Development and modelisation of a hydro-power conversion system based on vortex induced vibration

David Lefebure*, Nicolas Dellinger**, Pierre François***, Robert Mosé****
*PhD student, ICube, 2, Rue Boussingault, 67000 Strasbourg  
E-mail: david.lefebure@icube.unistra.fr
**Associate professor, ICube, 2, Rue Boussingault, 67000 Strasbourg  
E-mail: dellinger@unistra.fr
***Associate professor, ICube, 2, Rue Boussingault, 67000 Strasbourg  
E-mail: pierre.francois@icube.unistra.fr
****Professor, ICube, 2, Rue Boussingault, 67000 Strasbourg  
E-mail: mose@unistra.fr

Abstract. The Vortex Induced Vibration (VIV) phenomenon leads to mechanical issues concerning bluff bodies immersed in fluid flows and have therefore been studied by numerous authors. Moreover, an increasing demand for energy implies the development of alternative, complementary and renewable energy solutions. The main idea of EauVIV project consists in the use of VIV rather than its deletion.

When rounded objects are immersed in a fluid flow, vortices are formed and shed on their downstream side, creating a pressure imbalance resulting in an oscillatory lift. A convertor modulus consists of an elastically mounted, rigid cylinder on end-springs, undergoing flow-induced motion when exposed to transverse fluid-flow. These vortices induce cyclic lift forces in opposite directions on the circular bar and cause the cylinder to vibrate up and down. An experimental prototype was developed and tested in a free-surface water channel and is already able to recover energy from free-stream velocity between 0.5 and 1 m.s$^{-1}$. However, the large number of parameters (stiffness, damping coefficient, velocity of fluid flow, etc.) associated with its performances requires optimization and we choose to develop a complete tridimensionnal numerical model solution.

A 3D numerical model has been developed in order to represent the real system behavior and improve it through, for example, the addition of parallel cylinders. The numerical model build up was carried out in three phases. The first phase consists in establishing a 2D model to choose the turbulence model and quantify the dependence of the oscillations amplitudes on the mesh size. The second corresponds to a 3D simulation with cylinder at rest in first time and with vertical oscillation in a second time. The third and final phase consists in a comparison between the experimental system dynamic behavior and its numerical model.
1. Introduction

Development of micro-power plants is required by the desire to more efficiently exploit energy from renewable sources. Hydraulic energy recovery systems are mainly limited by the availability of sufficiently high speed of resources (higher than $2 \text{ m.s}^{-1}$), most of them basing their efficiency on high flow rates. This project uses vortex induced vibrations phenomenon and is able to produce efficiently electricity from flow speeds of the order of $1 \text{ m.s}^{-1}$, which corresponds to the most frequently encountered natural flows.

Vortex induced vibration has been studied for decades because of its potential to destroy immersed or exposed to the wind structures. Actually, if conditions are met, vortices may appear downstream of an object immersed in a flow. These vortices are accompanied by a cyclic force that will tend to oscillate the object, which may cause damages.

The idea, initially proposed by Bernitsas et al via the VIVACE project [1], is to build on these vibrations via adding a degree of freedom allowing vertical translation of the cylinder. This allows, on one hand, to recover energy efficiently in untapped rivers, and on the other hand to preserve the environment, because this system is able to limit its impact on both ecological and sedimentary continuities.

This system will be modeled in three dimensions and be compared with experimental results obtained with a prototype in a free surface channel. We aim at finding, in future, the optimum set of operating parameters in a given environment.

2. Background

2.1. Vortex induced vibrations

When an object, for example cylindrical with diameter $D$, is immersed in a flow of speed $v$, the flow regime is defined by the dimensionless parameter $Re$

$$Re = \frac{vD}{\nu}$$

Numerous authors [6, 4] studied the causal link between Reynolds number $Re$ and both drag and lift forces applied on the immersed body by the flow. It follows from these studies that there is a $Re$ range in which mean amplitude of lift coefficient has a maximum. The cylinder diameter has been designed in order to correspond to this optimum. In this case, if we want to produce energy from a flow range between 0.1 to 1 $\text{ m.s}^{-1}$, the diameter $D$ has been set to 0.11 m.

