# Unsteady CFD prediction of von Kármán vortex shedding in hydraulic turbine stay vanes

**Thi C. Vu**
GE Energy, Hydro
795 George V, Lachine
Québec, Canada, H8S-4K8

**Bernd Nennemann**
École Polytechnique de Montréal
GE Energy, Hydro
795 George V, Lachine
Québec, Canada, H8S-4K8

**Philippe Ausoni, Mohamed Farhat, François Avellan**
École Polytechnique Fédérale de Lausanne (EPFL),
Laboratory for Hydraulic Machines (LMH),
Av. de Cour 33bis, CH-1007 Lausanne,
Switzerland

## 1. Introduction

All hydraulic turbines containing a casing with stay vanes face the potential dynamic problem of stay vane von Kármán vortex shedding. For hydraulic efficiency purposes the stay vanes tend to be relatively slender in the direction normal to the flow thus being flexible in this direction. As a result structural vibrations may be excited by the von Kármán vortices shedding at the trailing edge of the vanes. When the excitation frequency coincides with one of the natural frequencies of the stay vane, resonance occurs, causing vibration and potentially initiating cracks in the vane geometry if the amplitude of the excitation force is sufficient. Although the problem occurs rarely for high head or medium head Francis turbine, it is particularly true for low head turbines such as Kaplan and propeller type turbines. The frequency and amplitude of the von Kármán vortices are highly dependent on the free stream velocity and on the trailing edge profile of the vane. At the design stage, it is important not to match the vortex shedding frequency with the vane natural frequency. Figure 1 shows an example of von Kármán vortex shedding phenomena in a Kaplan turbine distributor as predicted with CFD.

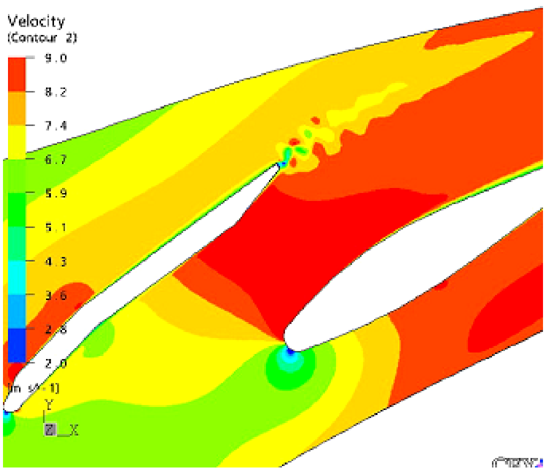



***Figure 1**: Instantaneous velocity magnitude distribution of von Kármán vortex shedding in a Kaplan turbine stay vane – guide vane combination as predicted by CFD*

The traditional method of determining the vortex shedding frequency is by using the empirical Strouhal number with the given wake thickness and free stream velocity at the vane trailing edge. But this approach is not valid for geometries that are very different from standard geometries and the dependency of the Strouhal number on the flow Reynolds number prevents us from obtaining a good empirical correlation with experimental data. The Strouhal number varies in the range of 0.15 to 0.3, and it is dependent on the vane geometry and the local Reynolds number. The determination of the Strouhal number from the literature is usually for model testing where the flow Reynolds number is usually not higher than 3.0E5, but for prototype size operating condition, the local Reynolds number (based on the stay vane length) may vary from 1.E6 to 5.E7.

A CFD methodology for the prediction of the von Kármán vortex shedding frequency using unsteady flow computation has recently been developed at GE Energy, Hydro. An accurate prediction of excitation frequency and exciting forces is paramount in order to prevent damage to the structure in the prototype. The validation of the CFD results were made for both prototype and model sizes. The validation with site measurements from prototype size is important in order to evaluate the accuracy of the prediction in the context of the machine as a whole where there are a lot of unknowns such as: measurement precision, presence or absence of lock-in, actual local incidence angle to stay vane, 3D effects etc. This kind of validation gives us an indication of the level of confidence we can have in the prediction method. However, with prototype frequency measurements at site, it is impossible to validate the CFD code regarding its ability to predict frequencies and flow patterns correctly due to the quite often presence of lock-in phenomena. On the other hand, the study of the stay vane vortex shedding phenomena with a whole turbine assembly in model size is difficult due to limited access for instrumentation. Therefore an experimental investigation of the vortex shedding phenomena of a NACA profile in the EPFL high-speed cavitation tunnel has been initiated in the context of the Hydrodyna project. The reliable high fidelity experimental measurements help to benchmark and to fine tune the CFD tool such as the choice of mesh size, computational time step, type of turbulence model or wall function, etc.

In the following paragraphs, we will present:

- The CFD methodology for unsteady state computation.
- Experimental investigation of the vortex shedding phenomena in the cavitation tunnel of EPFL-LMH.
- Validation of the CFD result with EPFL experimental data.
- Validation of the CFD result with site measurements.

