Numerical and experimental investigation of the runner channel vortex in Francis turbines regarding its dynamic flow characteristics and its influence on pressure oscillations

M V Magnoli, D Anciger, M Maiwald
Voith Hydro Holding GmbH & Co. KG, Alexanderstraße 11, 89522, Heidenheim, Germany
E-mail: Marcelo.Magnoli@voith.com

Abstract. With the increasing demand for more flexibility in the operation of hydraulic machines, the operation range of Francis turbines has been progressively extended towards deep part load, where the channel vortex is present in the runner. Its set-in is often associated with increased pressure oscillation amplitudes in the vaneless space and runner. The runner channel vortex (RCV) is commonly observed during model test at reduced flow conditions. However, there is limited research available on its fluid dynamic fundamentals, on its origin, on its transition from incipient to fully developed vortical structure and on its relation to pressure oscillations.

In order to acquire deeper understanding of the runner channel vortex phenomenon, computational fluid dynamic (CFD) simulations of the flow through the complete turbine were carried out. Several operating points along a given hill chart cross-section, starting from full load, passing through the optimum and part load and achieving deep part load, were calculated with the aim of observing the set-in and evolution of the runner channel vortex. The transient simulations made use of hybrid turbulence models, as scale adaptive simulation (SAS), to capture the fine dynamic flow characteristics with adequate accuracy.

The numerical simulations can give some insight on the basics and fluid patterns related to the channel vortex, which cannot be directly observed at the model test rig. Nevertheless, the model test can deliver important information about the pressure oscillation amplitude in the hydraulic machine when the channel vortex is present. Moreover, special data of pressure oscillations at the model runner is available for selected projects. The test data offers the possibility to draw conclusions about the influence of the channel vortex on the pressure oscillation amplitude and delivers values for the validation and calibration of the CFD simulations.

The numerical simulations and model test were done for a mid-specific speed Francis machine, for which pressure oscillation amplitudes were also measured at the model runner.

1. Introduction
In recent years Francis machines experienced the extension of their operating range in order to satisfy the energy market requirements for regulating services and to maximise the owner benefits. The operating range was often extended towards reduced power and flow, leading to turbine operation at part load and deep part load. At these operating conditions significantly pronounced dynamic flow effects become present, respectively rotating vortex rope in the draft
Figure 1. Flow patterns in the turbine hill chart regions associated to dynamic effects.

tube cone (DTI) and runner channel vortex (RCV), as seen in Figure 1. Part load and its associated rotating vortex rope were investigated by some in the past, as e.g. by Magnoli [1]. Regarding deep part load and its associated runner channel vortex there is rather few research done.

With the objective to acquire more knowledge about the runner channel vortex, numerical and experimental investigations were carried out. The numerical fluid flow simulation had the objective to understand the phenomenon, identify the set-in of the runner channel vortex, follow its development and characterise its structure and topology. The dynamic fluid flow associated to the runner channel vortex at deep part load causes the variation of the pressure field at the runner, which is at the origin of pressure oscillations. The determination of the pressure oscillations amplitude was investigated numerically but more extensively experimentally. With an instrumented model runner equipped with several dynamic pressure transducers, the pressure oscillation field could be experimentally measured.

The numerical and experimental investigations were performed for a mid-specific-speed Francis turbine with $n_{opt} \approx 60 \text{ min}^{-1}$. The numerical and experimental setups and results are presented for this machine, named here as FT 60. This turbine offered interesting test cases, due to the available data and wide operating range, with the head ratio $H_{max}/H_{min} \approx 1.8$ and lower power limit at $P/P_{max} \approx 0.60$.

2. Setup
2.1. Numerical Setup
The numerical simulation offers the benefits of determining the complete pressure and velocity field at the complete fluid flow domain. Experimental methods can deliver pressure and velocity measures at limited number of singular locations. Often during model testing the number of pressure transducers and their applicability are limited. The experimental determination of the
The application of this method for Francis runners is extremely limited due to the only partial optical accessibility of the narrow and elongated runner channels through the model Plexiglas draft tube cone. The visual observation of the runner channel vortex during the model test is also restricted to reduced optically accessible regions and to cavitating conditions. For these reasons, the numerical fluid flow simulation could offer more detailed information about the runner channel vortex topology.

