Research Article

Dimitrios G. Koubogiannis*

Parametric CFD study of micro-energy harvesting in a flow channel exploiting vortex shedding

DOI 10.1515/eng-2016-0016
Received December 30, 2015; accepted May 6, 2016

Abstract: Miniature energy harvesting devices are increasingly used in various fields. For example, Wireless Sensor Networks have recently made great progress in many applications. However, their main drawback, i.e. the limited duration of operation, poses the requirement for an effective way to recharge their batteries. In this context, the present work focuses on the study of micro-energy harvesting from flow by exploiting vortex shedding behind bluff bodies, in order to cause oscillations to a piezoelectric film and generate the required electrical power. To this end, a Computational Fluid Dynamics (CFD) tool is validated on a particular miniature device configuration proposed in the literature and implemented for the numerical simulations of flow around bluff micro-bodies in a very small channel. Aiming to enhance vortex shedding, parametric studies corresponding to different bluff body shapes and arrangements for a fixed Reynolds number are performed, the main parameters involved in the phenomenon are highlighted and the potential for vortex shedding exploitation is qualitatively assessed.

Keywords: Energy harvesting, vortex shedding, bluff body, piezoelectric phenomenon, CFD

1 Introduction

Wireless sensor networks (WSN) have recently made a great progress in many applications. An important characteristic of them is their limited size. However, their main drawback is their limited life of operation unless an effective way to recharge their batteries is implemented [1]. WSN have been proposed and used in industrial process monitoring and control, machine health monitoring, structural health and bridge monitoring, predictive maintenance, environment monitoring, intelligent buildings, etc. Thus, micro-energy harvesting for powering WSN is a state-of-the-art research topic. WSN is just a characteristic application among others, where millimetre-scale energy harvesting devices could be implemented. Among energy sources offered for micro-energy harvesting (mechanical, electromagnetic, thermal, solar, etc), miniature pneumatic power systems that are used to convert flow energy to electricity either make use of micro-turbines [2, 3] or of bluff bodies [4]. In the former case, precise fabrication of millimetre-scale turbomachinery components is required, while the latter offers the advantage of much simpler design and ease of application [4]. In such a case, one or more bluff bodies are installed into the flow in order to produce vortex shedding behind them for a wide range of Reynolds numbers. The flow unsteadiness caused by vortex shedding provides pressure fluctuations which, in turn can be utilized by appropriate energy-converting materials, like piezoelectric membranes, to generate electrical power.

Application of the piezoelectric effect to harvest energy from flow can be realized by using flexible structures that can exploit fluid structure interaction phenomena. Various configurations using flexible membranes in combination with bluff bodies, in either external or internal flows, have been proposed and assessed in the literature [5–7]. In these works, fabricated prototype devices are presented and experimentally tested and/or numerical simulations of their operation is performed. Some of these studies concern the shape of the bluff body to be installed in a channel in order to continuously produce significant vortex shedding. This objective is the same with that for the design of an effective flow meter [8, 9]. An interesting perspective aiming to enhance vortex shedding relies on the use of multiple bluff bodies in tandem [10, 11]. A very good and compact literature review on these issues can be found in [4], according to which, further research is required to enhance power produced by such devices for practical use.

The present work uses a miniature pneumatic energy generating device proposed in [4] as a case study. In this device, one or two bluff bodies in tandem are installed in
a micro-channel having a flexible diaphragm located at its upper wall. Pressure fluctuations and unsteady forces due to vortex shedding are induced to the diaphragm which is connected to a piezoelectric film, vibrating and converting mechanical energy to electrical. Such a device can be used in liquid or gas pipeline systems. Its installation on the top of the flow channel is rather easy. The whole arrangement facilitates miniaturization and massive production avoiding the need for micro-assembling processes. However, the main drawback of such a device is its low power output and research is required to enhance its performance. Thus the test case under consideration, due to its simplicity in manufacturing, simple operation principle and ease in numerical modelling, is used as the departure point to appropriately setup and validate a CFD model for its study. This is required in order to understand the mechanism and the role of the involved parameters, gain experience on the physical characteristics of the phenomenon and assess the requirements and the limits of the numerical model. In contrast to [4], extensive use of the CFD model in parametrically studying the operation of the device is demonstrated herein. The costs of manufacturing required in the testing of numerous cases can thus be avoided or at least be postponed till the moment that an efficient configuration will have been designed. The corresponding study of the configuration in [4] is mainly experimental and focused on a particular configuration for which a prototype has been manufactured and tested; however it has the advantage of providing the capability to measure actual electric power output.

