Unsteady numerical simulation of the flow in the U9 Kaplan turbine model

Ardalan Javadi, Håkan Nilsson
Department of Applied Mechanics, Chalmers University of Technology, Gothenburg, SE-412 96, Sweden
E-mail: ardalan.javadi@chalmers.se

Abstract. The Reynolds-averaged Navier-Stokes equations with the RNG k-ε turbulence model closure are utilized to simulate the unsteady turbulent flow throughout the whole flow passage of the U9 Kaplan turbine model. The U9 Kaplan turbine model comprises 20 stationary guide vanes and 6 rotating blades (696.3 RPM), working at best efficiency load (0.71 m³/s). The computations are conducted using a general finite volume method, using the OpenFOAM CFD code. A dynamic mesh is used together with a sliding GGI interface to include the effect of the rotating runner. The clearance is included in the guide vane. The hub and tip clearances are also included in the runner. An analysis is conducted of the unsteady behavior of the flow field, the pressure fluctuation in the draft tube, and the coherent structures of the flow. The tangential and axial velocity distributions at three sections in the draft tube are compared against LDV measurements. The numerical result is in reasonable agreement with the experimental data, and the important flow physics close to the hub in the draft tube is captured. The hub and tip vortices and an on-axis forced vortex are captured. The numerical results show that the frequency of the forced vortex in 1/5 of the runner rotation.

1. Introduction
Hydropower is the longest established source for the generation of electric power, which developed into an industrial size plant following the demonstration of the economic transmission of high-voltage AC. The increasing need for more power during the early years of the twentieth century also led to the invention of a turbine suitable for small heads of water, i.e., 3-9 m, in river locations where a dam could be built. In 1913, Viktor Kaplan revealed his idea of the propeller (or Kaplan) turbine, see figure 1, which acts like a ship’s propeller but in reverse. At a later date, Kaplan improved his turbine by means of swiveling blades, which improved the efficiency of the turbine appropriate to the available flow rate and head. The Kaplan turbine incorporates the essential feature that the setting of the runner blade angle can be controlled by a servomechanism to maintain optimum efficiency conditions. This adjustment requires a complementary adjustment of the guide vane angle to maintain an almost swirl-free flow at the exit from the runner.

According to Drtina and Sallaberger [1] and Nilsson [2], the use of computational fluid dynamics (CFD) for predicting the flow in these machines has brought further substantial improvements in their hydraulic design and resulted in a more completed understanding of the flow processes and their influence on the turbine performance. Details of flow separation, loss sources, and loss distributions in components both at design and off-design, as well as detecting low-pressure levels associated with the risk of cavitation, are now amenable to analysis with the aid of CFD [3]. Many investigators have
applied CFD as a numerical simulation tool for the analysis of other turbines [4-9] while there are only a handful profound numerical investigations of the Kaplan turbine. Recently, studies have been performed experimentally and numerically, showing that the low-frequency pressure fluctuation in the draft tube possesses a frequency of 0.15–0.33 times the runner’s rotating frequency, and that this pressure fluctuation is mainly induced by the flow at the area near the inlet of the draft tube. Wu et al. [10] studied the pressure fluctuations of a prototype Kaplan turbine using RNG k-ε and ended up with close similarity between pressure field in the prototype and the model. Liu et al. [11] analyzed pressure fluctuation in a Kaplan turbine draft tube and captured a low frequency vortex rope that rotated in the opposite direction of the runner. Owing to the complexity of the flow field which contains, acceleration, deceleration, transition, relaminarization, separation and reattachment, an adequate numerical analysis of such a flow is necessary to help experiment to widen the understanding about the coherent and turbulent structures.

In this paper a model Kaplan turbine is investigated numerically using RNG k-ε at best efficiency point. The full runner passages and guide vanes with all geometrical details are included in the computational domain. The coherent structures, and the instantaneous and periodic features of the flow in the draft tube are discussed. The results show a close agreement with experimental observation and the structures are further clarified. Although the diffusion of the vorticity and the decay of the concentrated vortices possess highly complicated properties and is cumbersome to be captured by time-dependent solutions, the physical properties of the on-axis vorticity pocket is presented.

