Numerical simulation of air injection in part load operating point

P Shcherbakov¹, D Chirkov¹, V Skorospelov², P Turuk³.

¹Institute of Computational Technologies SB RAS, Novosibirsk, Russia
²Sobolev Institute of Mathematics SB RAS, Novosibirsk, Russia
³pkshcherbakov@gmail.com

Abstract. The operation hydro turbines in the off-design operating points is often accompanied by pressure pulsations with a large amplitude. One way to deal with such pulsations is to inject the air into hydro turbine flow passage. The aim of this work is to simulate this process. Previously we carried out calculations in frames of homogeneous incompressible three-phase «liquid – vapor – non-condensable gas» mixture model. The computational domain consisted of wicket gate, runner and draft tube. The wicket gate and runner were considered in the periodic stage approach. The results obtained were in good agreement with experimental data at full load operating point. It was shown that the calculations on the previous model failed to accurately simulate influence of air flow rate on the amplitude and frequency of pressure pulsations in part load. Therefore the part load regime is studied carefully in this work: different computational domains are considered, different solvers are used, including ANSYS CFX and CADRUN, developed at the Institute of Computational Technologies. Different models of «liquid – gas» mixture are used in the calculations, compressibility is also taken into account. The flow rate of the injected air ranged from 0.5 to 2 percent of the water flow rate. The influence of the injected air on the flow pattern, including the vortex rope, is investigated. The results obtained are compared with experimental data.

1. Introduction

Francis turbines are working at wide range of operating points: in the part load, high-load and optimum. We are focusing on the part load operating point in this work. Pressure pulsations occurring in this operating point caused by the rotation of the vortex rope negatively affect the efficiency and turbine construction. The most common way to reduce this phenomenon on site is to inject air into hydro turbine [1]. The numerical investigation devoted to air injection in hydraulic turbines is researched in the literature poorly [2]. The simulation of the process is complex because of unsteadiness and ratio of water density to air density. We have already considered this problem in [4]. It was shown that air injection eliminates cavitation from the flow field in part load and full load operating points [6]. In that earlier studies we used our own solver CADRUN in order to simulate air injection through the runner shaft. The aim of this paper is to simulate air injection using different solvers and different numerical settings. We used ANSYS CFX and CADRUN.

The data obtained with usage of different solvers are compared to experimental measurements.
2. Governing equations
The «liquid – gas» mixture is assumed to be homogenous, i.e. all phases have the same pressure and velocity fields. The flow in the turbine is described by the following equations:

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

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

\[
\frac{\partial \alpha_G}{\partial t} + \text{div}(\alpha_G \mathbf{v}) = 0. \quad (3)
\]

Here \( \rho = (1 - \alpha_G) \rho_L + \alpha_G \rho_G \) is the mixture density; \( \alpha_L \) is water volume fraction; \( \alpha_G \) is the gas (air) volume fraction; \( \mathbf{v} \) is the velocity vector; \( \hat{p} = p + 2\rho k/3 \); \( p \) is the static pressure; \( k \) is the turbulence kinetic energy. The densities of water \( \rho_L \) and gas \( \rho_G \) are assumed constant. Dynamic viscosity of mixture is computed as

\[
\mu = (1 - \alpha_G) \mu_L + \alpha_G \mu_G,
\]

where \( \mu_L, \mu_G \) are the dynamic viscosity coefficients of liquid and gas (air) respectively. The SST \( k-\omega \) turbulence model with log-law wall functions was used in CADRUN to close the mean flow equations (1)-(3). The SST \( k-\omega \) and SAS SST turbulence models were used in ANSYS CFX with the same purpose.

Since we considered only part load operating point, the cavitation effect was neglected. Air was treated as incompressible fluid with constant density \( \rho_G = 10 \text{ kg m}^{-3} \). It should be noted that use or real air density casus stability problems both in CADRUN and ANSYS CFX.

