Numerical simulation and analysis at partial load in Francis turbines: Three-dimensional topology and frequency signature of inter-blade vortices

To cite this article: F Doussot et al 2019 IOP Conf. Ser.: Earth Environ. Sci. 240 022045

View the article online for updates and enhancements.
Numerical simulation and analysis at partial load in Francis turbines: Three-dimensional topology and frequency signature of inter-blade vortices

F Doussot1, G Balarac1, J Brammer2, O Métais1, C Séguoufin2
1Univ. Grenoble Alpes, CNRS, Grenoble INP, LEGI, 38000 Grenoble, France
2GE Renewable Energy, Hydro Solutions, 82 Avenue Léon Blum, 38041 Grenoble Cedex 9, France
E-mail: francois.doussot@univ-grenoble-alpes.fr

Abstract. Hydraulic machines are designed to operate in flow conditions close to the best efficiency point. However, to respond to the increasing demand for flexibility mainly due to the integration of renewable energy in the electric grid, the operating range of Francis turbines has to be extended towards smaller discharge levels without restriction. When Francis turbines are operated typically between 30% and 60% of the rated output power, the flow field is characterized by the appearance of inter-blade vortices in the runner. In these off-design operating conditions and due to these phenomena, dynamic stresses level can increase, and potentially lead to fatigue damage of the mechanical structure of the machine. The objective of this paper is to present investigations on the dynamic behaviour of the inter-blade vortices and their impact on the runner by using numerical simulations. Computations were performed with different turbulence modelling approaches to assess their relevance and reliability: Reynolds-Averaged Navier-Stokes (RANS) and Large-Eddy Simulation (LES). Computations aimed to better understand the emergence condition of the inter-blade vortices. The analysis showed that vortices can be generated due to poor inlet adaptation at part load, however other vortices can also be due to a local backflow in the runner. The competition between these both phenomena leads to various topologies of the inter-blade vortices. The numerical results were compared to experimental visualizations performed on scaled model as well as to previous numerical studies results. The impact of these inter-blade vortices on the runner were also investigated by considering the pressure fluctuations induced on the blades. The dynamic loading on the blade has to be known in order to evaluate the lifetime of the runner by mechanical analysis. A previous experimental study [S. Bouaïja et al., IOP Conf. Ser.: Earth Environ. Sci., 2016] has shown that the appearance of the inter-blade vortices can be correlated with a large-band frequency signature in the pressure fluctuations measured on the blades. The numerical simulations presented in this paper focused on the prediction of this frequency signature as well as on the analysis of its origin.

1. Introduction

The recent development of new renewable energy technologies, such as wind and solar, leads to an uncontrolled amount of energy dispatched. The need for flexibility is brought by their integration in the grid. Supply and demand has to be kept in balance in real time to guarantee electric grid stability [1].

Hydroelectric power therefore has a major advantage in the regulation of the power supply
compared to other technologies. Hydro energy sources can respond in few seconds. But flexibility is not only a matter of time response, but also a matter of amount of energy produced. For hydroelectric power, this flexibility can be brought by operating in a wide range of flow conditions, especially using flow rates smaller than usual. But these off-design operating conditions can lead to the appearance of secondary flow phenomena. This turbulent flow field could generate pressure fluctuations in the machine. This paper deals with Francis turbines, which are the most widely used type of hydro turbine with about 60% of the global hydropower capacity in the world.

In part load conditions, a vortex rope can be seen in the draft tube cone. For lower discharges, with an output power roughly between 30% and 60% of the nominal value, inter-blade vortices can appear in the runner. Under certain conditions still not fully understood, an increase in pressure fluctuations in the runner can be observed, which could cause fatigue damage on the mechanical structure.

Inter-blade vortices have been studied for several years using numerical and experimental approaches, however the sources and the driving phenomena of the vortices have not been fully understood. Away from the design point, the flow field entering the runner is misaligned with the geometry of the blades. Wack [2] suggests that this poor incidence angle generates the inter-blade vortices. Zhou et al. [3] also studied the occurrence and the location of the vortices depending on the operating point. According to their study, the mechanism of this phenomenon is driven by the reverse flow near the hub and is not related to the incidence angle. Yamamoto [4] presents similar results: a numerical simulation shows a significant backflow region at the center of the draft tube cone. Flow analysis in the runner shows that this backflow region plays a major role in the formation of the inter-blade vortices. In another numerical investigation, Zhou et al. [5] show with RANS computations and k-ω SST model that the vortex related to the reverse flow is the most prone to cavitation. Moreover this article suggests a classification of the inter-blade vortices appearing in the runner into four types. According to this study, they can be due to the incident angle, to a cross-flow occurring on the blade pressure side, to the geometry the junction between the leading edge and the shroud, or finally to the reverse flow.