In the light of the above, the present study utilizes a CFD model to conduct some preliminary parametric studies. Different bluff body shapes and configurations are numerically simulated for a fixed Reynolds number. Pressure fluctuation characteristics (frequency and amplitude) are numerically predicted and the vortex shedding severity is assessed in terms of the maximum achieved unsteady pressure fluctuation. Results are presented and discussed and the potential for exploiting vortex shedding in energy harvesting applications is assessed. The aim at this stage of research was not intended to be exhaustive but aimed to setup a methodology in order to proceed to a deeper and thorough investigation at a next step. Thus, suggestions for future research mainly refer to the directions of further extending parametric studies, as well as to attempt to correlate power output to calculated pressure fluctuations.

2 Test case description

2.1 Definition of flow domain geometry

Figure 1 presents a configuration containing two triangular bodies installed in a very small flow channel. Above these two bluff bodies a flexible diaphragm has been installed. Under the action of the unsteady pressure produced by vortex shedding, the diaphragm causes vibrations to a piezoelectric film connected to it; the latter converts mechanical energy to electrical. The greater the pressure fluctuation amplitude acts on the diaphragm the better the performance of the device. Such a device has been fabricated, as well as experimentally and numerically tested in [4], where two different configurations, namely those using either one or two triangles in tandem have been comparatively studied.

Figure 1: Miniature energy harvesting device using two triangles in tandem (from [4]).

The flow domain used for the numerical simulations in the present study is defined in Figure 2. For comparison and validation purposes, the dimensions of the domain used in [4] have also been implemented herein. The channel length is \( L = 77.06D \) and its height is \( H = 3.76D \), where \( D = 4.25 \text{ mm} \) is the width of the bluff body. The flexible diaphragm is located on the upper wall at a distance of 23.53D from the inlet. The length of the diaphragm is 8.94D and after it, a wall length of 44.59D follows up to the outlet of the channel. The upper and lower boundaries of the channel are treated as solid walls, the left boundary is considered as the inlet of the domain and the right one as the outlet (Figure 2). In the same Figure 2, S denotes the center of the diaphragm, while BB1 and BB2 denote the sections where the bases of the first and the second bluff body, respectively, are installed. In the configurations used in [4], the bluff body was an isosceles triangle, the base of which faced the incoming flow (Figure 2).

2.2 Different bluff body configurations

In the present study, the following bluff body shapes were used in the simulations:
Table 1: Nomenclature and description of cases simulated in the present study.

<table>
<thead>
<tr>
<th>Incoming flow on</th>
<th>triangle-base</th>
<th>triangle-sharp</th>
<th>semi-circle</th>
<th>triangle-turned</th>
<th>rectangle</th>
<th>triangle-based</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of bodies</td>
<td>1 A1 B1 C1 D1</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>2 A2 B2 C2 D2</td>
<td>E2 F2</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

- the isosceles triangle mentioned above (to be called “triangle-base”)
- the same isosceles triangle installed in a way that its sharp edge faces the incoming flow, i.e. the “triangle-base” configuration turned by 180° (to be called “triangle-sharp”),
- a semi-circle with its circular arc shaped side facing the incoming flow (“semi-circle”),
- an orthogonal triangle turned 35° with respect to the vertical direction and pointing to the diaphragm (“triangle-turned”),
- a rectangular (“rectangular”).