The numerical simulations were carried out with the finite volume method (FVM) applied to the complete turbine with all its hydraulic components: spiral case, stay vanes, guide vanes, runner and draft tube. The calculations were conducted in a coupled manner, i.e. considering all the turbine components and interfaces between them, as long as the dynamic effects in the fluid flow through the turbine arise from the interaction between its components, caused by the rotating runner and the stationary parts.

The computational mesh counted with around 18 million cells and part of it can be seen in Figure 2. The employed FVM made use of second-order interpolation schemes for the time dependent, convective and diffusive terms.

The computational domain was artificially extended at the spiral case inlet and draft tube outlet with the objective to eliminate any boundary effects on the turbine inlet and outlet sections. As inflow boundary condition the volume flow was prescribed together with 5% turbulence intensity. At outlet a reference integral pressure level was prescribed and allowed any eventual backflow.

For the simulation of the transient effects in the fluid flow through the turbine, like the rotor-stator interaction and the flow instabilities in the draft tube, adequate time resolution and turbulence models were needed. Approximately 400 time steps were calculated for each machine revolution, in order to capture not only the effects from the draft tube instabilities, but also from the rotor-stator interaction and in the runner channel.

Reliable turbulence models are of main importance for the precise calculation of transient fluid phenomena such as the rotating vortex rope in the draft tube cone and the runner channel vortex. For the numerical simulations of the transient fluid flow through the FT 60 turbine, the scale adaptive model (SAS) from Menter and Egorov [2] was the chosen turbulence model. As investigated by Magnoli [1, 3], Magnoli and Schilling [4, 5], Hasmatuchi [6], Benigni et al. [7] and...
Wunderer [8], hybrid turbulence models deliver accurate numerical solutions for the transient fluid flow through hydraulic turbines with currently acceptable computational costs.

Each transient simulation required approximately over 30 machine revolutions, depending on the operating point and on the prescribed initial solution, until the stable transient flow patterns were established. The computation of one machine revolution took, in average, 12 hours in a Linux cluster with 72 parallel threads running at Intel Xeon X5680 processors with 3.33 GHz and 24 GB memory.

The numerical model described and employed here was previously validated by Magnoli [9] using available experimental data.

3. Experimental Setup
For the FT 60 Francis turbine, transient pressure measurements were performed at the model runner blades and crown. The model turbine geometry was hydraulic homologous to the prototype machine, including the instrumented runner blades.

A total of 28 dynamic pressure transducers were mounted on the runner blades and band. Four rows from crown to band with three sensors on the blade pressure side (PS) and another three on the blade suction side (SS) were installed along the blade length, going from the trailing edge pressure side to the trailing edge suction side. Additionally four sensors were inserted along the band. The location of the pressure transducers at the blade can be seen in Figure 3. The physical installation of the dynamic pressure transducers at the model runner blades and of the wiring is shown in Figure 4.

The sensors are numbered from 1 to 28. Their location at the blades is defined by the pair of coordinates \((u, v)\). Both coordinates are normalised from 0 to 1. The \(u\) coordinate starts at the trailing edge on the pressure side, \(u = 0.00\), and runs to the trailing edge on the suction side \(u = 1.00\). The \(v\) coordinate runs from crown to band, going from \(v = 0.00\) to \(v = 1.00\). The transducers were placed at the runner blades at the coordinates, \(u = 0.05, u = 0.25, u = 0.40, u = 0.60, u = 0.75, u = 0.95\) and \(v = 0.20, v = 0.40, v = 0.60, v = 0.80\).

For the data transmission between the dynamic pressure transducers in the rotating runner to the data acquisition system a telemetry system was used. Part of the electronic was mounted inside the runner crown, as seen in Figure 5. The wiring was routed through the inner part of the turbine shaft up to its top, where the transmitting part of telemetry system was mounted, as observed in Figure 6.

