Runner Cone Separation in Kaplan Turbines

Philipp Seibert and Michel J. Cervantes

Division of Fluid and Experimental Mechanics, Luleå University of Technology
SE-971 87, Luleå, Sweden, philipp.seibert@ltu.se, michel.cervantes@ltu.se

Abstract

Low head machines of the Kaplan type are composed of several components to ensure high efficiency. Downstream the runner, a draft tube ensures pressure recovery of the flow leaving the runner. Modern draft tubes are mainly composed of a conical diffuser followed by an elbow and a straight diffuser. The runner cone, attached to the runner, is included in the conical diffuser and thus counter-acts the function of the draft tube since adding angular momentum to the fluid. Variation of the runner cone angular velocity may increase draft tube pressure recovery by delaying eventual separation. While previous research focused only on the draft tube flow, this work presents calculations including runner and draft tube in order to point out possible joint effects. The initial calculations with $\omega^*=-1$ are validated by comparison of mean and angular resolved axial and tangential velocity with LDA measurements. Two runner cone angular velocities are simulated: $\omega^*=-1$ (counter-rotating) and $\omega^*=0$ (stationary). A significant efficiency increase is obtained for a stationary runner cone.

Keywords: Kaplan Turbine, Runner Cone, Boundary Layer

1. Introduction

Turbines of the Kaplan type are used for low heads, up to 60 m. A distinctiveness of low head machines is the large amount of kinetic energy relative to the total head leaving the runner. The draft tube, a diffuser found immediately after the runner is used to recover most of this kinetic energy and is therefore of fundamental importance for high overall hydraulic efficiency, see figure 1. In the draft tube, the runner cone region, found between cross-section Ia and Ib in figure 1, plays a central role. About 70% of the overall pressure recovery occurs near the runner cone. The flow in this region needs to be attached to the draft tube wall and runner cone to ensure optimal pressure recovery. An earlier separation in the runner cone region leads to a poor pressure recovery since conditioning the flow in the rest of the draft tube but also upstream.

![Fig. 1 Draft tube model used for the Turbine-99 workshops, Engström el al. [1]](image)
The runner cone is attached to the runner and thus rotates faster than the fluid. Angular momentum is added to the fluid by the runner cone. This is an inconsistency since the runner cone is found in the draft tube, which function is to convert the remaining flow kinetic energy leaving the runner into pressure. The angular momentum added to the fluid promotes separation on the runner cone and thus a potential poor pressure recovery. Cervantes [2] investigated numerically the effect of the runner cone angular velocity on a Kaplan draft tube model; the Turbine-99 test case [3]. The author obtained a substantial gain on the draft tube pressure recovery (7%) by contra-rotating the runner cone. A clear delay of the separation point on the runner cone was obtained. The simulations were performed from section Ia, found just below the runner blades see figure 1, with constant inlet boundary conditions. Consequently, potential effects of the runner cone angular velocity on the runner-draft tube interaction could not be investigated. The runner-draft tube interaction is fundamental in low head machines. The effects of a runner cone angular velocity should be investigated with the runner and draft tube.

The work presented herein investigated the influence of the runner cone angular velocity on the Turbine-99 test case; runner and draft tube are included. The same operating point as proposed to the 3rd Turbine-99 Workshop is used, i.e., 60% load. Simulation with a standard runner cone angular velocity is validated against mean and angular resolved experimental velocity measurements performed by Andersson [4]. Identical setup with a variable runner cone angular velocity is then presented.

2. Test Case

The model investigated is known as the Turbine-99 (T99) test case. The T99 draft tube has been experimentally investigated by Andersson [4] and Lövgren [5]. The numerical investigation has been performed by a large amount of authors and presented in workshops, conferences and journals. In the present work, a stage calculation is performed. The domain starts at the trailing edge of the guide vanes and include one of the five runner blades and the draft tube, see figure 2.

![Fig. 2 Numerical domain of investigation.](image)

2.1 Geometry and Grid

The geometry investigated is presented in figure 2. The hub clearance was found of importance to get reliable results with the T99 test case, Nilsson [6]. Nilsson used a hub clearance at the trailing edge of the runner blades. The hub clearance ensured enough momentum in the runner cone boundary layer to delay separation. The hub clearance was unknown for this work. Assuming a constant hub clearance, a sensitivity analysis was performed. A value of 2 mm was found appropriate after comparison with the experimental results from Andersson [4] and used for all the simulations. The shroud clearance was set to zero.

