Numerical Investigation of the Flow Structure in a Kaplan Draft Tube at Part Load

R Maddahian¹, M J Cervantes² and N Sotoudeh¹

¹Faculty of Mechanical Engineering, Tarbiat Modares University, Tehran, I.R. Iran
²Division of Fluid and Experimental Mechanics, Lulea University of Technology, SE 971 87, Lulea, Sweden

Abstract. This research presents numerical simulation of the unsteady flow field inside the draft tube of a Kaplan turbine at part load condition. Due to curvature of streamlines, the ordinary two-equations turbulence models fail to predict the flow features. Therefore, a modification of the Shear Stress Transport (SST-SAS) model is utilized to approximate the turbulent stresses. A guide vane, complete runner and draft tube are considered to insure the real boundary conditions at the draft tube inlet. The outlet boundary is assumed to discharge into the atmosphere. The obtained pressure fluctuations inside the draft tube are in good agreement with available experimental data. In order to further investigate the RVR formation and its movement, the $\lambda_2$ criterion, relating the position of the vortex core and strength to the second largest Eigen value of the velocity gradient tensor, is employed. The method used for vortex identification shows the flow structure and vortex motion inside the draft tube accurately.

1. Introduction

Hydraulic turbines are generally designed to work at the Best Efficiency Point (BEP). The deregulation of electricity market, the variation of the electricity demand and the increasing share of renewable energy contribute to run hydraulic turbines more often at off-design conditions with an increase in start/stop. On the other hand, many installed hydraulic turbines are aged and need renovation. The output power, efficiency and the operating range of renovated hydro power plant should be improved in comparison to old one. Due to the capital cost of the construction process, usually the runner and guide vanes are redesigned. Ignoring refurbishment of spiral case and draft tube may cause an unstable flow, especially in the draft tube. Unstable flow in the draft tube mostly occurs at off-design condition. Therefore, an accurate investigation of turbines at off-design may allow designing/redesigning them with a larger operational capability [1].

A draft tube is installed at the runner outlet of hydraulic turbines to convert the remaining kinetic energy to pressure energy. Up to fifty percent of the losses may occur in low head draft tube therefore the overall efficiency of low head turbines with fixed blades is significantly affected by the performance of the draft tube. The amount of swirl at the runner exit is adjusted to avoid flow separation even for cone angle up to 14°. At part load condition, the amount of swirl at the runner exit increases leading to a vortex breakdown characterize by a low velocity region, sometimes recirculating, in the middle and a rotating vortex rope (RVR). Low frequency and large amplitude pressure fluctuations appear in the draft tube. The RVR seems to be created by the high shear at the...
interface between the recirculation zone and main flow [2-4]. Two types of RVR were seen in draft tubes of Francis turbine [5]. The spiral type occurs below 60% of full load. At higher part load (up to 85%); the vortex breakdown phenomenon may occur and the spiral type RVR changes to bubble type. The former one generates severe pressure fluctuations in the draft tube and may damage the whole turbine’s installation. Furthermore, the presence of a recirculation zone at part load with a RVR reduces the turbine performance, decreases the available area of the diffuser and increases the axial velocity of flow near the wall. The high-velocity gradient and non-uniform flow near the wall increase the hydraulic losses.

The application of Computational Fluid Dynamics (CFD) has been grown not only to understand flow field in hydraulic turbines, but also to design new installations [6-9]. The hydraulic stability in a wide range of operating conditions is the key issue of new designs.

Previous investigations on unsteady flow inside draft tube focus on the relation between low frequency fluctuations and RVR rotation [10-12]. A few of them have studied the structure of flow in the draft tube with RVR or during the RVR formation [5, 13-17].

Zhang et al. [5] numerically investigated the unsteady flow inside the draft tube of a Francis turbine during the part load condition. The reverse axial flow in the inlet cone section was found to be the reason behind the instability of the swirling flow. This flow reversal continued through the elbow segment. The structure of the flow was also visualized using $\Delta$-criterion. The pressure fluctuations can be controlled by jet injection from the runner cone. Foroutan and Yavuzkurt [13, 14] developed Partially Averaged Navier-Stokes (PANS) turbulence model and showed that using PANS turbulence model for predicting flow behavior improves the prediction of the pressure recovery factor and pressure fluctuations specially at part load condition. They used the iso-surface of pressure to visualize the structure of the RVR.