In each case, two configurations were tested, namely those containing either one or two bodies of the corresponding shape, respectively. Whenever one bluff body was used, this was located at section BB1. In case of two bluff bodies, the second one was located in front of the first at section BB2 (Figure 2). Table 1 presents the nomenclature that was used for the set of the (ten) different test cases simulated in the context of the present study. The corresponding geometries are shown in Figure 3, where focused views of the computational grids in the vicinity of the bluff bodies are presented. In all cases but the last one (case F2), the width D of the bluff body was the same, corresponding for the channel under consideration to a blockage ratio $BR = D/H = 0.27$. The last case in Table 1 concerns the same geometry with A2, but the bluff body width D has been modified from 4.25 mm to 5.274 mm in order to achieve a blockage $BR = 0.33$ in the channel.
3 Numerical simulations

The ANSYS commercial CFD software FLUENT [12] (version 14.0.1) was used for the numerical simulations. The necessary geometry modeling and grid generation tasks were accomplished by means of the relevant ANSYS software modules. All the simulations were performed on a Personal Computer with Intel® Core™ i7 CPU 930@2.8GHz, 6Gb installed RAM and 64-bit operating system x64 based processor.

3.1 Governing equations and their numerical solution

The unsteady incompressible flow two-dimensional Reynolds Averaged Navier-Stokes (RANS) equations (continuity and momentum ones) were numerically solved. Based on the eddy-viscosity assumption for the turbulence modeling, the realizable variant of the k-ε two-equation model was used, ensuring that only physically realistic (realizable) viscous stresses will arise during the simulations [13]. The finite volume method was implemented, in conjunction with the SIMPLE pressure correction scheme. Second order accuracy was used for the convective terms of the mean flow equations, while first order was used for the turbulence model ones. Transient solution of the governing equations was sought by means of first order Euler scheme in time with a constant physical time step. Inlet velocity was prescribed at the inlet boundary. Zero pressure value was set to the outlet boundary. No slip conditions were used for velocity at all walls. Wall functions were implemented to model velocity profiles at wall boundaries. In particular, the enhanced wall treatment version provided by the software was used, that automatically switches to a two-layer low Reynolds approach and resolves the boundary layer up to the wall wherever a small value of y+(e.g. of the order of 10 or less) is met.

3.2 CFD model setup

Unstructured two-dimensional grids consisting of triangular elements were created in order to discretize the flow domain as shown in Figure 2. Figure 4 shows a representative picture of the grid in the vicinity of the bluff bodies for each of the ten cases presented in Table 1. The mesh generated for case B2 consists of 11244 triangles and 5970 nodes and this is the order of magnitude of the grid size for all other cases.

Air was used as the working fluid in the channel. An inlet velocity value \( V_{in} = 20.7 \) m/s was considered as in [4]. The air density was \( \rho = 1.225 \) kg/m\(^3\) and its dynamic viscosity coefficient was \( \mu = 1.789 \times 10^{-5} \) Pa.s. As characteristic length of the flow, the width of the bluff body \( D = 4.25 \) mm = 0.00425 m was considered, resulting to a Reynolds number \( \text{Re}_D = \rho V_{in} D / \mu = 6024 \) that dictates turbulent flow. The turbulence inlet conditions were prescribed by providing the turbulence intensity \( I_t \) and the turbulent length scale \( l_t \) at the inlet. The latter was estimated for fully developed flow by the formula \( l_t = 0.07D = 0.3 \) mm. Turbulence intensity was computed by the formula \( I_t = 0.16(\text{Re}_D)^{-1/8} = 0.054 = 5.4\% \). To assess the necessity of using the enhanced wall treatment version of the wall functions, the value of \( y^+ \) was computed for the various simulations. This was found to be in the range \( 15 \pm 75 \) for the upper and lower walls of the flow domain and about \( 1 \pm 7 \) at the vicinity of the bluff body walls. Thus, the use of the enhanced wall treatment version is justified, as
it is well known that the use of standard wall functions $y^+$ should ideally be in the range $30$ to $300$.

Concerning the flexible diaphragm, this was considered to be rigid wall in the simulations. Thus, fluid structure interaction phenomena were ignored, relying on the assumption that the diaphragm has small inertia and is able to oscillate with the frequency of vortex shedding, the piezoelectric film is strained laterally following the vibrations of the diaphragm and, according to the piezoelectric phenomenon, produces electrical power. By ignoring the fluid structure interaction phenomena, the displacement of the fluid due to the diaphragm motion is also ignored and feedback effects from the diaphragm to the flow are neglected. Although the actual geometry is three-dimensional, two-dimensional simulations along the symmetry plane of the channel were performed in the present study.