These off-design conditions can increase dynamic stresses on the blades. S. Bouajila [6] has shown that these flow regimes are characterized under certain operating conditions by pressure fluctuations on the blades with a high and wide-band frequency signature. This hydraulic phenomenon could potentially cause fatigue damage and reduce the lifetime of the turbine. However this frequency signature has never been predicted by numerical simulations and its origin is still not well understood.

The analysis presented in this paper was conducted on a 17 blades low head Francis turbine, with a distributor composed by 24 guide vanes and 24 stay vanes. An experimental study was previously performed in the GE Hydro Laboratory [7]. The visualisation of the cavitating vortices and on-board pressure measurements have been compared with the numerical results.

The first part of this paper describes investigations on the topologies and causes of the vortices thanks to steady simulations. In a second part, the wide-band frequency signature is analysed considering pressure fluctuations on the runner. Large eddy simulations (LES) were used in order to resolve the turbulent length scales which could generate this characteristic frequency signature.

2. Characterization of the inter blade vortices using steady simulations

RANS simulations were computed using the solver CFX to study the mean flow of a wide range of operating conditions thanks to their low computational cost.

Operating points were described by their reduced velocity $n_{111} = \frac{nD}{\sqrt{H}}$ and the reduced discharge $Q_{11} = \frac{Q}{D^2\sqrt{H}}$. $Q$ is the flow rate, $D$ the outlet diameter of the runner, $n$ the runner
speed and $H$ the net head. A reduced discharge can be defined by $\phi = \frac{V_m}{U}$ where $V_m$ and $U$ are respectively the meridional and the circumferential velocity at the outlet surface of the runner. $\phi$ is normalised by its value at the best efficiency point, defining the non-dimensional value $\bar{\phi}$.

2.1. Numerical setup
The computational domain has been reduced to a single blade and guide-vane channel using periodic boundary conditions to reduce the simulation time. The draft tube was not included in the simulation. However, as part load operating conditions are characterized by a back-flow in the outlet of the runner, in order to avoid a strong interaction between the outlet and the recirculation zone the cone section was virtually extended. Furthermore, to ensure model convergence, a constant flow rate was used as the outlet boundary condition and a constant total pressure set at the inlet. The Shear-Stress-Transport turbulence model has been chosen. Initial tests showed a strong mesh sensitivity. Therefore, a refined structured mesh of 2.3 million nodes was used to correctly discretise the vortices, leading to a dimensionless wall distance ($y^+$) around 2 in the runner, with a maximum of 5 near the shroud.

The value $y^+$ has to be close to 1 to resolve the boundary layer in order to estimate correctly the losses. The operating points are characterized by their $n_{11}$ and $Q_{11}$ values. As $n_{11}$ and $Q_{11}$ values depend on the head, which is a result of the simulation, an iterative method has been used. The net head was estimated using the results of the CFD calculations for the runner torque and head loss, combined with assumed analytical values for the other components not included in the simulation.

2.2. Validation of the model
The numerical results were validated using the experimental data. A good agreement on the torque was reached, with an over-estimation between 5 to 10% of the torque. This over-estimation could be expected because some experimental losses were not considered in the simulation, such as the losses due to leakage flow through labyrinths.

Assuming that the presence of cavitation would not change the location of the vortices, experimental and numerical results can be compared. The CFD simulations in this study used a single-phase model. Therefore, the vortices have to be identified using a suitable criterion. Among the criteria suggested in [8], an iso-surface of Q-criterion has been used to identify coherent vortices, because this criterion is particularly adapted to the identification of large scale vortices in a turbulent flow.

![Iso-Q-criterion](image1.png)  ![Experimental cavitation visualisation](image2.png)

Figure 1. Comparison between numerical and experimental results at OP1.
The results show a good prediction of the location of the vortices, as shown on figure 1 and 2. The corresponding operating points are identified on a reduced hill chart in figure 4. The location of the cavitating vortex depends on the operating condition. It can be attached to the leading edge such as in figure 1, or to the hub as shown in figure 2. Other operating points have been compared, showing also relevant vortex location prediction.