2. Flow configuration and numerical aspects

The studied test case is the 1:3.1 scale model of the U9 Kaplan turbine prototype. The model located in Ålvkarleby and the prototype located in Porjus, Sweden. The operational head of the prototype is 55 m with a maximum discharge capacity of 20 m³/s for a power of 10 MW. The turbine is composed of a spiral casing, 18 stay vanes, 20 guide vanes, 6 runner blades and an elbow draft tube. The prototype runner diameter is 1.55 m. The diameter of the model runner is \( D = 0.5 \) m, with an operational net head of \( H = 7.5 \) m, a runner speed of \( N = 696.3 \) RPM, a flow rate is \( Q = 0.71 \) m³/s and a guide vane angle of 26°. The Reynolds number based on draft tube cone diameter and bulk velocity is about \( 1.5 \times 10^7 \). The present work investigates the turbine at BEP and compares the numerical results with experimental measurement of Mulu [12].

The calculations reported herein are performed using the finite-volume method in the OpenFOAM open source CFD code. The governing equations are the continuity and momentum equations for incompressible flow. The code is parallelized using domain decomposition and the Message Passing Interface (MPI) library. The simulation is performed using an AMD Opteron 6220 Linux cluster and 64 cores. The second-order central difference scheme is used to discretize the diffusion terms, and the second-order linear upwind difference scheme is adopted to approximate the convection term. The time-marching is performed with an implicit second-order accurate backward scheme. The General Grid Interface (GGI) [13] is used at the sliding interfaces between the rotor and the stator. The main advantage of the GGI is that it allows for non-conformal meshes at the interface. It makes mesh generation easier for complex geometries, and facilitates a sliding grid approach. It has been shown to give a close agreement for the velocity results between non-conformal and conformal meshes. However, in the present study it was required to have a proper circumferential mesh resolution on each side of the GGI to yield good results for the turbulent kinetic energy. Nilsson et al. [14] validated the use of non-conformal meshes arguing that the spacing should be comparable in the radial and axial direction.

Figure 1 shows the computational domain which includes, the 20 guide vanes, the 6 runner blades and the draft tube. The total number of cells in the domain is \( 5.7 \times 10^6 \) where the guide vanes, the runner and the draft tube have \( 2.653 \times 10^6, 1.211 \times 10^6 \) and \( 1.847 \times 10^6 \) cells, respectively. The mesh generation is a challenging task, especially since the geometrical details such as the clearance of the guide vanes, the hub and tip clearance of the runner blades are included in the computational domain.
The domain was realized in ICEM Hexa, and is divided in three different parts. Those different parts are coupled in OpenFOAM. Figure 1b shows the mesh in the guide vane passage and the runner. A total number of 70 cells with angle smaller than 25 degrees is reported which shows the adequate mesh generation process. The quality of the mesh is a vital element in capturing correct coherent structures, particularly in such a vortex dominated flow.

A constant velocity, yielding the required mass flux, is applied at the inlet, no-slip at walls and homogenous Neumann at outlet. All boundary conditions for pressure are homogenous Neumann. Due to the geometrical complexity, the inlet velocity and the rotational speed are ramped up from very small values to the physical ones. The relaxation factors are also step-wise increased during the simulation to make the solution converged. The unsteady simulation with maximum CFL number of 6.2 is utilized to capture more accurate unsteadiness and coherent structures.

![Figure 1. a) computational domain b) mesh in guide vane passage and runner.](image)

3. Results and discussion

The simulation is conducted for more than 25 runner revolutions to establish fully periodic flow. The velocity components are averaged for 5 complete runner revolutions. The velocity is normalized by the bulk velocity at the draft tube cone. The survey axis, $S^*$, is normalized by the cone radius. Figure 2 shows the mean axial velocity in the draft tube and three survey axis (I, II and III) where the results are extracted. The mean axial velocity contour shows a small recirculation region after the runner cone. There is a central low velocity region which is contracted by two high velocity regions. These two high velocity regions are generated by the jet from the hub clearance, (hereafter referred as the hub jet) and expanded downstream.