A constant time step was used in physical time. Estimating as a characteristic time value of the problem the convective time $T_c = D/V_{in} = 2.05 \times 10^{-4}$ s, a physical time step of $\Delta t = (0.05)T_c = 10^{-5}$ s (or 0.01 ms) was used in the unsteady simulations as in [4] for comparison purposes. At each case, the solution was marched in physical time for 8 ms (800 time steps) starting from an ambient initial velocity field. Such a time was enough to establish periodicity and let the flow field evolve for about three periodic cycles. The wall clock time required for each such run, executed on the above mentioned hardware, was about 2 hours and 15 minutes.

4 Results and discussion

4.1 Vortex shedding and velocity field

Figure 4 presents plots of the instantaneous velocity isolines at time $t=8$ ms for the various cases. In these pictures, asymmetries in the flow field can be observed due to vortex shedding. In any case, periodicity was established after an initial transient stage of about 5 ms, as it is shown in Figure 5 (left), where the pressure evolution on the center of the diaphragm in time has been plotted for cases A1 and A2. According to this figure, the period of vortex shedding is about 100 physical time steps that correspond to 1 ms. The corresponding Strouhal number (nondimensional frequency) is $St = fD/V_{in} = 0.2$.

4.2 Unsteady pressure evolution on the center of the diaphragm

The unsteady pressure evolution in time was computed and compared for each case at the center of the diaphragm $S$ (see Figure 2). The pressure fluctuation amplitude, i.e., the quantity $\Delta p = p_{\text{max}} - p_{\text{min}}$ attained during a vortex shedding period, was considered to be an indicative quantity of the vortex shedding severity.

Figure 5 (right) presents results for the pressure evolution at the center $S$ of the diaphragm for the cases A1 and A2 by the present study together with corresponding results from the literature [4]. These have been appropriately shifted in order to facilitate their comparison. As it can be noticed in this figure, the use of two triangles in tandem (case A2) significantly increases pressure variation amplitude (240 Pa) compared to that (165 Pa) caused by the one triangle (case A1). The present results produce pressure evolution curves that are similar with those of [4]. However, the present results predict a little greater pressure amplitudes, namely 165 and 240 Pa instead of 155 and 215 Pa, respectively.

Although the same CFD model (FLUENT commercial software) and case parameters have been implemented both in [4] and the present study, some differences in the two approaches could be pointed out that may explain the discrepancies in the corresponding results. These are: the different density of the computational grid (the present one is denser near the bodies compared to the upper and lower walls, while that of [4] is isotropic everywhere), the enhanced wall treatment variant of the wall functions used herein, as well as the inlet values for the $k$ and $\varepsilon$ variables (the authors in [4] do not provide $y^+$ values for their simulations, neither inlet values for $k$ and $\varepsilon$).

Figure 6 (left) depicts the pressure evolution at the center $S$ of the diaphragm for the cases A1, B1, C1, D1 corresponding to the four different obstacle shapes and containing only one such body. It is evident that case A1, in which the base of the triangle faces the incoming flow, causes the greater pressure amplitude at $S$. The same triangle pointing the incoming flow with its sharp edge (B1) produces a pressure evolution of smaller amplitude, while the other two bodies, namely the semi-circle (C1) and the orthogonal triangle turned by $35^\circ$ (D1), both cause variations of much smaller amplitudes.

Figure 6 (right) presents the corresponding pressure evolutions for the cases A2, B2, C2, D2, E2 containing two bluff bodies. Again, the use of triangles facing the incoming flow with their base (case A2) is the most effective in enhancing vortex shedding. The two rectangles in tandem produce the next significant pressure variation in terms of
Table 2: Pressure amplitudes predicted at each case.

<table>
<thead>
<tr>
<th>Case</th>
<th>A1</th>
<th>A2</th>
<th>B1</th>
<th>B2</th>
<th>C1</th>
<th>C2</th>
<th>D1</th>
<th>D2</th>
<th>E2</th>
<th>F2</th>
</tr>
</thead>
<tbody>
<tr>
<td>(\Delta p) [Pa]</td>
<td>165</td>
<td>240</td>
<td>75</td>
<td>35</td>
<td>35</td>
<td>35</td>
<td>28</td>
<td>23</td>
<td>100</td>
<td>335</td>
</tr>
</tbody>
</table>

Figure 6: Pressure evolution at the center S of the diaphragm for the cases A1, B1, C1, D1 containing one bluff body (left) and for the cases A2, B2, C2, D2, E2 containing two bluff bodies (right).

its amplitude and the results by the shapes of cases B2, C2 and D2 follow.