2.3. Three types of vortices
The previous observations have shown different topologies of inter-blade vortices, as shown in figure 3. In order to understand their driving phenomena, the inter-blade vortices can be classified into three categories.

2.3.1. Reverse flow vortices: Small discharges correspond to small inertia of the fluid. Therefore the centrifugal force keeps the incident flow at the periphery of the runner. It creates a depression at the runner outlet, underneath the runner tip, leading to a large recirculation zone. This
backflow is driven on the suction side of the blades by the Coriolis force. The interaction between the incident flow and this backflow generates a shear flow, which creates the inter-blade vortex. The greater the backflow, the closer the vortex is to the leading edge. This type of vortex, shown in figure 3a, appears independently from the following vortices. This result matches one of the vortices characterized by Zhou et al. [3].

2.3.2. Incidence vortices: Deep part load operating conditions are characterized by small opening of the guide vanes. Then, the flow field at the inlet of the runner is strongly misaligned. These vortices are also already presented in the literature [5].

The RANS simulations show that the small inertia of the incident fluid has a major impact in the vaneless space. In these operating conditions, viscous forces are then not negligible and change completely the direction of the fluid. The flow angle at the leading edge is different from the one given by the guide vanes. The incident angle generates a flow separation on the suction side. As the meridional velocity in the vaneless space is the smallest closed to the hub, this flow separation appears at first closed to the hub. The Coriolis force then brings back the stalled streamlines to the suction side, generating the vortex. Therefore, this type of vortex is attached to the leading edge, as shown in figure 3b.

2.3.3. Generalized incidence vortices: This last category is a degraded form of the previous phenomenon. In this case, the flow separation appears on a large part of the leading edge and cannot be advected by the incident flow. As the separation zone creates a vertical stall region after the leading edge, the streamlines swirl around vertical axis closed to the edge. Thus this type of vortex is attached to the hub and takes a large part of the channel. Figure 3c shows this phenomena.

2.4. Which vortices for which operating condition?

According the previous classification, each type of vortex is associated to a physical phenomenon. These phenomena depend on the operating conditions of the machine, and can be identified on the hill chart. Thanks to the low cost of the RANS simulations, many operating points can be simulated. In order to explore the entire part load conditions, a wide range of \( n_{11} \) and \( Q_{11} \) values have been used. Five values of \( n_{11} \) have been chosen, using each eight values of \( Q_{11} \). Several points with lower and higher values of \( n_{11} \) have been added to extend the simulated range. The results are presented on a typical \( n_{11}, Q_{11} \) hill chart in figure 4. The limits presented here correspond to the case where the velocity streamlines begin to swirl. There is no link between these limits and the appearance of cavitation or mechanical stress.

The vortex created from the reverse flow appears for most of the part load operating points. However, for high \( Q_{11} \) values, this vortex is very small and stays close to the trailing edge. The incidence vortex is due to the poor adaptation of the fluid angle at the leading edge. It appears that this vortex is generated for low \( n_{11} \) values. These first two vortices can appear in the runner at the same time. Their structures can be merged into a single vortex in the inter-blade channel. The limit between reverse flow vortices and generalized incidence vortices is not clear. When the reverse flow is strong enough to go back to the leading edge the created vortex makes the incident flow separate from the blade. Therefore, these two types of vortices share the same topology for low \( Q_{11} \) values: a large vortex attached to the hub can be expected.

This observed behaviour is likely to differ with other blade geometries, however the exact impact of design parameters on the vortex formation is still not fully understood. RANS simulations are able to approximately evaluate the topology of the vortices according the operating point. But no information about the dynamic behaviour of the flow field in the runner, especially about the pressure fluctuations on the blade, can be brought by these simulations.
3. Large eddy simulation of the large-band frequency signature of deep part load

3.1. Numerical setup
LES can be used for such industrial test cases. The biggest scales are explicitly resolved while the smaller scales, more universal, are modelled using a subgrid scale model. LES simulations have been run in solver YALES2. It uses a finite volume method and a fourth order scheme. The incompressible solver has been chosen for this test case. The maximum Courant number is set to 1.