Recently, Javadi and Nilsson [15-17] investigated the flow structure and pressure fluctuations inside a draft tube of Kaplan turbine at BEP, Porjus U9. The coherent structure of the flow was visualized using Q-criterion. They showed that the forced vortex in the center of the draft tube is surrounded by hub vortex. Although the obtained velocity filed near the runner was in good agreement with experimental measurements, a deviation was seen downstream the runner. The hybrid RANS-LES turbulence modal should be used in order to predict the main flow feature and coherent structure in swirling flow was recommended for such simulation. The injection of water from the runner cone was shown to be an effective way to control the RVR.

Several ongoing experimental and numerical research projects are carried out at Lulea University of Technology about flow features, unsteadiness and vibrations of different parts of the Porjus U9 Kaplan turbine [3, 4, 18-20]. Nearly all previous numerical studies were performed at BEP point. The pressure fluctuations inside the draft tube of Kaplan turbine run as a propeller is more complex due to the interaction of tip vortex with stationary shroud wall [19, 20]. On the other hand, the low frequency pressure fluctuations become dominant at part load condition. Therefore, the nature and reasons of pressure fluctuations and controlling methods at part load are two important subjects which need more investigations.

The previous vortex identification inside Kaplan draft tube could not accurately recognize the structure of RVR. Therefore, the authors try to identify the swirling flow structure in Kaplan draft tube and the mechanism behind RVR formation. The present work is the first step of numerical investigation of Kaplan draft tube at part load condition and is devoted to the unsteady motion of the RVR. Governing equations are presented in section 2. Considered physical model and numerical method are reviewed in section 3 and 4, respectively. Section 5 is devoted to the results of the simulation and discussions.

2. Governing Equations and Mathematical Model

The fundamental transport equations for isothermal and incompressible fluid are considered. A general time averaging filter is applied to unsteady Navier-Stokes equations and results in the following equations:
\[ \frac{\partial \bar{u}_i}{\partial x_i} = 0 \]  

\[ \rho \left( \frac{\partial \bar{u}_i}{\partial t} + \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} \right) = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \mu \frac{\partial \bar{u}_i}{\partial x_j} \right) - \frac{\partial}{\partial x_j} \left( \rho \bar{u}_i' u_j' \right) \] (2)

where \( \bar{u}_i \) is the time-averaged velocity, \( u_i' \) is the velocity fluctuation, \( \bar{p} \) is the time-averaged pressure, \( \rho \) and \( \mu \) are the fluid density and viscosity, respectively. The last terms on the right hand side of Eq. (2) is called Reynolds stress tensor and needs additional equations to resolve it.

The eddy-viscosity model is employed in the present work to resolve the Reynolds stress tensor. Therefore, an additional equation for calculating Reynolds stress tensor can be written as follows:

\[ -\rho \left( \bar{u}_i' u_j' \right) = 2 \mu S_{ij} - \frac{2}{3} \rho k \delta_{ij} \] (3)

where \( \mu \) is the turbulent viscosity, \( k \) is the mean turbulent kinetic energy and \( S_{ij} \) is the mean strain tensor. The mean kinetic energy and strain rate are defined in Eq. (4) and (5).

\[ k = \frac{1}{2} \left( \bar{u}_i' u_i' \right) \] (4)

\[ S_{ij} = \frac{1}{2} \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) \] (5)

In order to calculate the turbulent viscosity the Shear Stress Transport-Scale Adaptive Simulation (SST-SAS) turbulence model is employed [21]. In SST-SAS model, two equations for \( k \) and \( \omega \) (specific dissipation rate) are solved with an additional source term in \( \omega \) equation. The SAS development of SST model is based on using Von-Karman length scale for turbulence length scale and gives the capability of modeling unsteadiness to ordinary SST model. Using SAS for unsteady flow showed superior results improvement in comparison with old one [22]. The formulation of SST-SAS can be found in [21, 22].