## 2. CFD methodology for stay vane von Kármán vortex shedding predictions

### 2.1 Computational approach

The von Kármán vortex-shedding phenomenon is significantly influenced by the boundary layer thickness at the trailing edge. Therefore it is important to obtain a good boundary layer resolution for an accurate prediction. Wall functions are not suitable in such application. A very fine mesh resolution near the solid wall is required in order to well resolve the flow behaviour in the boundary layer region. The k-ω based turbulence models are well suited for this task because they allow a direct integration to the wall without having to subdivide the flow domain into low- and high-Reynolds number regions as required with k-ε based turbulence models. At GE Energy, Hydro we use the commercial RANS CFD code ANSYS-CFX for the von Kármán vortex shedding simulation. Available turbulence models for this task are standard k-ω, SST (shear stress transport) and Reynolds-Stress-ω. Studies on the truncated NACA009 profile presented in the experimental section below show that the difference in the results among the three turbulence models mentioned above is less than 1% and therefore negligible for practical purposes. In conjunction with the SST turbulence model, ANSYS-CFX provides a model for the laminar-turbulent transition of the boundary layer. As we will see below, this is essential for the validation of the CFD results with the laboratory measurements. Within the ANSYS-CFX solver we use the "High resolution" spatial discretisation scheme. In time we use second order backward Euler discretisation. Standard approach at GE Energy, Hydro is the use of k-ω or SST turbulence model with laminar-turbulent transition at model scale and without for prototype scale predictions.

### 2.2 Mesh and time step sizes

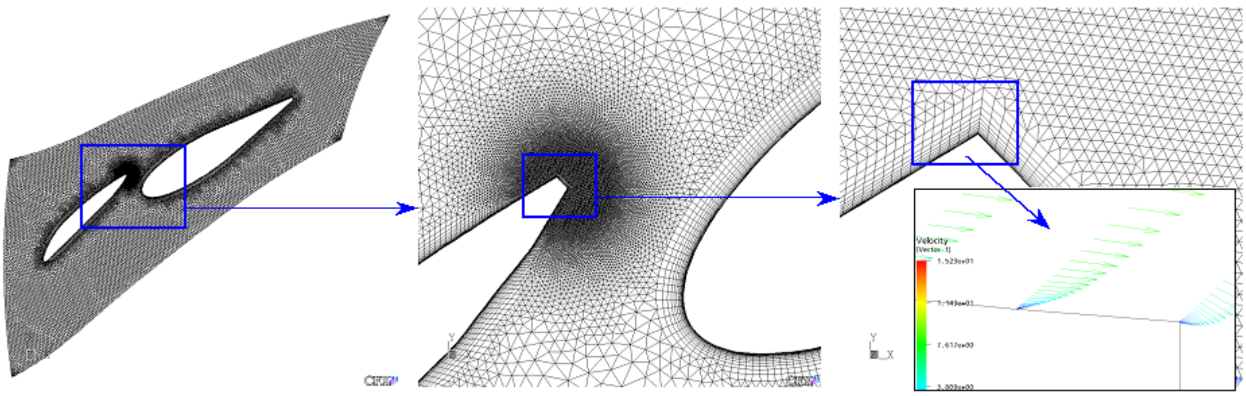


***Figure 2**: Mesh resolution (~100k nodes) as well as resulting boundary layer velocity profile for von Kármán predictions*

As mentioned above, a good spatial resolution, i.e. a fine mesh is essential for an accurate prediction of the von Kármán vortex shedding phenomenon. In order to allow for a time-efficient computation we employ a 2-dimensional approach. Strictly speaking the vortex shedding phenomenon is 3-dimensional, Ausoni et al. [1] but we believe that a 2D approach is sufficiently accurate and permits us to use CFD calculations as predictive tool in day-to-day work. Figure 2 shows an example of a hybrid mesh for a stay vane-guide vane combination. The quasi-3D (with one element thick in the 3rd direction) hybrid mesh consists of pyramidal elements in the free stream region and hexahedral elements near the walls. The mesh, being generally fine in the whole flow domain, is additionally refined near the walls and around the trailing edge of the stay vane. Near-wall y+ values are in the order of 1. A typical mesh for a stay-vane von Kármán vortex shedding prediction has approximately 100,000 nodes. It is recommended that the computational flow domain would comprise together stay vane and guide vane.
A study of the influence of the time step size on the vortex shedding frequency, as shown in Figure 3a, indicates that minimum time step size of about 100 time steps per shedding period is required to obtain sufficient accuracy within 1% error.