The grid used for the simulation was composed of 2 domains. The first domain starts at the trailing edge of the guide vanes and ends at the beginning of the draft tube elbow. This configuration is unusual for Kaplan runner simulation. Generally, a general grid interface (GGI) is found between the hub and the runner cone. The present configuration was chosen to avoid any filtering of the blade wakes by the GGI present between both domains. The second domain started at the beginning of the draft tube elbow and ended at the draft tube outlet. The first and second domains were composed of 2.7 and 1.1 million hexahedral cells, respectively. The minimum angle was 20.1 degrees and was found in the draft tube elbow. Maximum aspect ratio was 64 and was also found in the draft tube after the elbow. The mesh near the wall was designed for the use of wall function: y⁺ values ranged from 2 to 385 for the different simulations.
Table 1 Mesh description

<table>
<thead>
<tr>
<th>Domain</th>
<th>Number of elements</th>
<th>Minimum Angle [°]</th>
<th>Maximum Aspect Ratio [-]</th>
<th>Maximum y⁺ [-]</th>
<th>Minimum y⁺ [-]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Domain 1</td>
<td>2 725 024</td>
<td>29.3</td>
<td>56</td>
<td>385</td>
<td>8</td>
</tr>
<tr>
<td>Domain 2</td>
<td>1 135 304</td>
<td>20.1</td>
<td>64</td>
<td>144</td>
<td>2</td>
</tr>
</tbody>
</table>

2.2 Boundary Conditions and Turbulence Modeling

The boundary conditions proposed to the 3rd Turbine-99 workshop were used in the calculation, i.e., a flow rate Q=0.522 m³/s. A mass flow rate with a specific direction was applied since the guide vanes were not included in the simulation. The guide vanes angle on the model was 28 degrees. For the simulation an inlet flow angle of 30 degrees was chosen after a sensitivity analysis ranging from 28 to 32 degrees.

All walls were set to smooth. Domain 2 containing the elbow and straight diffuser was set stationary, while domain 1 was rotating. The outlet boundary condition was set to opening allowing the fluid to flow in both directions. A GGI connected the rotating domain 1 to the stationary domain 2. The hub and blades were rotating with the domain at an angular speed ω₀=595 rpm. The runner cone angular velocity, characterized by ω₉₀=ωω₀, was varied for different simulations to quantify its effects. The runner cone rotated with the domain angular velocity for ω₉₀=1, was stationary for ω₉₀=0 and counter-rotated for ω₉₀=-1. The domain inlet was stationary. A GGI linked the meshes in the hub tip. Other regions were connected 1:1 by rotational periodicity.

For modeling turbulence, the Shear Stress Transport Model (SST) was used. It uses the k-ω model in the inner boundary layer, while the outer region is predicted by the k-ε model.

3. Results

The results are presented in two sections. The first section is dedicated to the validation of the model, i.e., simulations with an identical setup as the T99 model test are compared to detailed laser Doppler anemometry (LDA) velocity measurements performed by Andersson [4]. The second section presents the influence of the runner cone angular velocity on the main engineering quantities: relative pressure recovery and relative efficiency. A qualitative description of the flow function of the runner cone angular velocity is presented.

3.1 Validation

The validation is performed by comparing axial and tangential velocity obtained from the simulations to experimental data at two radial profiles, Ia and Ib downstream the runner, see figure 1. The results are made dimensionless with the bulk velocity. The position of these profiles was defined by Anderssson [4].

Mean axial and tangential velocity profiles obtained from the simulations are compared to the mean experimental values. A qualitative good agreement is achieved, see figure 3. The lower velocity in the calculation close to the shroud in section Ib is attributed to the shroud clearance not considered in the present calculations, see figure 4. A separation region is obtained in the simulation not present in the experiment. The tangential velocity is slightly underestimated at Ia, while the tangential velocity close to the hub is not captured at the correct place. Once again, a deficit of the velocity profile is obtained near the shroud. The peaks at r=0.3 in section Ib are influenced by the hub tip and the mesh quality with thin elements at the trailing edge.

![Fig. 3 Mean axial and tangential velocity profile at Section Ia](image-url)
More details of the axial and tangential velocity at section Ia and Ib are presented in figures 5 and 6. The blade wakes are still present at section Ib. The missing blade shroud tip has a noticeable impact on the simulations when compared to the experimental results, especially for the tangential velocity.

A qualitative good agreement between calculation and measurement can be detected for the complete plane. Quantitatively, the gradients in the calculation are slightly sharper. This can be seen by larger areas with higher speed and bigger areas with lower speed respectively, which occur due to the dense mesh while the measurements were interpolated between two measurement points with a larger distance.