3. Numerical method
In CADRUN solver governing equations of the model are solved using finite volume artificial compressibility approach. Dual time stepping was used for unsteady calculations. In pseudotime equations were marched by means of implicit finite volume scheme. Third order accurate MUSCL scheme was used for discretization of inviscid fluxes through cell faces. In order to prevent numerical oscillations, artificial dissipation was added on liquid-gas interfaces as suggested in [7]. Second order backward scheme was applied for physical time derivatives. Linearized system of discrete equations was solved by means of LU-SGS iterations. All equations (1)-(3) were solved for \((p, \mathbf{v}, \alpha_G)\) in a coupled manner. Details are described in [4].

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

We used the finite volume SIMPLEC numerical method in ANSYS CFX. The high resolution advection scheme for the mean flow equations and for turbulent quantities and 2nd order backward Euler transient scheme have been applied to perform computations.

4. Boundary conditions and computational technique
We considered two different computational domains in this work. The first computational domain includes one wicket gate channel, one runner channel and whole draft tube (Fig.1). Another computational domain includes draft tube only (Fig.2).
The number of cells in wicket gate channel is 48’000, 41’000 in runner channel and 295’000 in draft tube. The total number of cells is 381’000. The flow into the draft tube was of greatest interest, therefore the mesh is refined there.

The air was injected into draft tube through the small inlet area below the runner shaft (Fig.3) with constant velocity \( w \). The value of \( w \) is defined by the air flow rate \( Q_{air} \).

Computations were performed for the scale model turbine operating at head \( H=21.077\)m and unit speed \( n_{ui}=71.32 \). The guide wane opening was \( a_{w}=20\)mm. The time step was taken equivalent to 7.5 degrees of runner rotation. Total flow energy in the draft tube outlet was set corresponding to Thoma number of the scale model tests.

It was shown in [6] that air injection effectively eliminates cavitation. Therefore, we could assume that the mixture consists of water and air only and does not include vapor. This assumption simplified the model and reduced the computational time.

At first, the steady state computations were performed using both CADRUN and ANSYS CFX. The model was single-phased consisting of water only. We used the obtained flow field as initial values for transient simulations. Then we performed single-phase transient simulations with no air injection. Then
we started to inject air into draft tube. It should be noted that initial field for the transient simulations of air injection were taken from the transient computation with the use of single-phase model. The results obtained are discussed in the next section.

5. Simulation results

5.1. Simulation results without air injection

The computations in CADRUN were performed on mesh 1, and mesh 2 was taken for ANSYS CFX. The pressure pulsations obtained in computations are showed in Fig.4. It can be seen that the frequency of the pulsations in all cases is identical. The amplitude of the pulsations in ANSYS is slightly higher than in CADRUN. The largest amplitude of pulsations is observed in CFX calculation with the SAS SST model. Comparison of the computed data with experimental data is shown in Fig. 5. It can be noted that the experimental amplitude of pulsations is greater than amplitude obtained in computations.

![Figure 4. Pressure pulsations without air injection.](image)

![Figure 5. Amplitude of pressure pulsations without air injection. Comparison with experimental data.](image)
5.2. Simulation results with air injection

For further calculations, the SAS SST turbulence model was used in ANSYS CFX, as it showed better agreement with the experiment. The air was injected with constant velocity \( w \) in such way that air discharge \( Q_{air} = 0.5\% Q \), where \( Q \) is the nominal turbine discharge. The amplitude of pressure pulsations with usage of CFX SAS SST model is higher in average, see Fig. 6. The comparison with the experimental data is shown in Fig. 7.

\[
P, \text{ m.w.c.}
\]

![Figure 6. Pressure pulsations with air injection.](image)

\[
\Delta H/H, \%
\]

![Figure 7. Amplitude of pressure pulsations with air injection. Comparison with experimental data.](image)

The obtained pulsation amplitude was close enough to the experimental one. It can be seen from the experimental data that in the considered operating point the air injection reduced the amplitude of pulsations slightly. But in other operating points the air injection could significantly reduce the amplitude of pulsations.
Within the considered homogeneous mixture model the air injection didn’t have a significant impact on the flow field. The structure of the vortex rope remained unchanged, see Fig. 8.