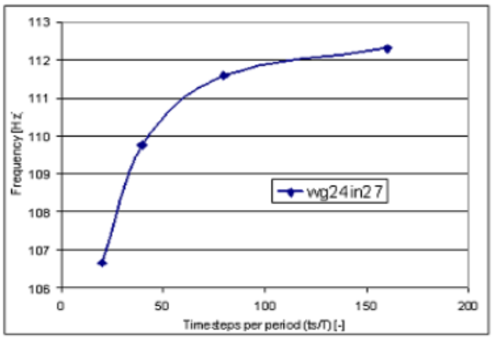



*Figure 3a*: *Influence of time step size on shedding frequency*

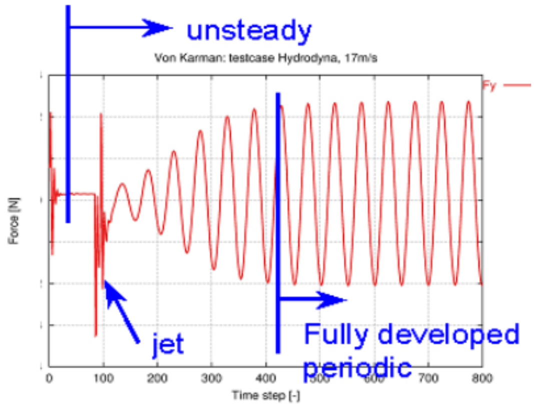



*Figure 3b*: *Start up of the unsteady shedding phenomena*

**2.3 Unsteady start-up procedure**

Unsteady calculations require a steady state solution as initial condition. After switching to unsteady calculation mode, the flow solution can take a large number of time steps before the unsteady shedding phenomenon begins. The transition from steady to unsteady state can be accelerated with a perturbation initiated in the vicinity of the trailing edge during a short period of time. Figure 3b illustrates this start up procedure by monitoring of the force acting on the vane. When the steady state solution is achieved within about 80 time steps, a jet normal to the mean flow direction is introduced in the vicinity of the trailing edge during 10 time steps. This is sufficient to start the shedding phenomena and the resulting pressure and force fluctuations. Full periodic condition is obtained at about 400 time steps after the start-up.

## 3. Experimental investigation of the vortex shedding phenomena

The experimental investigation of the vortex shedding phenomena of the truncated NACA0009 profile was conducted in the context of the Hydrodyna project [1]. The EPFL high-speed cavitation tunnel used for the study, outlined on Figure 4, is a closed loop with a test section of 150x150x750 mm, Avellan et al. [2].

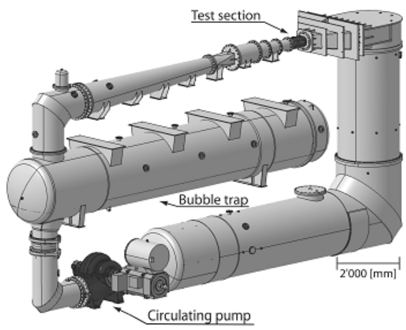



*Figure 4*: *EPFL high speed cavitation tunnel*

The experimental 2D hydrofoil, sketched on Figure 5, is a blunt truncated NACA0009 made of stainless steel. The profile has a chord length L=100 mm and a span b=150 mm with a truncated trailing edge thickness h. The hydrofoil mounting in the test section can be considered as a perfect embedding on one side and pivot embedding on the other side. The operating flow parameters are the upstream velocity $C_{ref}$, the cavitation index $\sigma$ and the hydrofoil incidence angle $\alpha$. For all the measurements, the incidence angle of the hydrofoil $\alpha$ is fixed at 0° and the cavitation index is set high enough to avoid any cavitation development.

Since the turbulence intensity of the incoming flow is low, the boundary layer on the hydrofoil stays laminar for the first half of the profile. Caron [3] shows that the boundary layer transition to turbulence for this hydrofoil geometry is located slightly downstream of the mid-chord for an incidence angle of 0° and Reynolds number $Re_h$ between $42 \cdot 10^3$ and $70 \cdot 10^3$. Ait Bouziad [4] reports boundary layer computations and evaluates its thickness at the leading edge, $\delta \sim 100 \mu m$.

In order to investigate the effect of a fully turbulent boundary layer on the vortex shedding frequency, we have triggered the turbulent boundary transition right at the leading edge profile with rough surfaces. A distributed roughness made of glue and 125μm diameter sand are placed on both sides of the hydrofoil and are located 4 mm behind the leading edge, see Figure 5. The length of the rough stripes are only 4 mm in the flow direction, the rest of the hydrofoil remains smooth. For such a configuration, the transition of the boundary to turbulent regime is uniformly forced. In the rest of the paper, this configuration will be designated as rough leading edge. The case without rough stripes will be designated as smooth leading edge. The experimental investigation was carried out for both configurations

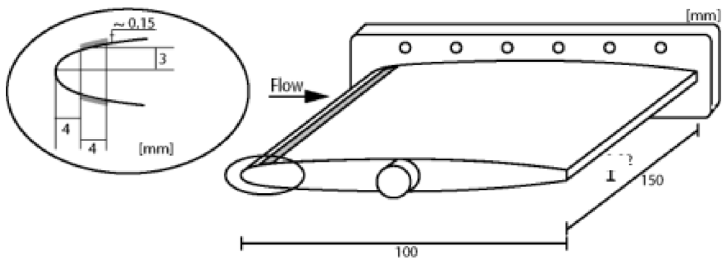



***Figure 5:*** *Blunt truncated Naca0009 hydrofoil with distributed roughness at leading edge*

The vortex-induced hydrofoil vibration is measured with the help of a laser vibrometer. The measurement principle of the device is based on the detection of the frequency shift of the reflected laser beam according to the Doppler effect; the frequency shift being directly related to the displacement velocity of the surface in the laser direction. Laser vibrometer measurements are synchronized with an accelerometer, which is fitted on the profile support. Considering the accelerometer signal as the reference signal, the amplitude and phase of the hydrofoil motion at each measurement point are measured and the eigenmodes identified for the detected hydro-elastic couplings. The data acquisition system has 16 bits A/D resolution, 16 inputs, a memory depth of 1 MSamples/Channel and a maximum sampling frequency of 51.2 kHz/Channel.