Figure 3 shows the axial and tangential velocity compared with experimental results at the three cross-sections, I, II and III. The velocity components at cross-section I and II are under high influence of the hub jet. This jet is diffused and less strong at cross-section III. There is a central low velocity region which is contracted by two high velocity regions. These two high velocity regions are generated by the jet from the hub clearance, (hereafter referred as the hub jet) and expanded downstream.

Figure 3 shows the axial and tangential velocity compared with experimental results at the three cross-sections, I, II and III. The velocity components at cross-section I and II are under high influence of the hub jet. This jet is diffused and less strong at cross-section III. There is an on-axis forced vortex which characterized by constant angular velocity (velocity / radius=constant). The forced vortex is basically surrounded by a free vortex which is characterized by inversely varying speed as the distance from the center (velocity × radius=constant) while in the current flow field the free vortex may not be formed because of the hub jet. Nevertheless, the forced vortex is stable for many axial and tangential disturbances, as expected [15]. The core of the forced vortex presents a linear distribution according to its definition, as it can be seen in $S^*<0.2$ at cross-section I. The vortex filament is isolated, strong, concentrated. The mid-part of cross-section I, $0.2<S^*<0.5$, is dominated by the hub jet. Thus, the maximum axial velocity occurs at $S^*=0.2$ which can be related to the hub jet. There is a local decrease in the axial velocity which should be related to the hub vortex. This vortex follows the hub jet and the
trajectory shows the axial trace of the vortex, see figure 5. The hub jet leads to a linear decrease in the tangential velocity at $0.15 < S^* < 0.3$. This decrease leads to a negative vorticity surrounding the central forced vortex, see figure 5. It is worth mentioning that the hub jet generates a strong vortex that is attached to the runner cone and finally detaches from the cone. This is the reason behind the hump-like peak of the axial velocity while the tangential velocity is sharp peaked, see figure 3a. There is runner blade wake separating from the trailing edge of the blade (suction side), traveling downward. This wake is responsible for the plateau in the axial velocity in $0.5 < S^* < 1$, see figures 3a and 5. Figure 5 also shows a coherent structure behind the wake with negative vorticity. The wall effects from the pressure side of the blade are the source of this structure. The tangential velocity increases linearly in this region ($0.5 < S^* < 1$) which can be related to centrifugal force. The detached vortex from the cone and the hub vortex rotate in the opposite direction of the runner and the strong forced vortex. This counter-rotation increases the rate of dissipation in the draft tube, see figures 3c and 5. The most powerful coherent structure in the flow field is the forced vortex which is twice stronger than the detached vortex from the cone. According to the numerical results the forced vortex is up to 10 times stronger than the hub vortex (not shown here). The hub vortex and the blade wake are in the same strength but counter-rotating. Regarding the RANS simulation, the results should be treated with caution, due to the diffusivity of the turbulence model. There is weak vortices close to the wall in figure 5. The blade tip is the source of these vortices which are counter-rotating. The vortex with positive vorticity comes from the blade and the vortex with negative vorticity is the detached wall-effect from the shroud.

![Figure 2. Mean axial velocity [m/s] at the center plane. The experimental data is available at cross-sections I, II and III.](image)
Figure 3. Axial and tangential mean velocity compared with experimental results at cross-section a) I b) II c) III.
Figure 4 shows the coherent structures captured by the numerical simulation in the runner and the draft tube. The structures are presented by iso-surface of $q$-criterion [16] colored by the mean pressure. The $q$-criterion is given in equation (1).