4. Numerical Simulation Results
4.1. Operating Points
The dynamic fluid flow through the FT 60 turbine was simulated at three relevant heads for the prototype machine. The considered heads were the maximum \(H_{\text{max}}, \frac{n'_1}{n'_{1\text{opt}}} = 0.9900\), rated \(H_{\text{rated}}, \frac{n'_1}{n'_{1\text{opt}}} = 1.1062\), and minimum \(H_{\text{min}}, \frac{n'_1}{n'_{1\text{opt}}} = 1.3380\). For each head seven flow rates were applied, corresponding to power outputs starting from 100% and going down to 40% of the maximum power at the respective head, \(0.40 \leq P/P_{\text{max}}(H) \leq 1.00\), in 10% steps. With the flow reduction from full to deep part load, it was possible to observe the set-in and evolution of the runner channel vortex.

4.2. Vortex Determination Methods
The vortex identification and visualisation in the fluid domain might be done with multiple approaches. Analogous to the model test observation when the cavitating vortex can be seen, the vortex can be shown in the simulation results with a pressure isosurface, i.e. isobar surface. However, this method is limited to vortex regions where the local pressure drops below the water vapour pressure. This approach is suitable for qualitatively comparison of numerical simulation
results with experimental observations. It is not applicable for the basic research of the runner channel vortex, since the vortex as velocity field might be present without necessarily being cavitating.

More appropriate for the runner channel vortex investigation are the velocity field and the vortical surfaces. The velocity field can be meaningfully visualised with the velocity streamlines as common practice. Pronounced vortices can be clearly seen. Characteristic for the vortex streamlines are helicoidal paths in opposition to smooth ones. In problems with relative motion, important viscosity effects and large strain, the visualisation of vortical surfaces might bring additional insight.

Several criteria for the vortical surface definition exist. In comparison to other methods, the $Q$-criterion of Hunt, Wray and Moin [10] and the $\lambda_2$-criterion of Jeong and Hussain [11] offered the most suitable results for hydraulic turbines, with sharp and well defined vortical surfaces and with coherence to the tridimensional velocity streamlines. The boundary layer contains
increased vorticity, but constitutes no free vortex. Isosurfaces of \( \lambda_2 \) were cut off near to walls, based on the wall distance \( y_w \).

In the \( Q \)-criterion the scalar function \( Q = (1/2) (\Omega_{ij} \Omega_{ij} - S_{ij} S_{ij}) \) is defined, where \( S_{ij} = (1/2) (\partial u_i / \partial x_j + \partial u_j / \partial x_i) \) is the rate-of-strain tensor and \( \Omega_{ij} = (1/2) (\partial u_i / \partial x_j - \partial u_j / \partial x_i) \) the vorticity tensor. The condition \( Q > 0 \) implies the vorticity magnitude greater than rate-of-strain magnitude and the local pressure lower than the surrounding one. This is the typical case at the runner channel vortex, due to the increased vortex streamlines velocity and consequent reduced local pressure. Vortical structures are shown as isosurfaces of \( Q \).

The \( \lambda_2 \)-criterion relies on the second eigenvalue of the tensor \( S_{ik} S_{kj} + \Omega_{ik} \Omega_{kj} \), i.e. \( \det [(S_{ik} S_{kj} + \Omega_{ik} \Omega_{kj}) - \lambda \delta_{ij}] = 0 \), \( \lambda_1 \geq \lambda_2 \geq \lambda_3 \). The condition \( \lambda_2 < 0 \) guarantees a local pressure minimum, being derived from the vorticity transport equation neglecting unsteady straining and viscosity. As explained before, this is the typical case at the runner channel vortex. Isosurfaces of \( \lambda_2 \) characterise the vortices.

The assessment of the vortex intensity is controversial. It might be defined by the size of the cavitating region, velocity magnitude of the helicoidal streamlines at vortex regions or extension of the vortical isosurfaces.