The measurement of the unsteady velocity field in the wake of the hydrofoil is carried out with the help of a Dantec 2D Particle Image Velocimeter. The seeding particles consist of hollow glass spheres (10 μm diameter). The laser sheet (1mm thickness) is provided by two pulsed lasers (Gemini Research minilaser) and is aligned to the mean flow direction, i.e. vertical plane configuration. Perpendicular to this plane, a high-sensitivity camera (Dantec highsens) with a 60-millimeter lens (Nikon) is placed. The configuration where the laser sheet illuminates the whole span is also investigated, i.e. horizontal plane configuration. The acquisitions are double frame, single exposure. The velocity field is derived from the cross-correlation of the two consecutive frames. The interrogation area size is 32x32 pixels with an overlap of 50 %. A Gaussian window function is used to reduce the cyclic noise from the correlation map. It multiplies the grayscale values with a factor between 0 and 1 depending on the position in the interrogation area. A validation of the correlation peak is done with the relative height of the highest correlation peak to that of the second highest (factor 1.2).

## 4. Validation of the CFD calculations with laboratory results

The traditional method of determining the vortex shedding frequency is by using the empirical Strouhal number with the given wake thickness and free stream velocity at the vane trailing edge which can be expressed as follows:

$$St^* = \frac{f_s \cdot h^*}{C_{ref}} = \frac{f_s \cdot (h + 2\delta^*)}{C_{ref}}$$

with $\delta^* = \int_0^\infty (1 - \frac{c}{C_{ref}})dy \cong \int_0^\delta (1 - \frac{c}{C_{ref}})dy$

$\delta$ boundary layer thickness

$\delta^*$ boundary layer displacement thickness

The wake thickness is the sum of geometrical trailing edge vane thickness and the boundary layer displacement thicknesses from both side of the vane.

Figure 6 shows the measured vortex shedding frequencies with the two configurations of the NACA009 profile, smooth and rough leading edges. The velocity of the test section varies from 6 m/s to 32m/s. The vortex shedding frequency varies quite linearly with the free stream velocity, except around the region of 900 Hz where the lock-in phenomenon is taking place. The lock-in occurs when the vortex shedding frequency is close to the natural frequency of the vane. In this case, with the smooth leading edge hydrofoil, the shedding frequency stays constant at 900 Hz for a range of upstream velocity between 11 m/s to 13.5 m/s. As mentioned earlier, with a very regular incoming flow upstream of the test section, the boundary layer on the hydrofoil with smooth leading edge remains laminar in the beginning of the profile and becomes turbulent only after the mid-chord of the vane. Whereas with rough leading edge configuration, the turbulent boundary layer is triggered by the rough stripes at the beginning of the hydrofoil and remains turbulent along the vane profile. Since the turbulent boundary layer grows much thicker than laminar boundary layer, it results that the trailing edge wake thickness of the rough vane is larger than the one with smooth leading edge. The vortex shedding frequency varies inversely with the vane wake thickness; therefore Figure 6 shows that the vortex frequency of the smooth leading edge vane is always about 20% above the one of rough leading edge.