Table 2 summarizes the pressure amplitudes \(\Delta p\) predicted for each of the cases that were simulated. According to this table and Figure 6, all other configurations apart from A (i.e. B, C, D, E) produce much smaller pressure amplitudes than those obtained in cases A1 and A2. In addition, the results in cases C1 and C2 are practically the same, i.e. the use of the second body does not affect vortex shedding severity, while in cases B2 and D2, less severe pressure variations are predicted compared to B1 and D1, respectively, i.e. the use of the second body suppresses the variation amplitude instead of enhancing it.

4.3 Unsteady pressure evolution for different positions of the diaphragm

According to the above results, the configurations causing the more severe pressure variations among those tested, are A1 and A2. In addition, the use of the second body in the latter significantly enhances vortex shedding compared to the former. All the studied pressure variations refer to the center of the diaphragm S, which has been arbitrarily located at the position described in [4] and adopted herein. However, apart from this definition, the exact diaphragm installation should be such that its center S is located at the position of the maximum pressure variation to better exploit the corresponding membrane vibrations.

In order to assess the proper location of the diaphragm for better performance of the device, 11 points along the upper wall, at positions other than S, were considered along the original diaphragm for each of the cases con-
containing two bluff bodies in tandem. The locations of these points, denoted by p0, p1,...,p10, are defined in Figure 7. Pressure variation amplitude at these points was evaluated and compared to the corresponding value obtained at S. Such a study was realized not only for case A, but for all the configurations to ensure the validity of the above statement concerning the superiority of configuration A.

The unsteady pressure evolution in time that was computed at these 11 points for case A2 are shown in Figure 8. The corresponding curves are presented in two diagrams. The first of them in Figure 8 (left) shows the pressure evolution at points p0,...,p6, namely at the points located upstream the original diaphragm center S (S is identical to point p6). The second one in Figure 8 (right) shows the corresponding curves at points p6,...,p10, which are located downstream of the diaphragm center S. For the purpose of comparison, in both diagrams the pressure evolution at the diaphragm center S (p6) which is considered to be the reference one has been highlighted (using points along the plot of its curve). Referring to Figure 8 (right), a first remark is that the points located downstream the center S exhibit smaller pressure variation amplitudes than the reference one. Thus, the center S should not be placed downstream its current position. By carefully examining Figure 8 (left), it seems that all points located upstream of S exhibit pressure variation amplitudes that, at a first glance, are comparable to that of S. The pressure amplitude values predicted at each of these points p0,...,p6 are summarized in Table 3. According to this table, points p3 and p5 located upstream point S (p6) exhibit a significant increase of the pressure variation amplitude that is of the order of 46% compared to the reference one ($\Delta p_{S,A2} = 240$ Pa).

Based on the main conclusion concerning case A2, i.e. that the maximum pressure variation amplitude is attained at a point upstream the diaphragm center S, similar studies were also performed for cases B2 (Figure 9), C2 and D2 (Figure 10). According to Figure 9 concerning case B2, points located downstream of S, exhibit pressure amplitudes less than that of S ($\Delta p_{S,B2} = 35$ Pa), while for points upstream of S amplitudes are comparable to that of S. In particular, points p3 and p5 exhibit $\Delta p$ comparable to $\Delta p_{S,B2}$, while the value at point p4 (55 Pa) supersedes it for about 57%.

Figure 10 (left) presents the pressure evolution for all of the points p0,...,p10 in one diagram for case C2, while Figure 10 (right) presents the corresponding results for case D2. In case C2 (2 semi-circles in tandem), all the curves exhibit pressure variation amplitude comparable to that of the center S ($\Delta p_{S,C2} = 35$ Pa) and no point, either upstream or downstream of S, exhibits a noteworthy increase. Similarly, in case D2 no point downstream of S exhibits greater amplitude, while for p3, p4 and p5 upstream of S, an increase of 117% is predicted (for example at point p4 where the amplitude is 50 Pa compared to the $\Delta p_{S,D2} = 23$ Pa of the center S).