Lift force is defined, by cylinder unit length, such as

$$F_L = \frac{1}{2} \rho v^2 D C_L L$$

with $\rho$ density of fluid ($kg.m^{-3}$), $C_L$ lift coefficient and $L$ cylinder length ($m$). Vortex shedding phenomenon begin cyclic, lift force vary periodically at $f_v$ frequency, corresponding to vortex shedding frequency ($Hz$). Key point of this system is to amplify the motion amplitude induced by lift force. Furthermore, Strouhal number is defined by

$$S = \frac{f_v D}{v}$$

with $f_v$ vortex shedding frequency around stationary and immersed object. It is recognized that for $Re$ of the order of $10^5$ and for a cylindrical body, Strouhal number equals 0.2, which leads to $f_v = 0.9 \text{ Hz}$ for $v = 0.5 \text{ m.s}^{-1}$ and $D = 0.11 \text{ m}$.

2.2. Hydro-power conversion system

A vertical motion in translation is allowed to the immersed cylinder and limited by an elastic element. Fig 1 represents the prototype with variable spring ($a$), polley ($b$) and generator ($c$).
Figure 1. Basic diagram of the instrumentation, the control system and the mechanical part of the hydro-power conversion system.

For a mass-spring-damper system, the cylinder motion equation derived from the fundamental principle of dynamics is given by

\[ m \ddot{a} = - \mu \ddot{v} + k_s (l_0 - \bar{p}) - m \ddot{g} + \bar{F} \]  

(4)

with \( \ddot{a} \) the cylinder acceleration, \( \dot{v} \) the velocity, \( \bar{p} \) the position, \( m \) the mass, \( \mu \) damping coefficient induced by mechanical friction \((N.m^{-1}.s^{-1})\), \( k_s \) spring stiffness coefficient, \( l_0 \) its length for \( m = 0 \ kg \) and \( \bar{F} \) an external force applied to the system. Projected on ascendant axis \( z_1 \), equation 4 leads to

\[ m \ddot{z} = - \mu \dot{z} - k_s z + F \]  

(5)

\[ F = m \dddot{z} + \mu \ddot{z} + k_s z \]  

(6)

with \( z = 0 \ m \) for a spring length \( l = l_0 \). We deduce from (6), by way of a Laplace domain transform, dimensionless damping parameter \( \xi \)

\[ \xi = \frac{\mu}{2k_s} \sqrt{\frac{k_s}{m}} \]  

(7)

An experiment was conducted to determine \( \mu \) for mass-spring-damper system free oscillations in air. An automatic loop was then implemented in the motor controller to simulate low mechanical losses. The spring stiffness has been adjusted manually so that the oscillating system natural frequency \( f_{o,e} \) would be as close as possible to natural vortex shedding frequency \( f_v \). In future, a control modulus will be set up to adjust it automatically at any time.

\[ f_{o,e} = \frac{1}{2\pi} \sqrt{\frac{k_s}{m_{eq} + m_a}} \]  

(8)

with \( m_{eq} \) mass-spring-damper system equivalent mass \((kg)\) and \( m_a = C_a \rho_f V_{cyl} \) the added mass, \( \rho_f \) fluid density and \( V_{cyl} \) the cylinder volume. \( C_a \) corresponds to added mass coefficient \([5]\) that we assume to be equal to 1.
The vicinity between $f_v$ and $f_{oe}$ is sufficient to allow large cylinder oscillations. Actually, the lock-in (or synchronisation) phenomenon consists in the vortex shedding frequency tendency to naturally adjust with system frequency when they are close. This operation is made possible by an automatic control of spring equivalent stiffness, controlled by a stepper motor. The energy mechanically transmitted by the system is then recovered by imposing a resistive torque through a generator, opposed to the system motion in translation, via a belt.

2.3. System efficiency determination
We choose to impose a torque $C_g$ through the generator such as the force induced by the generator, $F_g$, is a function of cylinder displacement speed $v_c$

$$F_g = \frac{C_g}{r_p} = -k_pv_c^{n_p}$$

with $k_p$ and $n_p$ respectively the proportional and exponential coefficients, and $r_p$ radius of the polley. The force follows $z_1$ direction and is opposed to cylinder vertical translation.