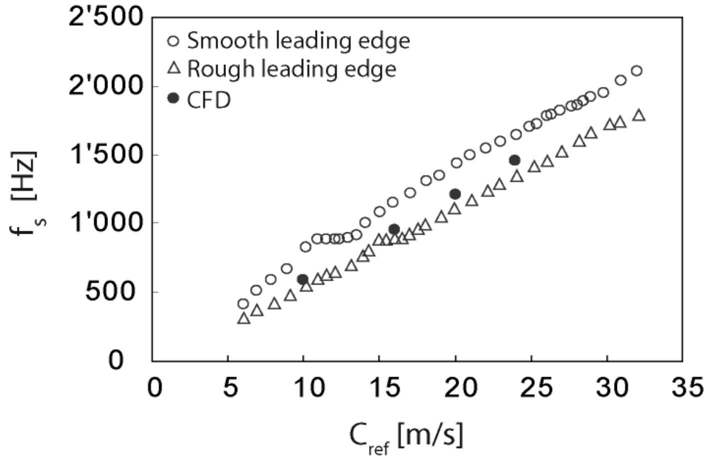


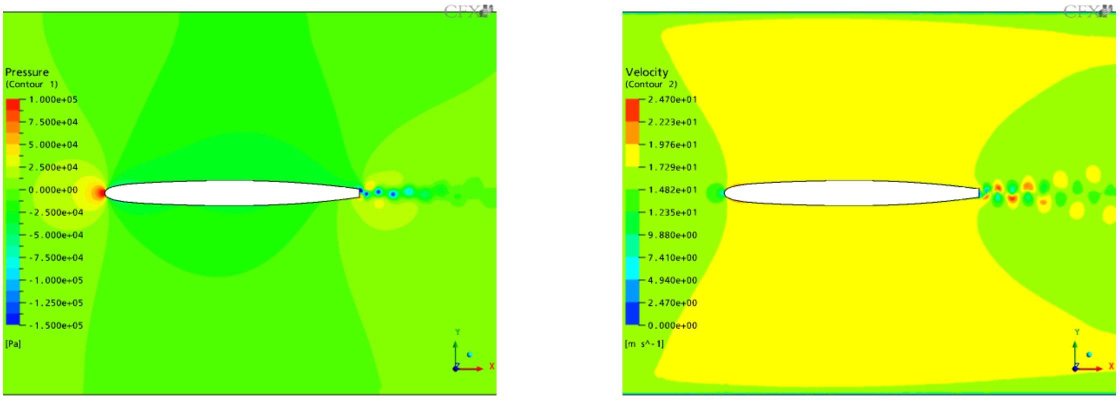

***Figure 6:*** *Vortex shedding frequency at different upstream velocities*

Table I shows the results obtained for a specific upstream velocity at 17 m/s. At this condition, the von Kármán vortex shedding frequency for the smooth leading edge is 1214 Hz whereas for the rough leading edge the shedding frequency is 929 Hz.

**Table 1:** Comparison of experimental and numerical results, reference velocity 17m/s

| Case number | Case description | Frequency [Hz] | Strouhal [-] based on wake thickness |
|---|---|---|---|
| 1 | Experimental, smooth LE | 1214 | |
| 2 | Experimental, rough LE | 929 | |
| 3 | CFD, SST with transition model | 1220 | 0.24 |
| 4 | CFD, SST without transition model | 1000 | 0.23 |

We use this lock-off test condition, $C_{ref}$=17 m/s, for the CFD validation. Figure 7 shows instantaneous pressure and velocity magnitude distributions of the vortex shedding phenomena simulated with ANSYS-CFX10 solver for the truncated NACA0009 profile. Low-pressure regions in the vortex cores can be discerned as seen in Figure 7a. The plot of the velocity norm shows alternating high and low regions created by the vortices. Numeric values such as forces on the stay vanes or local pressures can be obtained from the solution monitoring. An example of this is shown in Figure 3b. Time-varying monitor values can be used to identify the shedding frequency by means of a Fourier transform.




***7a** - Instantaneous pressure distribution* ***7b** - Instantaneous velocity magnitude distribution*

***Figure** 7:von Kármán vortex shedding – Numerical results for Vref=17 m/s*

Concerning the prediction of the vortex shedding frequency it is important to consider the exact configuration of the test setup in order to specify the boundary conditions and to select appropriate turbulent model for the computation. In practical CFD calculations the flow is considered fully turbulent, which is generally valid for prototype and model test situations. However in the cavitation tunnel of EPFL-LMH, the flow upstream of the measured profile is very regular, such that the flow along the profile (with smooth leading edge) starts out as laminar flow and exhibits a laminar-turbulent transition in the boundary layer only at the hydrofoil mid-chord. Therefore initial attempts to validate CFX calculations, using fully turbulent boundary layer model, with the measurements failed as a comparison of cases 1 (experimental, smooth LE) and 4 (CFD, without transition model) in Table 1. But this CFD result without the laminar-turbulent transition model (case 4) correlates quite well with experimental data of the NACA009 profile with rough leading edge (case 2), 1000 Hz versus 929 Hz. The measured shedding frequency is 7% lower than the CFD result. This can be explained by the fact that the rough stripes have created a thicker turbulent boundary layer than it should be. In order to simulate correctly the test configuration with smooth leading edge, the laminar-turbulence transition model is selected for case 3. Table 1 shows a very good correlation between the numerical prediction and experimental data (cases 1 and 3).
The evolution of the boundary layer thickness along the chord of the NACA profile is illustrated clearly for both vane configurations in Figure 8. The calculation with standard SST model without the laminar-turbulent transition model and with turbulent inlet boundary condition show a much thicker boundary layer thickness at the trailing edge than the calculation with the transition model, 3mm vs. 0.5mm. In the calculation with transition model, the boundary layer remains laminar – and therefore thin – for approximately half the chord length of the profile.