### 4.3. Runner Channel Vortex Analysis

Figures 7 and 8 show the calculated time-averaged velocity streamlines and vortical surfaces for the three chosen heads and variable power output. The three-dimensional geometry in Figure 7 is represented in meridian coordinates \((x, y, z) \rightarrow (r, z)\). Three runner channels are depicted. The blades are seen from behind with the trailing edge at the pictures foreground and the leading edge at the background. The crown is located at the top and the band at the bottom. The blade surface is almost completely shown from the suction side.

Generally analysing the pictures, three independent vortices can be identified. The first vortex starts at the runner crown, crosses the conformal surfaces into direction to the band, tends to the spanwise direction and leaves the runner around the middle of the blade. This vortex will be denoted as runner crown channel vortex. The second vortex begins near to leading edge close to the band, going from the leading to the trailing edge always tight to the band. It will be called the runner band channel vortex. The third vortex appears at the blade leading edge, flows in direction to the trailing edge, changing its direction and progressively approaching the band. It will be identified as the leading edge vortex.

Near to full load \( P/P_{\text{max}} \geq 0.80 \) at all three heads almost no vortical structures are present (therefore, \( P/P_{\text{max}} = 1.00 \) and \( P/P_{\text{max}} = 0.90 \) are omitted here). With the reduction of flow and power the vortices get thicker and considerably longer. This is the common behaviour repeatedly observed at numerous model tests and numerically reproduced here. This is also the expected behaviour, with the set-in and development of the channel vortices as part load and deep part are successively reached.

The runner crown channel vortex starts being present around \( P/P_{\text{max}} \geq 0.70 \). At the beginning it is located near to the blade trailing edge and with decreasing power and flow it moves forward to the middle of the blade in spanwise direction. It exits the runner channel around the middle of the blade in span direction. Due to its start at part load near to the trailing edge, it is sometimes taken for as runner trailing edge vortex. The runner crown channel vortex comes from the flow separation in spanwise direction at the blade suction side near to the crown. The cross-flow region from crown to band, crossing the conformal planes or spanwise direction, becomes progressively larger towards part load and deep part load. This cross-flow region together with the flow blockage caused by the rotating vortex rope in the draft tube cone, leads to the flow separation starting at the crown and generates the vortex. Figure 9 presents the time-averaged wall shear lines, i.e. skin friction lines, at the blade suction side. The cross-flow region with decreasing flow becomes evident as well as the separation location, where the wall...
\[ \frac{P}{P_{\text{max}}} = 0.80 \]
\[ \frac{P}{P_{\text{max}}} = 0.70 \]
\[ \frac{P}{P_{\text{max}}} = 0.60 \]
\[ \frac{P}{P_{\text{max}}} = 0.50 \]
\[ \frac{P}{P_{\text{max}}} = 0.40 \]

\[ H_{\text{max}}, \frac{n_1'}{n_{1\text{opt}}} = 0.9900 \quad H_{\text{rated}}, \frac{n_1'}{n_{1\text{opt}}} = 1.1062 \quad H_{\text{min}}, \frac{n_1'}{n_{1\text{opt}}} = 1.3380 \]

**Figure 7.** Time-averaged velocity streamlines and vortex Q-criterion isosurfaces at different heads and power.

Shear lines converge. The cross-flow region and flow separation are coherent with the velocity streamlines and vortical surfaces at Figures 7 and 8.

At extreme heads, i.e. towards \( H_{\text{max}} \) and \( H_{\text{min}} \), the misalignment of the runner inflow to the blade inlet angle leads to strong flow momentum variation, mainly directional. Near to the blade leading edge the band curvature is significant, also importantly changing the flow momentum. These geometrical features at the runner inlet near to the leading edge and band
force the flow streamlines to strong curvatures, eventually leading to separation and vortex generation. As its name suggests, the runner band channel vortex keeps tight to the band. It can be noted at Figures 7 and 8, mainly at the extreme heads, especially at $H_{\text{min}}$. Around the rated head the runner band channel vortex is much weaker or even not present. The rated head is mostly located around the middle of the head range, where the flow incidence angle is quite low, significantly reducing the flow curvature around the blade leading edge and thus almost eliminating the runner band channel.
\[ P/P_{\text{max}} = 0.80 \]
\[ P/P_{\text{max}} = 0.70 \]
\[ P/P_{\text{max}} = 0.60 \]
\[ P/P_{\text{max}} = 0.50 \]
\[ P/P_{\text{max}} = 0.40 \]