Instantaneous power recovered by the generator $P_g$ is averaged over the duration of experiment $t_e$ to obtain the average power recovered $\bar{P}_g$.

$$\bar{P}_g = \frac{1}{t_e} \int_{0}^{t_e} P_g \, dt = \frac{1}{t_e} \int_{0}^{t_e} F_g v_c \, dt$$

Instantaneous hydraulic power $P_f$ is given by the integration of the fluid kinetic energy on the fluid area crossed by the cylinder. The average power $\bar{P}_f$ is then obtained by integration

$$\bar{P}_f = \frac{1}{t_e} \int_{0}^{t_e} P_f \, dt = \frac{1}{t_e} \int_{0}^{t_e} \frac{1}{2} \rho_f S u^3 \, dt$$

with $S$ the surface described by the maximum amplitude of the cylinder oscillations and $u$ fluid velocity. Average speed $u$ is given by $u = Q_v/lh$ with $Q_v$ the instantaneous flow rate, $l$ the channel width and $h$ height of free-surface. The power coefficient, or efficiency, $C_p$ is defined by

$$C_p = \frac{\bar{P}_g}{\bar{P}_f}.$$ 

3. Material and methods
Experiments were conducted in a free-surface channel, in ICube Laboratory, whose characteristics are given in right side of Table 1. The characteristics related to the prototype are given on the left side.

| Parameters          | Values          | Parameters          | Values          |
|---------------------|-----------------|---------------------|-----------------|
| Cylinder diameter   | 0.11 m          | Channel width       | 0.6 m          |
| Cylinder width      | 0.5 m           | Flow-rates available| 0 - 700 m³.h⁻¹ |
| Spring stiffness    | 400 - 3000 N.m⁻¹| Flow velocity       | 0 - 1 m.s⁻¹    |
| Mass ratio m*       | 1.22 - 2        |                     |                 |
| Equivalent mass $m_{eq}$ | 5.8 - 9.5 kg |                      |                 |
| Blockage ratio²     | 15 - 30 %       |                      |                 |

1 Equivalent mass is given by cylinder mass plus 1/3 spring mass.
2 Blockage ratio corresponds to the surface described by the cylinder during its oscillations divided by the channel wet surface without cylinder.
3.1. Conducted experiments

3.1.1. Velocity profiles measurement method

The tri-dimensional model of a stationary cylinder consists on a comparison between measured and calculated mean velocity profiles. Tests were conducted in order to study the effects of a cylinder immersed in the flow downstream. To do this, velocity fields were measured at different points. The measurement of these velocity fields has been made possible by the use of a ultrasonic sensor using pulse-pair method. This device, developed by ICube laboratory, allowed to record velocity profiles at a frequency of 12 Hz with a 8 mm spatial step or at a frequency of 8 Hz to a 4 mm step. Two sensors were used consecutively, the first one \(T_1\) transmitting an ultrasound signal in vertical axis and the second \(T_2\) at an angle of \(\theta = 75^\circ\) relative to the flow direction. Each sensor measures velocity component in its beam direction, respectively \(u_{T_1}\) and \(u_{T_2}\) (Fig 2).

![Figure 2. Velocity measurement method used.](image)

At this stage of the study, whereas the two velocity components have been measured neither at the same positions nor at the same time, instant horizontal velocity \(u\) could not be obtained. However, mean horizontal component in the time \(\overline{u}\) have been obtained by solving the following system

\[
\begin{align*}
\tau_{T_1} &= \overline{v} \\
\tau_{T_2} &= \overline{u} \sin \theta + \overline{v} \cos \theta
\end{align*}
\]  

(12)

The measuring system can be placed anywhere in flow direction \((x_1)\) or sideways \((y_1)\). In this article, transducers are always placed at a distance \(x_r = 31 cm\) downstream of cylinder axis under conditions given in Table 3. Four measuring points have been selected and are introduced in Fig 3.

Profiles obtained numerically at the same locations will be compared with experimental profiles. Similarly, a frequency analysis is performed both on numerical and experimental data.

