Numerical Prediction of Runaway Characteristics of Kaplan Turbines Applying Cavitation Model

Semenova A.V.,1, Chirkov D.V.,2, Ustimenko A.S.1

1. PJSC Power Machines Saint Petersburg, Russia
2. Institute of Computational Technologies SB RAS, Novosibirsk, Russia

E-mail: Semenova_AV@power-m.ru

In case of disconnection of generator from the network and failure of the governor, the rotational speed of the rotor rapidly increases and achieves maximum value, called the runaway speed. This value depends on the turbine type, the operating condition, the turbine flow passage and the runner geometry, and for Kaplan turbines might 2.5 times surpass the nominal rotational speed of the runner. The runaway speed is important for evaluation of mechanical resistance of the generator rotor. The value of runaway speed is determined based on the results of the model tests upon completion of the design works related to the runner. Therefore, accurate computation of the runaway speed will improve the design process for the runner and the whole turbine unit. Given in this paper is the numerical computation of the runaway speed for a Kaplan turbine of 40 m head. Calculations were carried out for a series of steady-state operating conditions with different runner speeds and the speed at which the torque on the runner shaft is equal to zero was determined. Two approaches for numerical simulation were compared. In the first one, the flow in the turbine was simulated using 3-D RANS equations of incompressible fluid using k-ε model of the turbulence. In the second approach, cavitation phenomena were taken into account using two-phase Zwart-Gerber-Belamri (ZGB) cavitation model. Steady-state computations were carried out in computational domain that included one guide vane channel, one runner channel, the whole draft tube, and the clearances between the runner blade and the hub as well as between the blade and the runner chamber. When setting the boundary conditions, the turbine head, being the difference of energies in the inlet and outlet cross-sections, is pre-set as a constant value, while the discharge and the runner torque are determined in the process of computation. The computed runaway speed is compared to that obtained in the model tests. It is shown that the numerical prediction of the runaway speed using the cavitation model achieves better matching with the experimental data.

1. Introduction

The runaway speed is the maximum runner speed that is achieved in case of emergency disconnection of the generator from the network and governor failure, thus creating great danger for the hydro-unit, especially for the generator rotor. Knowledge of this value is required for stress analysis of the components of the runner and other rotating parts connected with the shaft, stress analysis of the generator rotor, and calculation of the critical speed of the unit shaft rotation. At the present moment the runaway characteristics, i.e. runaway speed vs. turbine operating condition is obtained at the stage of model tests. Prediction of the runaway speed at the stage of runner design would allow to select a runner considering this characteristic and to guarantee fulfilment of the contract requirements. Given in the present paper is the method for CFD calculation of the runaway characteristics of the Kaplan turbine carried out using CADRUN software developed by ICT and Institute of Mathematics of Siberian Branch of Russian Academy of Sciences together with Power Machines.
2. Computational approach

The runaway speed depends on the turbine type, operating condition, the shape of the flow passage and runner blade system. The values of the runaway speed for different operating conditions of the turbine are determined via special model tests in course of which the model is brought to runaway with different positions of guide vanes and runner blades. Since during the runaway the turbine might get into cavitation condition, a constant value of the Thoma number (corresponding to the worst cavitation conditions at the power plant) is being maintained in the course of the model tests. The obtained values of the runaway speed are scaled up to the unit turbine \( D_l = 1 \text{ m}, H = 1 \text{ m} \). The runaway speed of the prototype turbine \( n_R \) is then calculated using corresponding unit runaway speed \( n_{11R} \) in accordance with the following formula:

\[
    n_R = n_{11R} \sqrt{\frac{H_{\text{max}}}{D_l}},
\]

where \( H_{\text{max}} \) is the maximum turbine head, \( D_l \) is the prototype runner diameter.

Two cases are possible when a Kaplan turbine is in the runaway condition. In the first case, failure of the governor makes it loose its ability to control the motion of the guide vanes and runner blades, and the relationship between the guide vane opening (GVO) and runner blade angle is broken. For this case, it is necessary to determine the greatest runaway speed value of all possible combinations of GVO and runner blade angle. As a rule, this value is achieved at small runner blade angle and high GVO. In the second case of governor failure, the guide vane opening/runner blade angle relationship is maintained, i.e. the runner blade angle depends on the GVO. In this case, runaway speed is less than in the first case since for small runner blade angle the GVO is also small.

We studied two CFD approaches for prediction of runaway characteristics. In the first approach, the flow in the turbine was simulated using 3D steady-state incompressible RANS equations with \( k-\varepsilon \) turbulence model. In the second approach the account of cavitation phenomena was added applying two-phase Zwart-Gerber-Belamri (ZGB) model.