As in RANS simulation, a single runner channel has been modelled. Periodic boundary conditions have been used, assuming that this phenomenon only depends of the dynamic behaviour of each channel. A similar extension of the domain has been used to improve the modelling of the reverse flow in the draft tube cone. Two full tetra meshes have been used. The first one, very coarse, with 3.6M cells, had around 5 cells per vortex diameter as shown in figure 5a. The second, especially refined in the vortex location was composed of 79M cells and is shown in figure 5b. As this work is focused on the dynamic behaviour of the vortices, no wall refinement was made, leading to a $y^+$ value of about 35 in the runner.

The solution has been solved in the rotating frame. The velocity field has been imposed at the inlet boundary condition which corresponds to the outlet flow at the guide vanes exit. This velocity field has been extracted from a RANS calculation of the entire machine (guide vane, runner and draft tube), and velocity components have been averaged circumferentially.

The torque was accurately predicted with errors lower than 10% for all the operating points investigated. Pressure was monitored at discrete locations which corresponds to the on-board sensors of the experimental runner. Four pressure sensors are used here: two at the pressure side and two at the suction side of the same channel, both close to the leading edge. The numerical pressure spectra are made from 15 runner revolutions, initialized with a converged solution.

3.2. Comparison of the vortex topologies between RANS simulations and LES
LES are used to study the dynamic behaviour of the inter-blade vortices. However it also provides also information about their topologies and their localisation that can be compared.
Whilst LES and high performance computational resources enable to perform high-fidelity simulations, it generates a large amount of data making it difficult to extract relevant information regarding the large-scale phenomena. Visualisation of Q-criterion with LES simulation is more prone to show smaller structures, making the visualisation of relevant topologies very difficult. Therefore, high order filtering based on the mesh size has been used to filter the smaller vortices and to visualize only the big scales of the structure \[9\]. As LES needs a lot of computational resources, a limited number of operating points have been studied. To illustrate the results, only two points are presented here.

Comparing the RANS and LES results, the location of the vortices shown in figure 6 are similar: in both cases, the reverse flow vortex and the incident vortex merge in a single vortex close to the leading edge. However, as shown in the figures 6b and 6c, LES predict a lot of small turbulent structures stretched around the vortex.

Figure 7 presents the same comparison for a higher \(Q_{11}\) value. Steady results clearly shows the reverse flow vortex in figure 7a and did not predict any flow separation at the leading edge. LES result shows small structures emerging from this edge in figure 7b. With the filtering operation, figure 7c shows the reverse flow vortex at a similar location than the RANS calculation.

This brief comparison shows that the results are similar in terms of topology between the
3.3. Prediction of the large-band frequency signature of deep part load

The following part aims to understand experimental results obtained by S Bouajila [6]. A large band frequency signature has been detected on pressure sensors in the runner. Even if the amplitude of this phenomenon is low, as the frequency signature is wide it can correspond to a significant part of the energy of the signal. Understanding the origin of this frequency signature is key in order to provide technical solutions to operate in these flow conditions.

3.3.1. Experimental results: The following results come from a study previously performed in the GE Hydro Laboratory [7]. For \( \bar{\phi} < 0.32 \), a wide band frequency signature appears in the pressure and strain signals frequency spectra. A significant influence of the cavitation number (\( \sigma \)) has been shown experimentally. When \( \sigma \) decreases, the frequency of this phenomenon also decreases and the amplitude of the pressure fluctuations rises. However, as the numerical model uses a single-phase model, only operating conditions without cavitation are studied here.

The following part is focused on the single operating point OP4, identified in figure 4, chosen to match the maximum amplitude of the large-band frequency signature. In order to avoid issues linked to the reference pressure in the following numerical study, only the pressure differences between the two sensors located on pressure side and between the two sensors located on suction side have been used. It has been shown that using pressure differences between sensors or absolute pressure does not affect significantly the experimental frequency signature. \( f_0 \) is the rotation frequency of the runner. Figure 8 shows that the pressure fluctuations are clearly higher at the suction side, with the characteristic wide-band frequency signature around \( 35f_0 \). The aim of this part is to reproduce and predict this high frequency behaviour.

3.3.2. Results: This operating point is characterized by a reverse flow which is strong enough to generate a vortex close to the leading edge.

The high frequency signature measured experimentally is not predicted by a simulation with a constant velocity profile at the inlet. Experimental and numerical pressure frequency spectrum are presented on figure 9a. With the coarse mesh, the cut-off frequency is around \( 20f_0 \), at lower frequencies that the ones of the investigated phenomenon. Therefore this setup is not adapted to solve small turbulent length scales that generate such high frequency pressure fluctuations. However with the fine mesh, the cut-off frequency is higher than the experimental
frequency signature. But no high amplitudes were predicted around $35f_0$. Low frequencies are not predicted as well, but the model has been chosen to investigate the high frequency signature, not the low frequencies which would require an extended model including the draft tube and the distributor.