3.1.2. Efficiency estimation

Another serie of tests was conducted in order, first, to quantify the efficiency of the system, and secondly to find the optimum coefficients for given stiffness and flow rate set. Tested parameters are \(k_p\) and \(n_p\) coefficients (see equation 9). In the current state of the project, the model should be able to properly represent the system behavior with \(k_p\) adaptable and \(n_p = 1\).

3.2. Numerical model setting

Numerical tests were conducted using the open-source software OpenFOAM. The first modeling phase involved a choice of turbulence model based on empirical method.
3.2.1. Turbulence model choice  The turbulence model choice was based among a selection of four models available on OpenFOAM and using two criteria:

- dimensional criteria which consists in the comparison with the average amplitude of the lift coefficients identified in the literature;
- temporal criteria which consists in considering the simulation time required to reach a steady state, from which an average steady vortex shedding cycle is emerging.

Existing data of $C_l$ was synthesized in a state-of-art article by (Blevins, 1990). A large range of $C_l$ was experimentally determined at a given $Re$ with different experimental conditions by several studies. The average amplitude of lift coefficient is experiencing a net increase for $Re$ between $10^4$ and $10^5$. Values of $C_l$ being larger in 2D than 3D cases [6], we decided to base our choice of turbulence model with $C_l = 1.2 \pm 0.3$ (which corresponds more or less to (Sin and So, 1987) data around $Re = 5.5 \times 10^5$, $\pm 25\%$).

Four turbulence models (1. oneEqEddy, 2. Smagorinsky, 3. kEpsilon and kOmegaSST) were compared with the same solver (pisoFoam) and in the same conditions. Smagorinsky and kOmegaSST perfectly fulfill the first criteria with mean $C_l$ values of respectively 1.05 and 1.26. With reference to the second criteria, it is important that the calculation time is not too long. In the case of two-dimensional stationary cylinder, computation time is not a problem but the model will be translated to three dimensions with dynamic mesh, which will considerably increase calculation time. The numerical test with Smagorinsky for turbulence model was the longest.

Our choice will be kOmegaSST (with $\omega = 0.003$ and $k = 0.5 \times 10^{-3}$) for its proximity to literature values and the fact it is at least two times faster than LES type models in our case.

3.2.2. Switching from 2D to 3D model  The 2D model also provides the basis for first numerical tests in the dynamic case. Solvers available with OpenFOAM software are chosen according to the characteristics of the studied model (presence of a free-surface with interFoam, dynamic mesh with interDyMFoam, etc.). It should nevertheless ensure that results obtained by a solver or another are the same regardless of its characteristics. To do this, three tests were conducted to switch from the first solver, pisoFoam, used when searching the turbulence model, to the one allowing cylinder motion near a free surface, interDyMFoam:

(i) A flow simulation without free-surface around a stationary cylinder using pisoFoam;
(ii) A flow simulation with a very high water level (low impact of free-surface) around a stationary cylinder using interFoam;
(iii) A last one with a water level similar to the second case around a cylinder set at rest by using a stiffness coefficient $k_s$ very high, using interDyMFoam.

These three cases have been studied with the same geometrical dimensions, the same number of meshes and with the use of turbulence model chosen above. Mean amplitudes of lift ($C_l'$) and drag ($C_d'$) coefficients and mean vortex shedding frequency ($f_{sv}$) were close (less that 5% relative error distinguishes $C_l'$ and $C_l$ coefficients obtained using pisoFoam of those found with interDyMFoam) in all three cases (Table 2), which means that solvers studied differ only in the possibility, on the one hand, to allow one or more degrees of freedom to an object (interDyMFoam) and on second hand to add an additional layer (interFoam allowing an air layer in addition to the initial water layer) and have little impact on calculation results.

3.2.3. Mesh distribution  Using a kOmegaSST turbulence model imposes a $y+$ value lower than 30, which is equivalent, with $Re = 5.710^4$, to impose a mesh size lower than a millimeter near the cylinder. The mesh is refined depending on its proximity to the cylinder. In upstream side,
Table 2. Results obtained with three solvers pisoFoam, interFoam et interDyMFoam.

| Solver            | $C_d^\prime$ (%) | $C_l^\prime$ (%) | $f_v$ (Hz) |
|-------------------|------------------|------------------|------------|
| pisoFoam          | 0.23             | 1.21             | 1.22       |
| interFoam         | 0.22             | 1.20             | 1.22       |
| interDyMFoam      | 0.22             | 1.20             | 1.22       |

Meshes are refined one time from a distance of 0.30 m from the cylinder, and a second time at the boundary layer (for a thickness of 3 cm) around the cylinder. In downstream side, meshes are also refined once, but over a greater distance than the upstream, namely 0.7 m to allow finer calculation in vortices forming zone. Beyond that, meshes are not refined.