In both cases, the combined method of calculation of energy losses was used [1]. The idea of the combined method is that energy losses in the main components of the flow passage are calculated directly using CFD, whereas energy losses in the remaining parts of the flow passage are calculated using simple empirical formulas without CFD computation. In this paper the losses in the spiral case and stay ring are calculated using empirical formulas, while the losses in the distributor, the runner and the draft tube are calculated using CFD analysis. One of the particularities of Kaplan turbines is the presence of clearances between the runner blade and the hub, and between the runner blade and the chamber. These clearances affect considerably the runner and draft tube as well as cavitation characteristics of the runner. Therefore, it is necessary to account the clearances in CFD analysis.

The calculations were carried out for several runner blade angles. And for each blade angle several guide vane openings were calculated, i.e. the worst condition (breakage of the guide vane opening/runner blade angle relationship) was studied. For each position of the guide vanes and runner blades a series of steady-state calculations were carried out for different rotational speed values and the speed at which torque on the shaft is equal to zero, was determined.

3. Boundary Conditions

In general, hydro-dynamic calculations of the flow in the water passages are carried out at fixed GVO, runner blade angle, and full head of the turbine. A priori, the discharge is not known and is determined in the course of solution. Such an approach was used by the authors for simulation of the single-phase incompressible and two-phase cavitating flows in turbine flow passage [2, 3, 4, 5]. Since computational domain consists only of one guide vane channel, one runner blade channel, and the draft tube, proper boundary conditions should be specified at the inlet of the guide vane domain. Here
the angle of the outflow from the stay ring is pre-set as a constant value as well as the total flow energy:

\[ E_{in} = E_{out} + H - h_{SP}, \]

where \( h_{SP} \) is the energy loss in the spiral case and stay ring evaluated a-priory in accordance with empirical formulas, \( H \) is the total turbine head, \( E_{out} \) is the flow energy in the draft tube outlet cross-section.

In case of calculation without considering cavitation phenomena, \( E_{out} \) can be set to zero:

\[ E_{out} = 0 \]  \hspace{1cm} (3)

In case of calculation applying cavitation model, \( E_{out} \) is calculated using Net Positive Suction Head (NPSH) or Thoma number. Let us assume that OZ axis of the coordinate system is directed downwards and level \( z=0 \) corresponds to the runner blade tilt axis (Fig.1). Then, in accordance to IEC standard 60193 [6] NPSH for a Kaplan turbine is determined using the formula:

\[ NPSH = \frac{p_{abs} - p_v}{\rho g} + \frac{V^2}{2g} z_2 \]  \hspace{1cm} (4)

Consider that Thoma number is equal to

\[ \sigma = \frac{NPSH}{H}. \]  \hspace{1cm} (5)

Therefore, energy \( E_{out} \) in the outlet cross-section can be expressed as follows:

\[ E_{out} = NPSH = \frac{p_v}{\rho g} = \sigma H - \frac{p_v}{\rho g}. \]  \hspace{1cm} (6)

Fig. 1 Scheme for calculating the energy in the turbine outlet cross-section

Thus, with a pre-set value of \( \sigma \) and known head \( H \), the energy \( E_{out} \) in the outlet cross-section is fixed, whereas discharge and pressure of the fluid in point 2 \( p_{abs,2} \) are not known as separate values and determined during the process of solution. The pressure at the inlet boundary is extrapolated from the inside of the computational domain, while all velocity components are extrapolated from the inside at the draft tube outlet boundary.
4. Mathematical models and numerical approach

Numerical simulation of 3D cavitating flow in turbine flow passage was carried out using homogeneous isothermal “liquid-vapor” mixture model:

\[
\frac{\partial \rho}{\partial t} + \text{div}(\rho \mathbf{v}) = 0, \quad (7)
\]

\[
\frac{\partial \rho \mathbf{v}}{\partial t} + \text{div}(\rho \mathbf{v} \otimes \mathbf{v}) + \nabla \rho = \text{div}(\mathbf{\tau}) + \rho \mathbf{f}, \quad (8)
\]

\[
\frac{\partial \alpha_L}{\partial t} + \text{div}(\alpha_L \mathbf{v}) = \frac{1}{\rho_L} (m^+ + m^-). \quad (9)
\]

Here \( \rho = \alpha_L \rho_L + (1 - \alpha_L) \rho_V \) is the mixture density [kg/m\(^3\)]; \( \rho_L, \rho_V \) are the densities of liquid and vapor, respectively; \( \alpha_L \) is the volume fraction of liquid phase, \( \alpha_V = (1 - \alpha_L) \) is the volume fraction of vapor phase; \( \mathbf{v} = (u_x, u_y, u_z) \) is the velocity vector [m/s]; \( \rho = p + \frac{2}{3} \rho k \); \( \rho \) is the static pressure [Pa]; \( k \) is the turbulence kinetic energy [m\(^2\)/s\(^2\)].