\[ H_{\text{max}}, \frac{n'_1}{n'_{1_{\text{opt}}}} = 0.9900 \quad H_{\text{rated}}, \frac{n'_1}{n'_{1_{\text{opt}}}} = 1.1062 \quad H_{\text{min}}, \frac{n'_1}{n'_{1_{\text{opt}}}} = 1.3380 \]

**Figure 9.** Time-averaged wall shear lines (i.e. skin friction lines) in meridian coordinates.

Figure 10 brings the time-averaged projected velocity streamlines at the blade conformal plane at the normalised span coordinate \( v = 0.25 \). The inlet flow misalignment at the blade leading edge is notorious at the extreme heads. At extreme high and low heads there might even be flow separation at the blade leading edge suction and pressure side, respectively. The consequence is the appearance of the runner leading edge vortex, as seen at Figures 7 and 8. The runner leading edge vortex evolutes towards the trailing edge, tending to the spanwise direction and leaves the blade near to the band. It can be noticed that with increasing head and reduced
flow at deep part load the runner leading edge vortex merges with the runner crown vortex. This behaviour is commonly watched during model test observations. At extreme low heads the simulation results suggest that these vortices cross each other, but without merging. Around the rated head the favourable incidence angle avoids the formation of the runner leading edge vortex along a long flow range.

It should be stressed again that the vortical regions do not necessarily cavitate. The vortices may be present in the runner velocity field, although they might not be visible. Whether the vortices cores cavitate or not, depends on the operating point, on the prototype plant conditions and on the hydraulic design quality or aggressivity. For example no inlet edge cavitation and no cavitating developed vortex were present at FT 60 during model test observations for the operating range between $H_{\text{max}}$ and $H_{\text{min}}$ and full load and part load.

4.4. Runner Channel Vortex and Pressure Oscillations

The varying runner inlet flow caused by the kinematic interaction with the guide vanes and the rotating vortex rope under the runner cone and in the draft tube cone dynamically influence the pressure and velocity fields in the runner channels. This variation associated to the helicoidal rotating runner channel vortex streamlines originates pressure oscillations in the runner channel.

Figure 11 depicts the instantaneous projected velocity streamlines at the blade conformal plane at the normalised span coordinate $v = 0.25$ for all runner channels at the rated head and deep part load, $n'_{1}/n'_{1\text{opt}} = 1.1062$ and $P/P_{\text{max}} = 0.40$. The flow in each channel is slightly different. Its variation along the time and runner rotational position suggests the arising pressure oscillations.
5. Experimental Results

The numerical fluid simulation model employed for the vortex identification was able to predict the pressure oscillations in the runner. Nevertheless, for the FT 60 model turbine, pressure oscillation measurements at the runner blades were available. Once such a complex experimental setup is available, it offers the possibility to investigate the complete turbine hill chart, offering more operating points than the numerical calculations. The limited number of pressure transducers offers low resolution for the measured transient pressure field. Nevertheless, the measurements can be used in combination with numerical simulations or to analyse the behaviour of individual locations as done here.

At Figure 12 the normalised pressure oscillation amplitude \((\Delta H/H)\) is plotted as function of the normalised power \(P/P_{\text{max}}\) for the three heads considered in the study. Four measuring locations were chosen at the blade suction side near to the leading edge near to crown, \(u = 0.60, v = 0.20\), near to the trailing edge near to crown, \(u = 0.95, v = 0.20\), near to the leading edge near to band, \(u = 0.60, v = 0.80\), and near to the trailing edge near to band, \(u = 0.95, v = 0.80\). The measuring points at the blade suction side were selected rather than at pressure side, as long as the vortices were closer to the suction side causing higher pressure oscillation amplitudes.