The effect of the flow rate of the injected air on the amplitude and frequency of pulsations was also investigated, see Fig. 9. A series of calculations were performed with different air flow rates. The flow rate ranged from 0.5 to 2 percent.

![Figure 8](image1.png)

**Figure 8.** Flow field without air (left) and with air injection (right) at the same time step.

The frequency increase is observed in the experiment and didn’t captured in the computations.

6. **Discussion and conclusion**

A series of calculations of single-phase flow in part load operating point were performed in different solvers. The results obtained are in a good agreement with experimental data on the amplitude and frequency. The choice of the turbulence model had almost no effect on the pulsations pattern. The results obtained in different solvers are almost similar.

When air is injected, a change of the pulsation frequency is observed in the experimental data. The results of calculations showed that is not possible to identify this effect applying incompressible model. Both solvers were unable to capture this effect. Therefore, a new model was developed that takes into account the compressibility of the phase. The density of the air is determined by barotropic law.
\[ \rho_G = \left( \frac{P}{K_g} \right)^{\frac{1}{\gamma}}. \]

The mixture density \( \rho = (1 - \alpha_L) \left( \frac{P}{K_g} \right)^{\frac{1}{\gamma}} + \alpha_L \rho_L \). The flow is assumed to be adiabatic thus no heat exchange is observed. Therefore, the energy equation can be omitted and the number of equations remains the same. It is proposed to use the preconditioning method to solve the governing equations [8].

\[
 K^{-1} \frac{\partial q}{\partial \tau} + \frac{\partial Q}{\partial t} + \sum_i \frac{\partial F_i}{\partial x_i} = 0. \text{Where} \quad Q = \begin{pmatrix} \rho \\ \rho u \\ \rho v \\ \rho w \\ \alpha_L \end{pmatrix}, \quad q = \begin{pmatrix} \rho \\ u \\ v \\ w \\ \alpha_L \end{pmatrix}.
\]

\[
 K^{-1} = \begin{pmatrix} \tilde{a} & 0 & 0 & 0 & \rho_L - \rho_G \\ \tilde{a}u & \rho & 0 & 0 & (\rho_L - \rho_G)u \\ \tilde{a}v & 0 & \rho & 0 & (\rho_L - \rho_G)v \\ \tilde{a}w & 0 & 0 & \rho & (\rho_L - \rho_G)w \\ \tilde{a} \alpha_L & 0 & 0 & 0 & 1 \end{pmatrix}, \text{where} \quad \tilde{a} = \frac{\partial \rho}{\partial \rho} + a.
\]

Here \( a \) is a constant. Further work is aimed at applying this model to the calculation of flows in hydraulic turbines. Computations using this model are ongoing.

References

[1] Papillon B, Kirejczyk J and Sabourin M 2000 Atmospheric air admission in hydro turbines. Proc. of Hydrovision paper 3C

[2] 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)

[3] Pitorac L-I 2017 Numerical simulation of air injection in Francis turbines / NTNU Master thesis

[4] 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

[5] Chirkov, D., et al. "Mitigation of self-excited oscillations at full load: CFD analysis of air admission and effects of runner design." IOP Conference Series: Earth and Environmental Science. Vol. 49. No. 6. IOP Publishing, 2016.

[6] Chirkov, D., et al. "Numerical simulation of air injection in Francis turbine." IOP Conference Series: Earth and Environmental Science. Vol. 240. No. 2. IOP Publishing, 2019.

[7] 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

[8] Shin, B. R., Yamamoto, S., & Yuan, X. (2004). Application of preconditioning method to gas-liquid two-phase flow computations. J. Fluids Eng., 126(4), 605-612.