In the simulation results, see figure 3, the axial velocity was over predicted close to the shroud (r=0.9). This is also shown in the whole plane, see figure 5. In contrast, the tangential velocity was slightly lower in the calculation compared to the experiments close to the shroud (~0.2 m/s in profile Ia, see figure 3), which is confirmed for both planes by lower velocities close to the shroud compared to the experiment.
The influence of shroud clearance on the boundary layer downstream the shroud as well as the influence of the hub clearance design on the boundary layer around the runner cone is obvious and has to be considered in further research.

The model is now used for an assessment of the flow including draft tube and runner with different angular velocities.

3.2 Runner cone angular velocity

After validation of the model, the runner cone rotation speed was varied. In this calculation, a positive effect of a stationary runner cone as in [2] could be confirmed. The engineering quantities, relative efficiency (calculated from the domain inlet to the domain outlet) and draft tube pressure recovery (calculated from section Ia to the domain outlet), are presented in Table 2. The engineering quantities increased for $\omega^* = 0$ but decreased for $\omega^* = -1$, becoming lower than for the reference case where $\omega^* = 1$. The efficiency increase is very large, about 2%. This result is only indicative since inlet boundary conditions are approximate, hub-blade clearance is guessed and the blade-shroud clearance is not included in the calculation.

Table 2 Engineering quantities

<table>
<thead>
<tr>
<th></th>
<th>$\omega^* = 1$</th>
<th>$\omega^* = 0$</th>
<th>$\omega^* = -1$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$c_p$</td>
<td>1</td>
<td>1.042</td>
<td>0.884</td>
</tr>
<tr>
<td>$\eta$</td>
<td>1</td>
<td>1.019</td>
<td>0.971</td>
</tr>
</tbody>
</table>
The mean axial and tangential velocity profiles at section Ia are presented in figure 7 for different dimensionless angular velocity. The effects of the runner cone angular velocity different from the runner velocity are localized near the hub as expected and significant. Therefore, the runner should be included in the simulation when analyzing potential impact of Kaplan runner cone angular velocity.

The surprising results is the efficiency drop for $\omega^*=-1$. As a matter of fact, the flow is very similar to $\omega^*=0$ in figure 7. Vector plot in the draft tube mid-plane show a strong separation near the shroud and in the elbow for $\omega^*=-1$. The separation near the shroud is attributed to the missing blade-shroud clearance. These strong separation regions deteriorate the draft tube pressure recovery and de facto the overall machine efficiency. For $\omega^*=1$, an early separation on the runner cone compared to the experience is captured. Such result was also obtained by Cervantes [2] when simulating only the draft tube.

**Fig. 7** Axial and tangential velocity distribution at section Ia for different runner cone angular velocity

**Fig. 8** Vector plot in the draft tube mid-plane for different runner cone angular velocity: $\omega^*=1$ (left), $\omega^*=0$ (middle) and $\omega^*=-1$ (right)

### 4. Conclusion

CFD calculations of the Turbine-99 test case were performed. In contrast to former calculations, the runner is included. The connection to the draft tube is achieved by a GGI at the beginning of the elbow instead of a GGI below the runner in order to avoid a filtering of the blade wakes. The model was evaluated by comparison to experimental measurements. The calculated velocities matched the experimental data pretty well. The difference can be explained by the approximate inlet boundary conditions as well as the lack of a shroud clearance.

Positive effects of a stationary runner cone were confirmed, while a counter rotating runner cone had a negative global effect due to a strong separation on the draft tube shroud. Further research has to be performed on the mesh quality and the geometry as the implementation of the shroud tip.
Acknowledgments

The research presented was carried out as part of “Swedish Hydropower Centre - SVC”. SVC has been 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 authors would like to thank for the financial support and wish to thank the people at our department that have provided help and good ideas.

Nomenclature

\begin{align*}
U & \quad \text{Axial velocity [m/s]} \\
V & \quad \text{Radial velocity [m/s]} \\
V_x & \quad \text{Velocity in the x direction [m/s]} \\
V_y & \quad \text{Velocity in the y direction [m/s]} \\
V_z & \quad \text{Velocity in the z direction [m/s]} \\
W & \quad \text{Tangential velocity [m/s]} \\
\omega & \quad \text{Angular velocity [rad/s]} \\
\omega^* & \quad \text{Dimensionless angular velocity [-]} \\
\eta & \quad \text{Relative efficiency [-]} \\
c_p & \quad \text{Relative pressure recovery [-]} \\
r & \quad \text{Dimensionless radius [-]}
\end{align*}

References