4. Results and discussion
Experimental features are given in Table 3.

Table 3. Hydraulic data used in the model.

| Parameters          | Values        | Parameters          | Values        |
|---------------------|---------------|---------------------|---------------|
| Flow rate           | $335 \, m^3.h^{-1}$ | Experiment duration $t_e$ | 60 s         |
| Upstream flow velocity | 0.52 m.s$^{-1}$   | Distance $x_T$      | 0.31 m       |
| Reynolds            | $5,7.10^4$     | Cylinder vertical position | 0.125 m    |
| $k_s$               | $830 \, N.m^{-1}$ | Water depth        | 0.30 m       |
| Blockage ratio      | 30%            |                     |              |

4.1. Cylinder at rest
The following figures shows experimentally measured velocity profiles (Fig 4) for four lateral positions 31 cm downstream of the cylinder. Fig 5 gives the numerical velocity profiles for the same conditions.

Figure 4. Mean velocity profiles (on $x_1$) measured for different lateral positions, 30 cm downstream of the cylinder.

Figure 5. Mean velocity profiles (on $x_1$) calculated with the same flow and geometrical conditions.

Numerical profiles generally have the same characteristics than experimental profiles. The rise in mean velocity as it approaches channel wall is well represented. This reflects the fluid
tendency to preferentially flow to the cylinder edges in both experimental and numerical cases. In
the center is also found a significant mean velocity decrease downstream of the cylinder because
of the obstacle it represents.

The measurement system used further allows to recover the temporal evolution of the vertical
speed at a given point. This can be used to know vortex shedding frequency and (or) draw a
velocity fluctuation profile. Fig 6 represents the amplitude of vertical velocity fluctuations
between the channel bottom and free surface in both experimental and numerical cases. Lateral
position corresponding to $y_{TM=1} = 0.30$ cm (31 cm downstream of the cylinder) shows fluctuations
evolution depending on height : fluctuations are more pronounced as we approach the depth of
cylinder axis.

![Figure 6. Amplitude of vertical velocity fluctuations between channel bottom and free surface measured then calculated by the model (y = 0.30 m).](image)

Maximum velocity fluctuation is at a height of 0.12 m. Fig 7 shows the experimentally measured spectrum at that point. The total peak energy corresponds of an amplitude of about $0.27 \text{ m.s}^{-1}$ for a frequency of 1.31 Hz. This peak is considerably larger than Strouhal frequency that is supposed to be, under these conditions, of the order of 0.9 Hz.

![Figure 7. Spectrum of vertical velocity at various lateral positions (measured).](image)

Similarly, a spectrum was drawn via CFD results (Fig 8). Here, peak frequency is shifted from 1.31 Hz to 0.78 ± 0.05 Hz, which is at contrary slightly below Strouhal frequency anticipated $f_S = 0.9$. Equivalent Strouhal number $S$ is 0.17 ± 0.01. The discrepancy between measured and simulated frequencies on one hand and predicted Strouhal frequency ($S$ should be equal to 0.2 for a 2D cylinder) on the other hand can be explained by the really 3D character of the flow due to the low aspect ratio (around $L/D = 5$) of the used cylinder.
4.2. Cylinder in motion

Experimentally, we observe that mean velocity profiles around the cylinder still have a nearly identical appearance to those around cylinder at rest but are flatter. The numerical tests are limited to $m^* = 1.22 \text{ kg}$ and $k_s = 830 \text{ N.m}^{-1}$, which leads to $f_{o,e} = 1.41 \text{ Hz}$ in both experimental and numerical cases. Fig 9 represents the mean oscillations amplitude evolution of the cylinder in motion and power coefficient (or efficiency) for different $k_p$ values. Knowing that with increasing $k_p$ the force induced by the generator also increases, it is natural to see the oscillations amplitude decrease.

