogy with self-excited oscillations in prototype turbines, investigated in [11,12]. In case of air injection the vapour cavity in the DT is completely eliminated. Figure 12, right shows the instantaneous vortex-like distribution of the air volume fraction in the DT.
No air

\[ Q_{\text{air}} = 0.5\% \]

**Figure 11.** Computed pressure pulsations in the DT.

No air

\[ Q_{\text{air}} = 0.5\% \]

**Figure 12.** Gaseous cavities in the DT. Full load operating point, SST model. Left: no air, vapor cavity is shown as iso-surface \( \alpha_v = 0.5 \). Right: air injection, air cavity is shown as iso-surface of \( \alpha_g = 0.1 \).

5. Discussion and conclusion  
Three phase “liquid – vapor – non-condensable gas” RANS model is used for numerical simulation of air injection in model Francis turbine. A homogeneous mixture model is used meaning that all phases share the same velocity and pressure field. The air phase is assumed incompressible, and distribution of its volume fraction is governed by a simple transport equation. Kim-Chen \( k-\varepsilon \) and SST \( k-\omega \) models are used for turbulence. The model is applied for simulation of turbine aeration through the central hole in the runner shaft. Both part load and full load operating points are considered. Draft tube pressure pulsations obtained in “water – vapor” and “water – vapor – air” simulations are compared to experimental data.

Computations show that air injection at flow-rate \( Q_{\text{air}} = 0.5\% \) effectively eliminates cavitation in the draft tube, both in part load and full load operating points. In full load operating point air injection significantly reduced pressure pulsation (more than twice), in good agreement with experimental data.

In part load the situation is more difficult. For the case of SST model the air injection does not take a significant effect on the frequency and amplitude of pressure pulsations. Moreover, the amplitude is even a little bit larger than for no-aeration case, and does not depend on the air flow rate, in contrary to experimental data. Similar result was obtained in [7]. The case of Kim-Chen turbulence model needs further investigation, since in the present computations air injection suppressed the formation of a rotating helical vortex rope, and thus eliminated the associated pressure pulsations.
Cavitation in the draft tube can be the source of pressure pulsations (as in the case of full load instability). In this case aeration through the runner cone removes cavitation, and as the result, reduces the pressure pulsations. This effect is well captured by the present 3-phase model.

In part load pressure pulsations are caused by the rotation of the helical vortex rope. Usually the amplitude of these pressure pulsations does not depend significantly on Thoma number. From the other hand, flow aeration considerably reduces the pulsations. Therefore it seems that aeration affects the flow-field in a more complex way than simply to prevent cavitation in the vortex core. Evidently, this mechanism is still not captured by the present 3-phase numerical model. Accurate simulation of air injection in part load operating point requires some issues that should be refined.

First, it should be noted that the absolute value of pressure pulsations both with and without air injection is lower than experimentally measured. One of the reasons is the lack of grid resolution in the draft tube. The other reason is the use of periodic stage approach for the runner, requiring circumferential averaging of flow parameters on the mixing plane in draft tube inlet section. Ideally, full 360° runner domain should be used for vortex rope computations. Then, in the present computations the hydro-acoustics of the water conduit of the test rig was not taken into account. In order to check its influence a 1D-3D computation should be carried out, similar to [12]. Another important improvement is to account the compressibility of the air phase. These are the topics of ongoing research.

Nomenclature

| Symbol | Description |
|--------|-------------|
| \(a_0\) | Model guide vane opening [mm] |
| \(C\) | Cavity compliance \([m^2]\) |
| \(E\) | Total specific energy \([m]\) |
| \(g\) | Gravity acceleration \([m/s^2]\) |
| \(H\) | Net head \([m]\) |
| \(p_V\) | Vapor pressure \([Pa]\) |
| \(Q\) | Discharge \([m^3/s]\) |
| \(V_c\) | Cavity volume \([m^3]\) |
| \(\alpha_L\) | Liquid volume fraction \([-]\) |
| \(\alpha_G\) | Air volume fraction \([-]\) |
| \(\rho\) | Mixture density \([kg/m^3]\) |
| \(\rho_L\) | Water density \([kg/m^3]\) |
| \(\rho_V\) | Vapor density \([kg/m^3]\) |
| \(\rho_G\) | Air density \([kg/m^3]\) |

References

[1] Dörfler P, Sick M and Coutu A 2013 Flow-induced pulsation and vibration in hydroelectric machinery (London: Springer-Verlag)
[2] Dörfler P 1986 Design criteria for air admission systems in Francis turbines Proc. of IAHR Symp. on Hydraulic Machinery and Systems (Montreal, Canada)
[3] March P 2011 Hydraulic and environmental performance of aerating turbine technologies Proc. of EPRI/DOE Conference on Environmentally-Enhanced Hydropower Turbines (Palo Alto, California)
[4] Papillon B, Kirjczyk J and Sabourin M 2000 Atmospheric air admission in hydro turbines Proc. of Hydrovision paper 3C
[5] Türkmenoglu V 2013 The vortex effect of Francis turbine in electric power generation Turkish Journal of Electrical Engineering and Computer Sciences 21 pp 26–37
[6] Nicolet C 2007 Hydroacoustic modelling and numerical simulation of unsteady operation of hydroelectric systems EPFL Ph. D. Thesis No. 3751.
[7] Yu A, Luo X and Ji B 2015 Numerical simulation and analysis of the internal flow in a Francis turbine with air admission IOP Conference Series: Materials Science and Engineering 72 042047
[8] Dekterev A A and Dekterev D A 2016 Computational investigation of the effect of air injection into the fluid flow with precessing vortex core Journal of Siberian Federal University: Engineering & Technologies 9 pp 1110–1119 (in Russian)
[9] Pitorac L-I 2017 Numerical simulation of air injection in Francis turbines / NTNU Master thesis
[10] Chirkov D, Scherbakov P, Cherny S, Zakharov A, Skorospelov V and Turuk P 2016 Mitigation of self-excited oscillations at full load: CFD analysis of air admission and effects of runner
design Proc. of 28th IAHR Symposium on Hydraulic Machinery and Systems (Grenoble, France)

[11] Chirkov D V, Shcherbakov P K, Cherny S G, Skorospelov V A and Turuk P A 2017 Numerical investigation of the air injection effect on the cavitating flow in Francis hydro turbine Thermophysics and Aeromechanics 24 pp 691–703

[12] 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 turbine at off-design conditions Proc. of IAHR Symp. on Hydraulic Machinery and Systems (Beijing, China)

[13] Kunz R F, Boger D A, Stinebring D A et al 2000 A preconditioned Navier-Stokes method for two-phase flows with application to cavitation prediction Computers & Fluids 29 pp 849–75

[14] Chen Y S and Kim S W 1987 Computation of turbulent flows using an extended k-ε turbulence closure model. NASA CR-179204.

[15] Zwart P J, Gerber A G and Belamri T A 2004 Two-phase flow model for predicting cavitation dynamics Proc. of ICMF 2004 International Conference on Multiphase Flow (Yokohama, Japan) Paper No. 152