The degree of unsteadiness of the inlet boundary condition could explain that the frequency signature is not predicted. To investigate how the inlet boundary condition can generate these pressure fluctuations, turbulence injection was used. Homogeneous isotropic turbulence was used to compute a velocity fluctuation at the inlet boundary condition. A turbulent flow field, depending on a characteristic length and the value of $U_{\text{RMS}}$, is added to the initial velocity field. In order to reproduce the turbulent flow generated by the small opening of the guide vanes, the characteristic length scale has been set to the opening of the guide vanes. $U_{\text{RMS}}$ has been set to 10% of the inlet velocity assuming that it represents the turbulence intensity of the flow downstream the guide vanes. With these values, turbulence injection led to a good prediction of the phenomenon at the suction side, as shown in figure 9b. However these results show the same frequency signature, with similar amplitude at the pressure side. Thus, numerical results do not reproduce properly the experimental measurements, which show that this phenomenon is mainly located at the suction side. These results do not clearly explain the origin of this phenomenon.

![Figure 8. Frequency spectrum suction side and pressure side.](image)

![Figure 9. Pressure difference spectrum at the suction side.](image)

Other values of the turbulence injection parameters have been used to investigate their
sensibility on the pressure frequency spectra. It appeared that $U_{RMS}$ mainly influences the amplitude of the fluctuations, and the characteristic length controls the frequency of the wide-band signature.

Further investigations have shown that this frequency signature is driven by the flow rate variations generated by the turbulence injection at the inlet. Indeed, with the turbulence injection model used here, the flow rate is not constant in the computational domain. Flow rate fluctuations correspond to large scale (actually to the entire domain) variations, and can create high frequency pressure fluctuations. This explains why the coarser mesh is able to predict these pressure fluctuations shown in figure 9b.

4. Conclusion
It has been shown that RANS calculations on a simplified low head Francis turbine runner model are able to accurately predict the location and the topology of the inter-blade vortices. The vortices can be classified into three categories depending on their topologies and their driving phenomena. Each type corresponds to a flow regime which has been identified by their operating conditions. Highly resolved large eddy simulations lead to similar topologies with a higher level of detail.

Pressure fluctuations with wide band frequency signature are measured experimentally at low discharges. Large eddy simulations have been run to understand the origin of this phenomenon. Without turbulence injection at the inlet, these pressure fluctuations are not predicted by the numerical simulations in a simplified domain. It shows that the dynamic pressure fluctuations at high cavitation numbers are not necessarily generated by the inter-blade vortices, but could be caused by their interaction with other phenomena.

It appeared that this frequency signature can be roughly reproduced using turbulence injection. Even if the results did not match all the experimental data and are not sufficient to conclude about the origin of this phenomenon, it suggests that additional unsteady phenomena could explain this behaviour. Further investigations of the generation of instabilities throughout the machine are therefore needed to explain this frequency signature.

References
[1] Grunewald P and Diakonova M 2018 Energy Research & Social Science 38 58–66
[2] Wack J and Riedelbauch S 2015 Journal of Physics: Conf. Series 656 012074
[3] Zhou L, Liu M, Wang Z, Liu D and Zhao Y 2016 28th IAHR Symposium on Hydraulic Machinery and Systems 49 1030–39
[4] Yamamoto K 2017 Hydrodynamics of Francis turbine operation at deep part load condition Ph.D. thesis École Polytechnique Fédérale de Lausanne
[5] Zhou L, Liu M, Wang Z, Liu D and Zhao Y 2016 Engineering Computations 34 364–376
[6] Bouajila S, De Colombel T, Lowys P and Maître T 2016 28th IAHR Symp. on Hydraulic Machinery and Systems 49 1010–19
[7] Bouajila S, Brammer J, Flores E, Séguin C and Maître T 2017 Advances in Hydroinformatics 931–945
[8] Zhang Y, Liu K, Xian H and Du X 2018 Renewable and Sustainable Energy Reviews 81 1269 – 85
[9] Guedot L, Lartigue G and Moureau V 2015 Physics of Fluids 27 045107