$$q = \frac{1}{2} (\Omega^2 - S^2),$$  \hspace{1cm} (1)

where $\Omega$ is vorticity and $S$ in strain rates. As it can be seen, the pressure decreases from the suction side of the blade which leads to separation of the flow from the blade. The pressure drop continues to the runner cone where minimum pressure occurs. The pressure increases in the downstream, since the swirl decreases and the cross-section of the draft tube increases. The flow field is intertwined and rich in the runner and the draft tube. The hub clearance is the source of two strong vortices, the former generated at the trailing edge and the latter generated where the blade meets the hub. As it can be seen in figure 4, the former is much stronger. Another coherent structure is the tip vortex which generated at the blade tip. The most powerful structure in the draft tube is force vortex which is highly stable. Another interesting structure which is found in this work is detached hub jet from the runner cone. There is a sudden change of angle in the surface of the runner cone, see figure 1, which causes this effects. The detached vortices from the runner cone are the second strongest coherent structure in the field. The tip vortices from the guide vanes are also captured but not reported here.

**Figure 4.** Iso-surface $q$-criterion in the runner and the draft tube colored by mean pressure [$m^2/s^2$].

**Figure 5.** Axial vorticity [$1/s$] in a plane close to cross-section I.
Figure 5 shows the axial vorticity at a horizontal cross-section close to the cone. The structures are described from the draft tube wall inwards. The outermost structures related to the tip vortices yielding streaks of positive and negative vorticity. The inner ones are the blade wake separated from the trailing edge of the runner blade. Further inwards, there are counter-rotating hub vortices which surround another counter-rotating structure. These vortices are detached from the runner cone. The central forced vortex is surrounded by the detached vortices, dissipates them quickly and will be dissipated in the downstream (not shown here). Because diffusive turbulence model is used, these interactions between counter- and co-rotating structures are overestimated. The structures lose their coherence fast, which lead to less interaction in the downstream. The overestimate of the velocity components in the downstream, figure 3, can be related to the diffusivity of the turbulence model. To investigate this issue, advanced numerical simulation of the current test case using hybrid RANS-LES is under way. Javadi and Nilsson [17] investigated various hybrid RANS-LES method in a swirl generator with rotor-stator interaction and confirmed the applicability of the method. Javadi and Nilsson [18] applied a scale-adaptive method for strongly swirling flows. I.e. there are a number of advanced methods to study the current flow field comprehensively.

Figure 6 shows the phase-averaged pressure on a point close to the wall at cross-section I. The horizontal axis is normalized by the runner rotation period. The frequency of the forced vortex is five times larger than the runner rotation which is equal to one fifth of a complete runner revolutions. The plot includes 5 revolutions with 30 blades time periods.

Figure 7 shows streamlines in the draft tube colored by instantaneous velocity magnitude. As it can be seen the rate of the swirl is not very high, as expected. The on-axis forced vortex is shifted to the right after the elbow. The flow mainly passes close to the inner wall of the draft tube elbow.

![Figure 6. Phase-averaged pressure \( \frac{m^2}{s^2} \) over one period of the forced vortex.](image)
4. Conclusion
A numerical simulation of the U9 model Kaplan turbine using RNG $k$-$\varepsilon$ is presented. The computational domain includes the guide vanes, full runner and the draft tube. The numerical results present reasonable agreement with the measured data. The forced vortex in the draft tube shows the frequency 5 times smaller than the runner frequency. The tip and hub vortices are captured, with good agreement with experimental results. Owing to the diffusivity of the numerical modeling, more advanced (hybrid RANS-LES) simulations are underway, first, to verify the validation of current numerical results and second, to clarify the ambiguities about the life cycle of the coherent structures and the effect of geometrical details.

Acknowledgement
The research presented was carried out as a part of the “Swedish Hydropower Centre – SVC”. SVC is established by the Swedish Energy Agency, Elforsk and Svenska Kraftnät together with Luleå University of Technology, The Royal Institute of Technology, Chalmers University of Technology and Uppsala University, www.svc.nu.

The computational facilities are provided by C³SE, the center for scientific and technical computing at Chalmers University of Technology, and SNIC, the Swedish National Infrastructure for Computing.