In the part load region around \(P/P_{\text{max}} = 0.80\) a local pressure oscillation peak can be noticed associated to the rotating vortex rope under the runner cone and in the draft tube cone. Reducing the load and entering the deep part load region below \(P/P_{\text{max}} = 0.60\) the pressure oscillation amplitude starts increasing due to the set-in and development of the runner channel vortex. In general the pressure oscillation amplitude was higher at the extreme heads, \(H_{\text{max}}\) and \(H_{\text{min}}\), than at the rated head. At the rated, the runner crown channel vortex dominates, while the runner band and leading edge channel vortices are almost not present. At extreme low head, \(H_{\text{min}}\), the pressure oscillation amplitude is considerably high at the leading edge due to the separated flow and significant leading edge vortex. At the trailing edge near to band the pressure oscillation amplitude tends to increase for all the head range, since the vortices exit the blade at the trailing edge approaching the band for reduced flows.

6. Conclusion

The fluid flow numerical simulations were able to identify the runner channel vortex initiation and development as well as its physics and topology. Actually up to three distinct and simultaneous runner channel vortices could be characterised: the runner crown channel vortex, the runner band channel vortex and the runner leading edge channel vortex. These vortices were present in the velocity field. However, they did not necessarily cavitate and could not necessarily be optically observed during model. Whether the vortices cavitate or not depends on the hydraulic design and prototype plant conditions. The FT 60 did not presented visual
developed vortex cavitation inside its normal operating range and they could only be assessed through the numerical fluid flow calculations.

The physical nature of the runner channel vortex and its behaviour as described here is present in other Francis machines with similar specific speed, $50 \text{ min}^{-1} \leq n_{\text{opt}} \leq 70 \text{ min}^{-1}$, and can be transferred to them. The simulation and vortex visualisation procedure could be extended in the future to other dynamic fluid flow phenomena and machine types as e.g. pump-turbines.

The dynamic nature of the runner channel vortex was responsible for the induction of pressure oscillation at the runner. They were experimentally measured at the model turbine runner using special setup. They showed the relation between the runner channel vortex and pressure oscillations at different locations of the turbine runner with different amplitudes, related to the vortex shape. Such complex experimental setups for pressure oscillation measurements at the model runner might eventually get more common in the future and bring the possibility to investigate other dynamic fluid flow effects and their associated pressure oscillation level.

References

[1] Magnoli M V 2014 Numerical simulation of pressure oscillations in large Francis turbines at partial and full load operating conditions and their effects on the runner structural behaviour and fatigue life Ph.D. thesis Technische Universität München

[2] Menter F R and Egorov Y 2001 A Scale-Adaptive Simulation Model using Two-Equation Models Tech. Rep. Paper 2005-1095 AIAA Reno, Nevada, USA

[3] Magnoli M V 2009 JASS 2009 – Joint Advanced Student School (St. Petersburg, Russia)

[4] Magnoli M V and Schilling R 2014 10th International ERCOFTAC Symposium on Engineering Turbulence Modelling and Measurements (ETMM10) (Marbella, Spain)

[5] Magnoli M V and Schilling R 2008 Proceedings of the 24th IAHR Symposium (Foz do Iguacu, Brazil)

[6] Hasmatuchi V and Avellan F 2012 SimHydro 2012 – New frontiers of simulation (Nice, France)

[7] Benigni H, Jaberg H, Lampi J and Franz E 2007 HYDRO 2007 (Granada, Spain)

[8] Wunderer R 2007 Aktive Strömungsbeeinflussung Tech. rep. Forschungsinitiative KW 21

[9] Magnoli M V 2016 Proceedings of the 28th IAHR Symposium (Grénoble, France)

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

[11] Jeong J and Hussain F 1995 Journal of Fluid Mechanics 69–94

Figure 12. Pressure oscillation amplitude at selected measuring locations and heads.