Static coordinate frame is used for the guide vane and the draft tube, while rotating coordinate frame is used for the runner, rotating around OZ axis with angular velocity \( \omega \). Thus, for the runner domain \( \mathbf{f} = (x_1 \omega^2 + 2u_2 \omega, x_2 \omega^2 - 2u_1 \omega, g) \), where \( g \) is the gravity acceleration.

In (8) \( \mathbf{\tau} = \tau_{ij} \) is the tensor of viscous stresses:

\[
\tau_{ij} = (\mu + \mu_T) \left[ \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} \frac{\partial u_k}{\partial x_k} \right],
\]

where \( \mu_T \) is the turbulent viscosity of the mixture, \( \mu \) is the dynamic viscosity of the mixture. The latter is calculated using the formula:

\[
\mu = \alpha_L \mu_L + (1 - \alpha_L) \mu_V, \quad (11)
\]

where \( \mu_L, \mu_V \) are the dynamic viscosities of liquid and vapor, respectively. Turbulent viscosity of the mixture is calculated using the formula:

\[
\mu_T = C_{\mu} \rho^k \frac{k^2}{\varepsilon}, \quad (12)
\]

where \( C_{\mu} = 0.09 \), \( \varepsilon \) is the rate of dissipation of turbulent kinetic energy [m\(^2\)/s\(^3\)]. For evaluation of \( k \) and \( \varepsilon \) equations (7)-(9) are supplemented by standard \( k-\varepsilon \) turbulence model with wall functions on solid walls.

Equation (9) describes the transport of liquid volume fraction \( \alpha_L \). Source terms \( (m^+) \) and \( (m^-) \) on the right hand side of equation (9) account for condensation and evaporation phenomena, respectively. These terms are evaluated using ZGB model [7]:

\[
\begin{align*}
    m^+ &= C_{\text{prod}} (1 - \alpha_L) \rho_V \sqrt{\frac{2 \max[0,p - p_V]}{\rho_L}}, & m^- &= -C_{\text{dest}} \alpha_L \rho_V \sqrt{\frac{2 \max[0,p_V - p]}{\rho_L}}.
\end{align*}
\]

Empirical constants \( C_{\text{prod}} \) and \( C_{\text{dest}} \) are evaluated as:

\[
C_{\text{prod}} = \frac{3F_{\text{cond}}}{R_b}, \quad C_{\text{dest}} = \frac{3\alpha_{\text{nuc}} F_{\text{vap}}}{R_b}, \quad (13)
\]

where \( p_V \) is the saturated vapor pressure, \( R_b = 10^{-6}\)m is the typical initial bubble radius, \( \alpha_{\text{nuc}} = 5 \times 10^{-4} \) is the volume fraction of non-condensable gases. According to recommendation of [7] the following
values of condensation and evaporation intensities are used in the present paper: $F_{\text{cond}} = 50$, $F_{\text{vap}} = 0.01$. In this case $C_{\text{prod}} = 3 \times 10^4$, $C_{\text{dest}} = 7.5 \times 10^4$.

When simulating 3D turbulent fluid flow without considering cavitation model, the liquid density $\rho = \text{const}$, and the equation (9) is excluded from the system. At that (7)-(8) reduce to well-known RANS equations for incompressible fluid.

Governing equations (7)-(9) of both single-phase and two-phase models are solved numerically using CADRUN solver, based on the artificial compressibility approach. Since only steady state solution is of interest, the system is altered by adding time derivatives of pressure to equations (7) and (9). The resulting system of PDE becomes evolutional and hyperbolic (in the absence of viscous terms). An implicit finite volume method for structured grids is used for discretization of the equations. Third order accurate MUSCL scheme is used for inviscid fluxes, while second order accurate central scheme is applied for viscous fluxes through cell faces. In order to prevent numerical oscillations, artificial dissipation is added on liquid-vapor interfaces, as suggested in [8]. Linearized system of discrete equations is solved using LU-SGS iterations. All equations (7)-(9) are solved for $(p, v, a_L)$ in a coupled manner. Details of the solver can be found in [9].