References
[1] Drtina P and Sallaberger M 1999 Hydraulic Turbines-Basic Principles and State-of-the-Art Computational Fluid Dynamics Applications Proceedings of the Institution of Mechanical Engineers, Part C, 213, 85102
[2] Nilsson H 2002 Numerical Investigations of Turbulent Flow in Water Turbines Ph.D Thesis (Sweden: Chalmers University of Technology)
[3] Dixon S L 2014 Fluid mechanics and thermodynamics of turbomachinery, 5th ed Elsevier Inc
[4] Munten S, Nilsson H and Susan-Resiga R 2009 3D Numerical Analysis of the Unsteady Turbulent Swirling Flow in a Conical Diffuser using Fluent and OpenFOAM, in Proc. of the 3rd IAHR International Meeting of the Workshop on Cavitation and Dynamic Problems in Hydraulic Machinery and Systems, (Brno, Czech Republic), 155 – 164.
[5] Ciocan G, Iliescu M, Vu T C, Nennemann B and Avellan F 2007 Experimental Study and Numerical Simulation of the FLINDT Draft Tube Rotating Vortex ASME J Fluid Eng, 129, 146-158
[6] Bosioc A I, Resiga R, Muntean S and Tănăsă C 2012 Unsteady Pressure Analysis of a Swirling Flow with Vortex Rope and Axial Water Injection in a Discharge Cone, ASME J Fluid Eng, 134(8), 081104, 1-11
[7] Tănasă C, Resiga R S, Muntean S and Bosioc A 2013 Flow-Feedback Method for Mitigating the Vortex Rope in Decelerated Swirling Flows, *ASME J. Fluids Eng.*, **135**(6), 061304, 1-11

[8] Resiga R, Muntean S, Hasmatuchi V, Anton I and Avellan F 2010 Analysis and Prevention of Vortex Breakdown in the Simplified Discharge Cone of a Francis Turbine *ASME J Fluids Eng.*, **132**(5), 051102-15

[9] Nennemann B, Vu T C, Farhat M 2005 CFD prediction of unsteady wicket gate-runner interaction in Francis turbines: A new hydraulic design procedure, *HYDRO 2005 International Conference and Exhibition*, (Villach, Austria)

[10] Wu Y, Liu S, Dou H, Wu S and Chen T 2012 Numerical Prediction and Similarity Study of Pressure Fluctuation in a Prototype Kaplan Turbine and the Model Turbine, *Computers & Fluids*, **56**(15), 128-142

[11] Liu S, Li S, and Wu Y 2009 Pressure Fluctuation Prediction of a Model Kaplan Turbine by Unsteady Turbulent Flow Simulation, *ASME J Fluids Eng.*, **131**(10), 101102-101102-9

[12] Mulu, B. An experimental and numerical investigation of a Kaplan turbine model, PhD thesis, (Sweden: Lulea University of Technology)

[13] Beaudoin M and Jasak H 2008 Development of a Generalized Grid Interface for Turbomachinery Simulation with OpenFOAM, *Open source CFD International conference* (Berlin, Germany)

[14] Nilsson H, Page M, Beaudoin M, Gschaider B and Jasak H 2008 The OpenFOAM turbomachinery working-group and conclusion from the turbomachinery session of the third OpenFOAM workshop, IAHR, *24th symposium on hydraulic machinery and system* (Foz do Iguassu, Brazil)

[15] Alekseenko V, Kuibin A and Okulov L 2007 Theory of Concentrated Vortices: An Introduction, *Springer*

[16] Hunt J C R, Wray A and Moin P 1988 Eddies, stream, and convergence zones in turbulent flows. *Center for Turbulence Research Report CTR-S88*

[17] Javadi A and Nilsson H 2014 LES and DES of swirling flow with rotor-stator interaction, *in Proc. of the 5th Symposium of Hybrid RANS-LES method* (Texas A&M University, USA)

[18] Javadi A and Nilsson H 2014 A comparative study of scale-adaptive and large-eddy simulation of highly swirling turbulent flow through an abrupt expansion, *27th IAHR Symposium on Hydraulic Machinery and Systems* (Montreal, Canada) (submitted)