![Figure 9. Evolution of oscillations amplitude and efficiency for different $k_p$ values.](image)

Both numerical and experimental curves exhibit same features. The oscillations amplitude is greatly underestimated by the numerical model (by a factor 5), but once it is "corrected" the calculated $C_p$ fits with experimental data.

The difference between the system natural frequency in water $f_{o,e}$ and shedding frequency in the case of the cylinder at rest $f_s$ may explain this underestimation. Regarding Fig 11, there is the presence of a large amplitude peak at a frequency equal to $1.64 \pm 0.05 \text{ Hz}$, which is equivalent to twice $f_s$ exhibited by numerical data while experimental data exhibit $f_{motion} = 1.23 \text{ Hz}$. Frequencies $f_v$, calculated from lift coefficient variations, and $f_{motion}$, from cylinder oscillations, are equal. Table 4 summaries encountered vortex shedding frequencies for cylinder at rest ($f_s$) and natural body frequencies in water $f_{o,e}$.

| Frequency          | Numerical | Experimental |
|--------------------|-----------|--------------|
| $f_s \ (\text{Hz})$ | $0.78 \pm 0.05$ | $1.31 \pm 0.1$ |
| $S(f_s) \ (-)     $ | $0.17 \pm 0.01$ | $0.29 \pm 0.02$ |
| $f_{o,e} \ (\text{Hz})$ | $1.41$ | $1.41$ |
| $f_{motion} \ (\text{Hz})$ | $1.64 \pm 0.05$ | $1.23 \pm 0.1$ |

Conditions being the same for both numerical and experimental cases, the numerical body natural frequency in water $f_{o,e}$ has been arbitrary set equal to $1.41 \text{ Hz}$, which is not coherent with encountered numerical $f_s$. Consequently, $f_{motion}$ runs around $f_v$'s second harmonic, which spoils oscillations amplitude : to avoid this situation, it is necessary to approach lock-in frequency by changing $f_{o,e}$. This situation is unsatisfactory because it is necessary, in our study, to retain the same features.
Figure 10. Cylinder and vertical flow velocity spectrum from experimental data.

Figure 11. A/D and $C_l$ spectrum from CFD cases.

5. Conclusion
An energy recovery prototype was designed to build on vortex induced vibration around a cylinder. This should enable the recovery of kinetic energy in the case of rivers at low velocity while limiting its impact on environment. Its establishing was used to study the technical feasibility of such a system, particularly through the study of flown around submerged object and the estimation of its efficiency. Many parameters are related to the system efficiency and adjustment is a new attempt to resolve problems as could do a three-dimensional numerical model.

Such a model is currently able to properly represent flows around cylinder in real conditions in term of velocity fluctuations amplitude but not in term of frequency. However, it fails to represent the mechanical behavior of the oscillating cylinder in the right proportions due to this frequency misestimation.

This problem will lead in the future to a change in strategy for dynamic modeling (using a sliding mesh instead of deforming, using the chimera method, etc.) or adaptation of the current solution in the case that meshes are very fine (automatic remershing method from a certain threshold shift). Once the model is finalized, it will undergo sensitivity studies to find the optimal settings and implement them in the real prototype.

References
[1] Bernitsas M et al 2008 VIVACE (Vortex Induced Vibration Aquatic Clean Energy) : a new concept in generation of clean and renewable energy from fluid flow. Journal of Offshore Mechanics and Artic Engineering, 130
[2] Blevins R D 2001 Flow-Induced Vibration (2nd ed) Krieger Pub Co. chapter 3 p 63
[3] Sin V K and So R M C 1987 Local force measurements on finite span cylinders in cross flow. Journal of Fluid Engineering 109 pp 136-143
[4] Zdravkovich M M 1997 Flow Around Circular Cylinders Volume 1: Fundamentals. Oxford University Press (England)
[5] Vandiver J K 2012 Damping parameters for flow-induced vibration. Journal of Fluids and Structures 35 pp 105-119
[6] Sin V K and So R M C 1987 Local force measurements on finite-span cylinders in a cross-flow. Journal of Fluids Engineering 109 pp 690-697