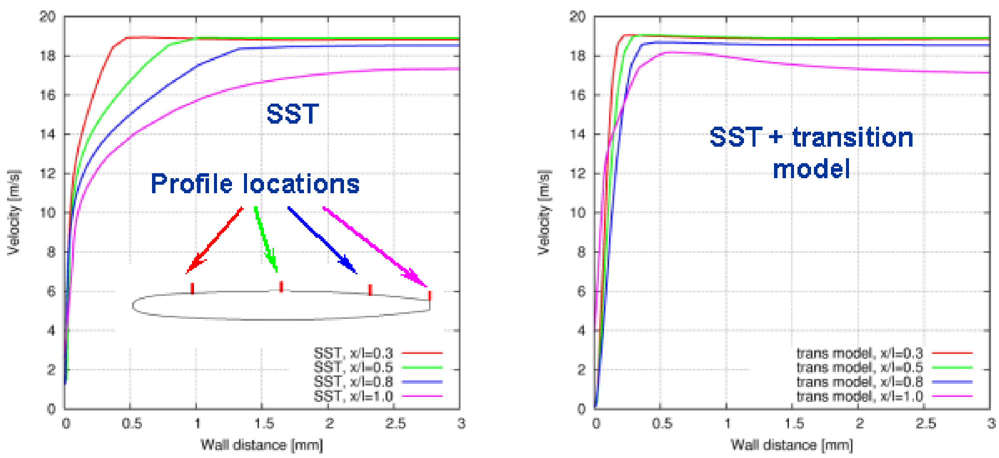
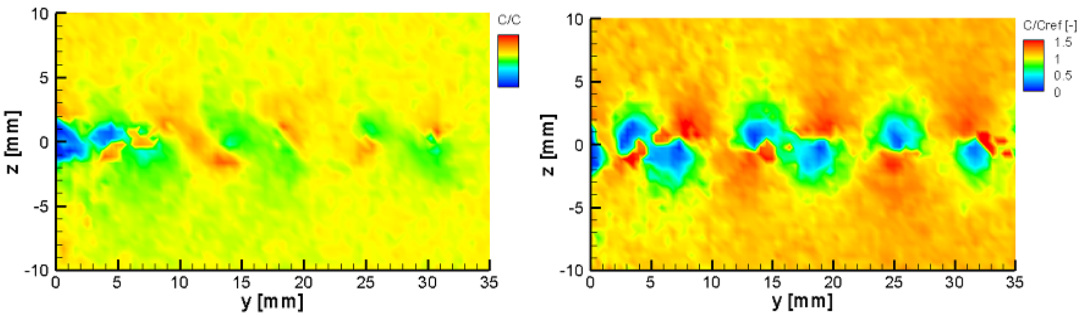



***Figure 8**: Profile wall boundary layer evolution for CFD calculation without (left) and with (right) transition model*

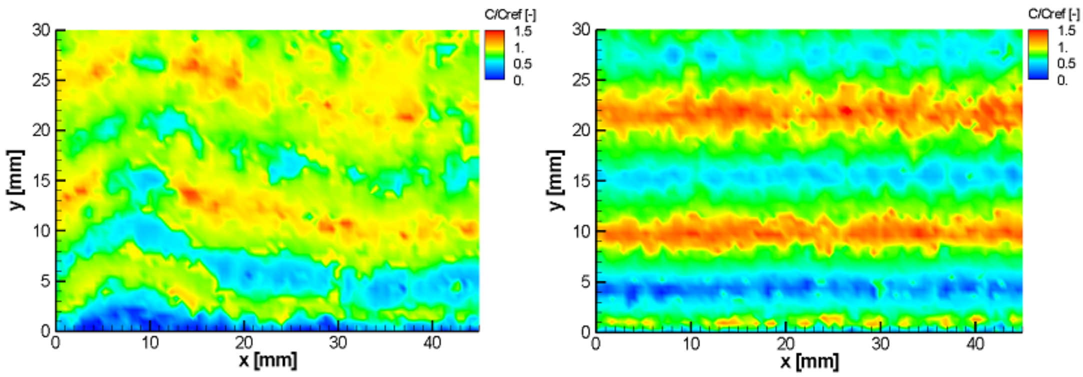
The boundary layer displacement thickness for both cases can be estimated with the CFD result. In this case we have made a rough estimation for the boundary layer displacement thicknesses from Figure 8. Using the profile trailing edge thickness plus boundary layer displacement thickness to compute the wake thickness leads to similar Strouhal numbers for both cases as shown in Table I.

The wake velocity fields as measured with PIV in two perpendicular planes, vertical and horizontal, are shown in Figure 9 and Figure 10. The left hand sides on these two figures, Figure 9a and Figure 10a, show the instantaneous velocity distributions at lock-off conditions ($C_{ref}$=17.5m/s and 16m/s). At the right hand side, Figure 9b and Figure 10b show the instantaneous velocity distributions at lock-in conditions ($C_{ref}$=12m/s). In lock-in condition, the vortex shedding street is clearly observed as shown in Figure 9b and it is similar to the CFD result as shown in Figure 7a. In lock-off condition, the flow is however three-dimensional and therefore a vortex shedding street is not consistently present in any single plane of observation. This illustrates the difficulty of validating entire unsteady flow field computations with measurements. In this case the difficulty arises due to the fact that the flow is 3D while the computations are performed in 2D, and a reasonable phase averaging to filter the von Kármán vortices is difficult. More appropriate data processing needs to be developed to be able to compare computed and measured flow fields. As we have seen above, frequency predictions match very well with the measurements and we expect comparisons of integral values such as forces and moments to be equally successful. This however has to be proven with future measurements.