3. Physical Model

A model scaled 1:3.1 from the Porjus U9 Kaplan turbine was built at Vattenfall research and development (VRD) facilities Älvkarleby, Sweden. The model turbine has a runner diameter of 0.5 m. The operational head, discharge and guide vane angle of the turbine at part load were set to 7.5 m, 0.62 m³/s and 20°, respectively. The runner speed was constant and set to \( N=696.3 \) rpm. The velocity measurements have been performed using a two-component laser Doppler anemometry (LDA) from Dantec. The turbine was installed between two pressurized tanks, which give the ability of various operating heads. Details of the measurements and test rig configuration could be found in [3, 4, 18].

The measurement locations and the considered geometry of the U9 draft tube are shown in Fig. 1.

4. Numerical Method

4.1. Geometry and grid

Following the work presented by Melot et al. [23], the geometry simulated consists in one stay vane, one guide vane, the complete runner and the draft tube. In order to simulate accurately the flow near the runner cone, the runner domain extends below the runner cone. The disk below the runner represents the runner – draft tube interface.

The mesh was generated using ANSYS-ICEM software. The hexahedral element is employed to ensure precise simulation of turbulent swirling flow, especially near the draft tube wall.
The total cell number for complete geometry is approximately $2.6 \times 10^6$. The cell numbers for the stay vane and guide vane, runner and draft tube are $0.3 \times 10^6$, $1.3 \times 10^6$ and $1.0 \times 10^6$, respectively. The grid topology has two interfaces between stationary domains (stay/guide vane and draft tube) and rotating domain (runner). In order to accurately simulate the unsteady flow between the domains, a General Grid Interface (GGI) handles the linkage between them. During the simulations, a cell zone in the rotating region slides relative to a stationary region, and the interface flux is calculated based on two adjacent cells.

4.2. Boundary conditions
The inlet boundary condition at the stay vane is derived from a full spiral simulation. The flow rate and prescribe flow angle with 5% turbulent intensity are set at the inlet. The no-slip condition is also used as wall boundary condition. Turbulent quantities near the wall are evaluated using standard wall function. A zero pressure is considered at the draft tube outlet considering an opening.

4.3. Solution Methodology
The commercial software ANSYS-CFX is used to solve the unsteady flow. Transient simulations are performed with time step $3 \times 10^{-4}$ s. Advection and diffusion terms are discretized using high resolution and central difference scheme, respectively. In order to control the turbulence quantities during the simulations, the upwind scheme is employed to discretize the turbulence equations. Total simulation time was 8.5 s. The Root Mean Square (RMS) value of residuals in the convergence state at each time step is less than $10^{-5}$.

5. Results and Discussions
5.1. Validation
In order to validate the numerical simulation of the swirling flow inside draft tube, pressures are compared with experimental measurements at four points over one period of the RVR. The comparison is shown in Fig. 2 (a) to (d). The agreement between numerical simulations and experimental measurements is poor at the first pressure tap. This pressure tap is located above the runner – draft tube interface, the pressure pulsation of the draft tube are damped. The interface is seen in Fig. 1. Above the interface, the domain is in the rotating reference frame while the stationary domain is considered below the interface. The sliding interface is employed as a boundary condition between domains. The pressure pulsation of the draft tube should be transferred from the stationary domain to rotating one. During the transfer, the draft tube pressure pulsations are damped. The other
pressure variations are in good agreement with the experimental results. The RVR period was experimentally determined to 0.492 s. The numerically obtained period is 0.488 s.

![Graphs showing comparison of pressure fluctuations at different locations in draft tube](image)

(a) (b) (c) (d)

Fig. 2 The comparison of pressure fluctuations at different locations in draft tube, (a) Point 1, (b) Point 2, (c) Point 3, (d) Point 4

5.2. Flow Structure

Three well-known criteria, the Q-criterion, Δ-criterion and λ2-criterion, have been used to visualize the swirling flow structure [24-26]. In the present work the λ2-criterion is employed due to its ability to identify a vortex core. The λ2-criterion is based on the second largest Eigen value of $S^2 + \Omega^2$ tensor where $S$ and $\Omega$ are the symmetric and asymmetric parts of the velocity gradient tensor. The negative value of $\lambda_2$ in a connected region corresponds to the vortex core.