5. Results

The calculations were carried out for a turbine designed for $H = 40$ m with number of runner blades $z_1 = 6$. The calculations were carried out for the conditions of the scale model turbine: $D_1 = 0.46$ m, $H = 3.4$ m. For calculations of the case with applying of the cavitation model, Thoma number $\sigma = 0.6091$ (corresponding to the worst power plant cavitation conditions if the turbine falls into runaway) was pre-set. Gravity $g = 9.81$ m/s$^2$ was taken into account in all calculations. The calculations were carried out for several runner blade angles $\varphi = -5^\circ$, $0^\circ$ and $5^\circ$ and several guide vane openings $a_0 = 36$mm, 40mm, 44mm, 48mm, 52mm. During the model tests, maximum runaway speed was achieved at $a_0 < 52$mm. Relationships between the shaft speed of rotation of the model and discharge through the turbine and the respective unit values were determined using the following formulas:

$$n_{11} = \frac{n D_1}{\sqrt{H}}, \quad Q_{11} = \frac{Q}{D_1^2 \sqrt{H}}$$

where: $n_{11}$ is the unit rotational speed [rpm], $n$ is the rotational speed of the model turbine [rpm], $Q_{11}$ is the unit discharge [m$^3$/s], $Q$ is the model discharge [m$^3$/s], $D_1 = 0.46$m is the nominal diameter of the model runner, $H = 3.4$ m is the model net head.

For all CFD analyses periodic stage approach was used, which implies that the flow in all guide vane channels and blade-to-blade channels are considered to be identical. In this definition, steady-state CFD analysis was carried out in one guide vane channel, in one blade-to-blade channel and in the draft tube. Block structured mesh for numerical simulation is presented in Fig. 2.
All the computations have been carried out in steady state problem statement with number of iterations \( N = 35\,000 \). Fig. 3 shows typical evolutions of normalized hydraulic runner torque, discharge, and head, during the iterations. It can be seen that after 10000 iterations these integral characteristics come to periodic oscillations. This behavior indicates the absence of stable steady state solution in these operating points that lay far from the best efficiency point. The amplitude of these oscillations is less than 2% of the mean value. Therefore at the postprocessor stage in order to get “steady-state” values of runner torque \( M \) and discharge \( Q \), these characteristics were averaged over last 2000 iterations.

Figure 4 shows the grid convergence of the computed discharge for both non-cavitational and cavitational computations. An operating point close to no load point (\( \phi =0^\circ, a_0 = 52\text{mm}, n_{11}=280\text{rpm} \)) is considered. The number of cells is normalized by \( n_{\text{base}} \), where \( n_{\text{base}}=406030 \) is the number of cells in the basic mesh. It can be seen that discharge changes by about 0.5% with doubling the number of cells. Although the discharge still changes with mesh refinement, the influence of mesh is relatively small. In the rest of the paper all the computations have been performed on the basic mesh.

The method of computation of runaway speed is the following. For each runner blade angle \( \phi \) several guide vane openings \( a_0 \) were considered in order to capture the worst condition (breakage of the guide vane opening/runner blade angle relationship). For each combination of \( \phi \) and \( a_0 \) a series of steady-state CFD computations were carried out for different rotational speed values \( n_{11} \). From CFD computation the values of discharge and hydraulic runner torque were determined. At that mechanical losses were neglected. Based on these data the speed at which runner torque on the shaft is equal to zero, was determined. This value is the computed runaway speed.

Given in Figs. 5-10 is the comparison between CFD runaway curves (runaway speed vs guide vane opening \( a_0 \) and runaway speed vs unit discharge \( Q_{11} \)) and curves obtained in the Laboratory of Hydraulic Turbines, property of Power Machines (these experimental data were not published in the open literature).
It is seen from the results that in some runaway regimes cavitation phenomena start having a considerable influence on the parameters, such as rotational speed of the turbine at which the turbine reaches the runaway regime as well as discharge through the turbine. As it will be shown below, the influence of cavitation phenomena on runaway parameters, depends on the plant Thoma number and on the position of the guide vanes and runner blades. Note that prior to computations it is not known whether the turbine falls into cavitation regime or not. This is well illustrated in Figs. 11-14. Cavitation phenomena start having influence on the parameters only from some discharge.

![Fig. 11 Hydraulic torque on the model shaft vs unit rotational speed for φ = 0°, a₀ = 40mm](image1)

![Fig. 12 Hydraulic torque on the model shaft vs unit discharge for φ = 0°, a₀ = 40mm](image2)

![Fig. 13 Hydraulic torque on the model shaft vs unit rotational speed for φ = 0°, a₀ = 52mm](image3)

![Fig. 14 Hydraulic torque on the model shaft vs unit discharge for φ = 0°, a₀ = 52mm](image4)
During the model tests, the influence of the Thoma number on the runaway speed was studied. Given in Figs. 15-17 is the comparison between the curves obtained via experiments and the curves obtained via calculations.