***9a:*** *Cref=17.5m/s – Lock-off condition* ***9.b:*** *Cref=12m/s - Lock-in condition*

***Figure 9:*** *Instantaneous velocity distributions in* ***a vertical plane*** *at lock-off and lock-in conditions*




***10a:*** *Cref=16.m/s – Lock-off condition* ***10.b:*** *Cref=12m/s - Lock-in condition*

***Figure 10:*** *Instantaneous velocity distributions in* ***a horizontal plane*** *at lock-off and lock-in conditions*

## 5. Validation of the CFD result with prototype frequency measurements

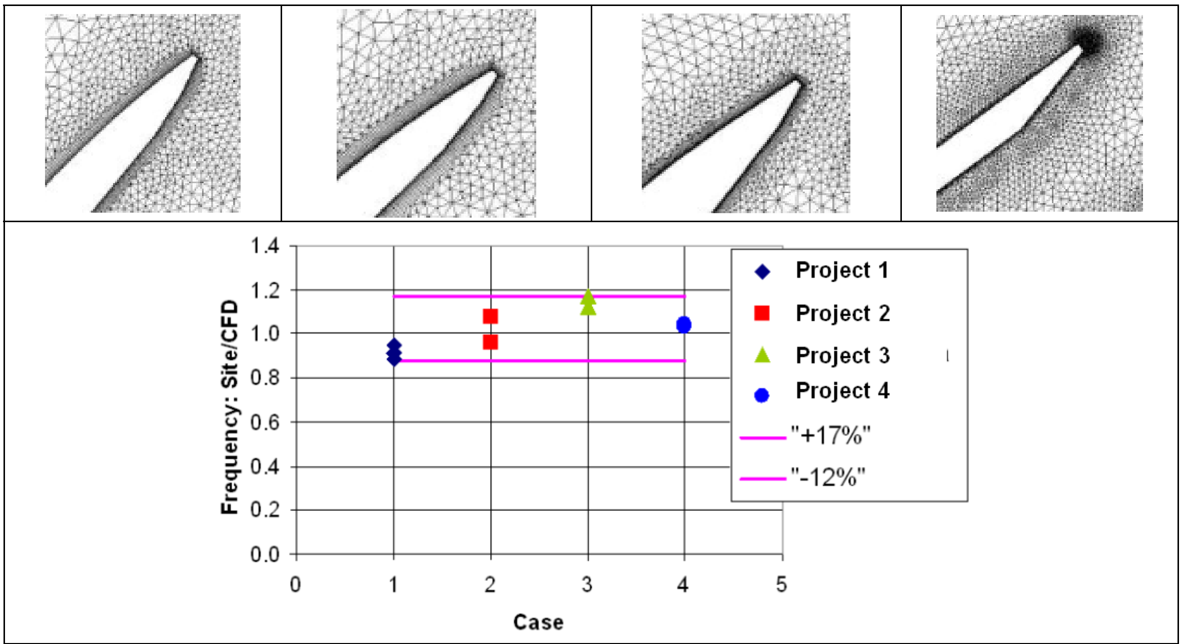



***Figure 11.*** *Comparison with prototype measurements at sites*

GE Energy Hydro has frequency measurements for a number of turbine sites where problems with von Kármán vortex shedding occurred. Problems range from noise to potential stay vane cracking in the worst cases. This means that all the study cases are at or close to the lock-in phenomena. The CFD methodology for 2D unsteady computation as described here is used to predict the vortex shedding frequency in prototype size. One of the uncertainties is the choice of exact flow angle at the stay vane inlet. Since the flow in the casing is highly three-dimensional, incoming flow angles to the stay vane are generally not uniform in the circumferential direction and also they vary from top to bottom of the distributor. Due to secondary flow developed in the casing, the incoming flow angle is smaller at the distributor centre and increases toward the top and bottom of the distributor. Therefore the range of inlet flow angles should be determined by performing a complete 3D flow analysis of the casing and distributor assembly. Usually we select 3 values of the inlet flow angle in order to cover the flow angle range from the top to the bottom of the distributor. For some stay vane geometries, the vortex shedding frequency is very sensitive to the inlet flow angle. Because the size of prototype turbine is many time superior to the one of model size and the flow in prototype turbine is highly turbulent, it is not necessary to use the laminar-turbulent transition model.

The CFD frequency predictions compared to the measurements are plotted in Figure 11. In general, we can conclude from the data comparison that the precision is within ±18% without considering the effect of the lock-in, which contributes largely to the discrepancy. In section 4, we have seen that the effect of the lock-in contributes up to about ±15% of uncertainty. In a controlled environment we obtain accuracy in the order of 1% at lock-off condition.

## 6. Concluding Remarks

In this paper, we have presented a CFD methodology to predict the von Kármán vortex shedding frequency of the turbine stay vane by means of 2D unsteady state computation. The accuracy of the method has been validated against experimental data obtained with model size from laboratory and with prototype size from site.

It has been clearly demonstrated that this methodology is a reliable CFD tool for hydraulic engineer during the design process. Then for mechanical side, it is important to develop methodology to estimate the natural frequency of hydrofoil in water. Considering the uncertainty of the CFD prediction within ±18%, it is recommended that the stay vane vortex shedding frequency should be beyond at least ±20% of the vane natural frequency to avoid resonance.

## Acknowledgment

The experimental part reported in this paper was carried out for the HYDRODYNA, Eureka Research Project No. 3246, whose partners are ALSTOM Hydro, EDF-CIH, EPFL, GE Hydro, UPC-CDIF, VATECH Hydro, and VOITH-SIEMENS Hydro Power Generation. CTI, the Swiss Federal Commission for Technology and Innovation Grant No. 7045-1 and NSF, the Swiss National Science Foundation Grant No. 2000-068320 also financially support the project. The authors are very grateful to the HYDRODYNA Technical Committee for its involvement and constant support to the project. Finally the staff of the EPFL Laboratory for Hydraulic Machines should be thanked for its support in the experimental and numerical work.