Figures 3(a) and 3(b) present the identification of swirling flow structure inside draft tube using pressure minimum criteria and $\lambda_2$-criteria, respectively. Although the minimum pressure criteria identify the structure of the RVR, the method is not capable of distinguishing between different vortices inside the draft tube. Two typical swirling structures can be seen in Fig. 3(b). The tip vortex which is created due to the runner rotation and the RVR structure. The tip vortex has the frequency of
runner and creates small amplitude pressure fluctuations (Fig. 2(a)), but the RVR has small frequency and creates large pressure fluctuations (Fig. 2(d)). The axis of rotation as well as the stretching of RVR to the elbow section are well identified by \( \lambda_2 \)-criteria. It should be mentioned that the structure captured by this method is in agreement with visualization using air injection [4].

Velocity vectors are shown in a section of the inlet cone. Recirculation zones created by RVR can be seen near the RVR axis of rotation. The recirculation zones, decrease the effect of diffuser and reduce the draft tube overall efficiency.

Tangential velocity contour at different axial cross sections is shown in Fig. 4. RVR rotation creates a local maximum and minimum peak in tangential velocity. Tangential velocity in the minimum velocity area becomes negative while the bulk flow has positive tangential velocity. Therefore, the rotational speed of RVR is different from bulk flow. The RVR acts like a rotating spinner which moves on the swirling plate. The movement of RVR depends on interaction of two swirling flows (i.e. RVR and bulk flow). The controlling methods of RVR should bring the spinner to the center of rotating plate or synchronize their rotational speeds.

Fig. 3 Flow structure inside draft tube identified by (a) iso-pressure contour (b) \( \lambda_2 \) iso-surface contour

Fig. 4 Tangential velocity contours at different cross section of draft tube, RVR is demonstrated by iso-pressure contour
6. Conclusion

Transient analysis of swirling flow inside draft tube of U9 Kaplan turbine is performed using numerical simulations. Numerical results are in good agreement with experimental measurements especially downstream of the runner. Therefore, simulations in unsteady mode using modified SST model is capable of predicting flow unsteadiness inside draft tube and can be used for further investigation of turbine at part load. Numerical results show that the movement of RVR in draft tube creates low frequency pressure fluctuations inside Kaplan draft tube similar to Francis one. The structure of RVR is identified using $\lambda_2$ criterion and shown that the RVR can stretch and extend after elbow in draft tube. Moreover, the tip vortex which is created due to the runner and shroud interaction is visualized. The tip vortex downstream of runner dissipates and the RVR controls the swirling flow inside draft tube. Any changes in flow structure which control the RVR rotational movement, can reduce low-frequency pressure fluctuations. The modification would be water admission from Kaplan propeller which is considered as future investigations.