![Fig. 15 Runaway speed vs Thoma number for $\phi = -5^\circ$, $a_0 = 48\text{mm}$](image)

![Fig. 16 Runaway speed vs Thoma number for $\phi = 0^\circ$, $a_0 = 46\text{mm}$](image)

![Fig. 17 Runaway speed vs Thoma number for $\phi = 5^\circ$, $a_0 = 46\text{mm}$](image)

It is seen from the above results that account of cavitation has a considerable influence on the computed runaway speed. Cavitation has also the influence on the discharge through the turbine corresponding to the runaway speed. Hence, these characteristics should be computed using cavitation model.

From Figs. 5-10 it is seen that calculations using the selected model of cavitation and statement of boundary conditions give a good agreement to experimental data both in shape of the curves and the values of the runaway speed. The value of the runaway speed is slightly over-predicted, especially for small runner blade angles. This behavior might be explained by neglecting the mechanical losses. The results of calculations presented in Figs. 15-17 also show a satisfactory coincidence with the experimental results.
6. Conclusions

Two approaches for predicting the runaway characteristics of a Kaplan turbine have been studied. In the first approach, the flow in the turbine is simulated using 3D steady-state incompressible flow model. In the second approach, the account of cavitation was added using a Zwart-Gerber-Belamri two-phase cavitation model. In both cases a constant the turbine head \( H = \text{const} \) was set as a boundary condition, while the discharge through the turbine was determined in the process of solution. It is shown that the cavitation phenomena have a considerable influence on the turbine runaway characteristics. Single-phase incompressible computations significantly over-predict the runaway speed especially for high guide-vane openings. At that calculations using the selected model of cavitation give a good agreement to experimental data both in shape of the curves and the values of the runaway speed. The value of the runaway speed is slightly over-predicted, especially for small runner blade angles. This behavior might be explained by neglecting the mechanical losses. This issue should be investigated further.

Thus, the described CFD approach considering cavitation is promising since it can predict runaway characteristics of the turbine and might be used when performing design works or justification via calculations.

References

1. A. Sotnikov, S. Cherny, D.V. Chirkov, D. Bannikov Two Approaches to Prediction of Hill Diagram for Francis Turbine. // Proceedings of Hydro Vision Russia – 2011, (Moscow, Russia) 2011. – P. 1-15.
2. Semenova A., Chirkov D., Lyutov A., Cherny S., Skorospelov V., Pylev I. Multi-objective shape optimization of runner blade for Kaplan turbine //27th IAHR Symposium on Hydraulic Machinery and Systems, Montreal, Canada 2014 IOP Conf. Series: Earth and Environmental Science 22 (2014) 012025 doi:10.1088/1755-1315/22/1/012025 P. 1-10.
3. Panov L.V., Chirkov D. V., Cherny S. G., Pylev I. M. and Sotnikov A. A.: Numerical modeling of steady-state cavitational flow of viscous fluid in Francis hydroturbine. //Thermophysics and Aeromechanics. 2012. Vol. 19, No. 3. pp. 415-427
4. Panov L.V., Chirkov D.V., Cherny S.G., and Pylev I.M. Numerical simulation of pulsation processes in hydraulic turbine based on 3D model of cavitating flow // Thermophysics and Aeromechanics, 2014, Vol. 21, No. 1 P. 31-43.
5. Semenova A.V., Varlamov A.A., Georgieva Yu.G., Zakharov A.V, Sokolova M.A., Improvement of cavitational characteristics of the water passages of Votkinskaya HPP // Hydro 2019.10.14-16, Porto, Portugal. P 1-6
6. IEC Standard 60193. Hydraulic turbines, storage pumps and pump-turbines. Model acceptance tests / IEC: International Electrotechnical Commission. – 1999.
7. Zwart P J, Gerber A G, Belamri T A 2004 Two-phase flow model for predicting cavitation dynamics. ICMF 2004 International Conference on Multiphase Flow. Paper No. 152 (Yokohama, Japan).
8. 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
9. Panov L V, Chirkov D V, Cherny S G, Pylev I M and Sotnikov A A 2012 Numerical modeling of steady-state cavitational flow of viscous fluid in Francis hydroturbineThermophysics and Aeromechanics 19 No. 3 p. 415-427