## References

1. **Ausoni P., Farhat M., Escaler X, Egusquiza E., Avellan F., (2007),** Cavitation influence on Kármán vortex shedding and induced hydrofoil vibrations, J.Fluids Eng, August 2007
2. **Avellan F., Henry P., Ryhming I., (1987),** A new high speed cavitation tunnel, ASME Winter Annual Meeting, Boston (USA), 57: 49-60
3. **Caron J.-F., Farhat M., Avellan F. (2000),** On the leading edge cavity development of an oscillating hydrofoil, ASME Fluids Engineering Division Summer Meeting, Boston (USA), FEDSM2000-11016
4. **Ait Bouziad Y. (2005),** Physical modelling of the leading edge cavitation: Computational methodologies and application to hydraulic machinery, PhD thesis, EPFL, N0 3353

## The Authors

Since May 2007 **Bernd Nennemann** is Hydraulic design engineer at GE Energy Hydro. Prior to that, he was Research Assistant in a joint contract by École Polytechnique de Montréal and GE Energy Hydro starting in 2002. Within the hydraulic design group of GE Energy Hydro he developed Computational Fluid Dynamics (CFD) tools and performed CFD analyses of hydro turbine components. He has a Diplom from the Technical University Berlin, Germany, and a Master of Engineering degree from the University of Canterbury, New Zealand, both in mechanical engineering.

**Thi C. Vu** is Senior Hydraulic Engineer, GE Energy Hydro. He is in charge of the development and integration of CFD methodology for numerical flow simulation in hydraulic turbine within GE Energy Hydro. He graduated in fluid mechanical engineering from Laval University in 1970 and obtained his M.Sc.A. from Ecole Polytechnique de Montreal in 1973. He joined GE in 1974 as Research Engineer and was appointed Head of Applied Physics Technology in 1980. During his stay in Research Service Department, he worked in the field of flow simulation in hydrodynamic bearing and non-contacting seal, and also of dynamic behavior and thermodynamic optimization of the pressing and dryer sections in paper making machine. In 1983, he joined the hydraulic group of GE Energy Hydro. He is member of the Flindt & Hydrodyna technical committees and responsible for several R&D joined projects with universities, such as CERCA, Ecole Polytechnique de Montreal and university of Florida.

**Philippe Ausoni** graduated in Mechanical Engineering from EPFL in 2004. During his Master, he has the opportunity to gain work experience in the R&D department of VaTech Hydro in Zürich. In October 2005, he joined the EPFL Laboratory for Hydraulic Machines to achieve a doctoral work in the field of the fluid-structure interaction. His research is focused on vortex induced vibration and cavitation influence on Kármán vortex shedding from 2D hydrofoil.

**Mohamed Farhat** is head of research group on Cavitation and Interface Phenomena at Ecole Polytechniaue Fédérale de Lausanne. He graduated from Ecole Nationale Supérieure d'Hydraulique et de Mécanique de Grenoble (ENSHMG, France). He completed a PhD thesis on the cavitation erosion process at Ecole Polytechnique Fédérale de Lausanne (EPFL) in 1994. He worked as Research scientist at Hydro Quebec (Varennes, Canada) from 1995 to 1999. He was mainly in charge of the design and operation of cavitation monitoring system for hydraulic turbines. Since 1999, he is head of the cavitation research group at EPFL where he is supervising several PhD thesis in both fundamental and applied areas of two phase flows as well as specific instrumentation for prototypes and models of hydraulic machines.

**Francois Avellan** is director of the EPFL Laboratory for Hydraulic Machines, graduated in hydraulic engineering from the ENSHG, Ecole Nationale Supérieure d'Hydraulique, INP Grenoble, in 1977 and became Docteur Ingénieur at the IMST, Institut de Mécanique Statistique de la Turbulence de Marseille (1980). He joined the Fluid Mechanic Laboratory at EPFL, Ecole Polytechnique Fédérale de Lausanne and became head of the Cavitation Research Group of the IMHEF in 1984. He was responsible for the design and installation of the IMHEF EPFL High Speed Cavitation Tunnel and for the cavitation studies and the flow analysis in hydraulic machines. He was appointed Professor in 1994, and is now Director of the EPFL Laboratory for Hydraulic Machines. His main research domains of interests are cavitation, hydroacoustics of machines and installations, and the design of hydraulic machine runners and the evaluation of machine performances. Moreover, he is involved in contractual activities concerning model acceptance tests of turbine and pump turbine. Prof. F. AVELLAN is the Chairman of the technical committee No 4 (hydraulic machines) of the Swiss Electrotechnical Committee. He is the Convenor of Working Group 23 of IEC, International Electrotechnical Committee, in charge of the new standards concerning the model acceptance tests of hydraulic machines. Prof. François Avellan is the present Chairman of the Section on Hydraulic Machinery and Systems of the IAHR, International Association for Hydraulic Research.