References

[1] S. Mauri, "Numerical simulation and flow analysis of an elbow diffuser," 2002.
[2] U. Andersson, An experimental study of the flow in a sharp-heel Kaplan draft tube: Luleå tekniska universitet, 2009.
[3] P. Jonsson, Flow and pressure measurements in low-head hydraulic turbines: Luleå tekniska universitet, 2011.
[4] B. Mulu, "An experimental and numerical investigation of a Kaplan turbine model," 2012.
[5] R.-k. Zhang, F. Mao, J.-Z. Wu, S.-Y. Chen, Y.-L. Wu, and S.-H. Liu, "Characteristics and control of the draft-tube flow in part-load Francis turbine," Journal of fluids engineering, vol. 131, p. 021101, 2009.
[6] P. Drtina and M. Sallaberger, "Hydraulic turbines—basic principles and state-of-the-art computational fluid dynamics applications," Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science, vol. 213, pp. 85-102, 1999.
[7] A. Ruprecht, M. Heitele, T. Helmrich, W. Moser, and T. Aschenbrenner, "Numerical simulation of a complete Francis turbine including unsteady rotor/stator interactions," in Proceedings of the 20th IAHR Symposium on Hydraulic Machinery and Systems, 2000, pp. 1-8.
[8] A. Ruprecht, T. Helmrich, T. Aschenbrenner, and T. Scherer, "Simulation of vortex rope in a turbine draft tube," in Proceedings of 22nd IAHR Symposium on Hydraulic Machinery and Systems, 2002, pp. 9-12.
[9] P. Ko and S. Kurosawa, "Numerical simulation of turbulence flow in a Kaplan turbine- Evaluation on turbine performance prediction accuracy," in IOP Conference Series: Earth and Environmental Science, 2014., p. 022006.
[10] R. Susan-Resiga, G. D. Ciocan, I. Anton, and F. Avellan, "Analysis of the swirling flow downstream a Francis turbine runner," Journal of Fluids Engineering, vol. 128, pp. 177-189, 2006.
[11] S. Liu, S. Li, and Y. Wu, "Pressure fluctuation prediction of a model Kaplan turbine by unsteady turbulent flow simulation," Journal of Fluids Engineering, vol. 131, p. 101102, 2009.
[12] Y. Wu, S. Liu, H.-S. Dou, S. Wu, and T. Chen, "Numerical prediction and similarity study of pressure fluctuation in a prototype Kaplan turbine and the model turbine," Computers & Fluids, vol. 56, pp. 128-142, 2012.
[13] H. Foroutan and S. Yavuzkurt, "A partially-averaged Navier–Stokes model for the simulation of turbulent swirling flow with vortex breakdown," International Journal of Heat and Fluid Flow, vol. 50, pp. 402-416, 2014.
[14] H. Foroutan and S. Yavuzkurt, "Unsteady Numerical Simulation of Flow in Draft Tube of a Hydroturbine Operating Under Various Conditions Using a Partially Averaged Navier–Stokes Model," *Journal of Fluids Engineering*, vol. 137, p. 061101, 2015.

[15] A. Javadi and H. Nilsson, "Unsteady numerical simulation of the flow in the U9 Kaplan turbine model," in *IOP Conference Series: Earth and Environmental Science*, 2014, p. 022001.

[16] A. Javadi and H. Nilsson, "Active Flow Control of Vortex Rope in a Conical Diffuser," in *IAHR WG Meeting on Cavitation and Dynamic Problems in Hydraulic Machinery and Systems*, 2015.

[17] A. Javadi and H. Nilsson, "Time-accurate numerical simulations of swirling flow with rotor-stator interaction," *Flow, Turbulence and Combustion*, vol. 95, pp. 755-774, 2015.

[18] K. Amiri, "An experimental investigation of flow in a Kaplan runner steady-state and transient," 2014.

[19] B. Mulu, P. Jonsson, and M. Cervantes, "Experimental investigation of a Kaplan draft tube--Part I: Best efficiency point," *Applied Energy*, vol. 93, pp. 695-706, 2012.

[20] P. Jonsson, B. Mulu, and M. Cervantes, "Experimental investigation of a Kaplan draft tube--Part II: Off-design conditions," *Applied Energy*, vol. 94, pp. 71-83, 2012.

[21] F. Menter and Y. Egorov, "A scale-adaptive simulation model using two-equation models," *AIAA paper*, vol. 1095, p. 2005, 2005.

[22] Y. Egorov and F. Menter, "Development and application of SST-SAS turbulence model in the DESIDER project," in *Advances in Hybrid RANS-LES Modelling*, ed: Springer, 2008, pp. 261-270.

[23] M. Melot, B. Nennemann, and N. Désy, "Draft tube pressure pulsation predictions in Francis turbines with transient Computational Fluid Dynamics methodology," in *IOP Conference Series: Earth and Environmental Science*, 2014.

[24] J. C. Hunt, A. Wray, and P. Moin, "Eddies, streams, and convergence zones in turbulent flows," 1988.

[25] M. Chong, A. E. Perry, and B. Cantwell, "A general classification of three-dimensional flow fields," *Physics of Fluids A: Fluid Dynamics (1989-1993)*, vol. 2, pp. 765-777, 1990.

[26] J. Jeong and F. Hussain, "On the identification of a vortex," *Journal of fluid mechanics*, vol. 285, pp. 69-94, 1995.