Figure 11 presents similar results for case E2 (two bodies of rectangular shape). In particular, Figure 11 (right) shows the pressure variation for points downstream of S, all of them having a smaller value than that of S ($\Delta p_{S,E2} = 100$ Pa), while Figure 11 (left) shows the corresponding variations for the points upstream of S. Again, the conclusion is the same, i.e. points p3, p4, p5 exhibit a significant increase over $\Delta p_{S,E2}$ (for example at point p4, where the amplitude is 155 Pa, this is of the order of 55%).

The main conclusions up to this point are:

- the triangular body shape located as in cases A1 and A2 causes the more severe pressure variations,
- the existence of the second body enhances the phenomenon,
- the maximum pressure variation amplitude occurs upstream the current position of the diaphragm center S.

It should be mentioned that these findings are in accordance with similar findings and results in the literature [4]. In [9] it is also stated that the maximum pressure variation occurs at the point where the minimum absolute pressure is located. This fact seems also to be valid herein, at least in case A2 (Figure 8 (left)), where the most significant results have been obtained.

### 4.4 Unsteady pressure evolution for different blockage ratios

According to the literature [8], the blockage ratio (BR) plays an important role in the pressure variation ampli-
Figure 8: Pressure evolution for case A2 along the diaphragm at points p0, . . . p6 (left) and p6, . . . p10 (right).

Table 3: Pressure amplitudes predicted for case A2.

<table>
<thead>
<tr>
<th>Point</th>
<th>p0</th>
<th>p1</th>
<th>p2</th>
<th>p3</th>
<th>p4</th>
<th>p5</th>
<th>p6</th>
</tr>
</thead>
<tbody>
<tr>
<td>∆p [Pa]</td>
<td>235</td>
<td>250</td>
<td>245</td>
<td>355</td>
<td>280</td>
<td>350</td>
<td>240</td>
</tr>
</tbody>
</table>

Figure 9: Pressure evolution for case B2 along the diaphragm at points p0, . . . p6 (left) and p6, . . . p10 (right).

Figure 10: Pressure evolution along the diaphragm at points p0, . . . p10 for cases C2 (left) and D2 (right).

tude. In the region near the point where maximum pressure variation occurs and pressure is minimum, the velocity accordingly becomes maximum. So, to achieve even smaller pressure values and, consequently, greater pressure variation amplitudes, greater nearby velocity should be attained. The latter could be achieved by decreasing the flow area or, equivalently, by increasing the width D of the bluff body leading to an increase of BR. In study [8], a BR value of 0.30 was experimentally found to be optimum, while in [4], a BR value of 0.33 was numerically found to cause the greater increase in pressure amplitude for the device under consideration. Besides the above reasoning, in practice, values of BR greater than the optimum may inhibit the vortex shedding phenomenon and lead to its sup-
pression, due to the proximity of walls in smaller distances to the bluff body.

In order to check the above statement for the optimum BR value, the case A2 (exhibiting the best results up to this point) was simulated again in a channel with BR = 0.33 instead of the original one (in which BR was 0.27) by increasing the triangle base length D from 4.25 mm to 5.274 mm (case F2 of Table 1). Figure 12 presents the pressure variation results for the different points along the diaphragm. Qualitatively, the conclusions are the same, i.e. upstream the center S the amplitudes are greater and downstream less pronounced, but quantitatively the results are impressive since the increase becomes important enough. Table 4 summarizes the pressure amplitude values for the same points upstream of S for both cases A2 and F2 (similar configurations with BR 0.27 and 0.33, respectively). The case with greater blockage (F2) exhibits a mean increase of 50% over the amplitudes of case A2.

5 Conclusions - future research

A millimeter-scale energy harvesting device proposed in the literature was used herein as a test case, in order to appropriately setup and validate a CFD model for the numerical simulation of its performance. This simulation tool was implemented to study the exploitation and enhancement of vortex shedding in micro-energy harvesting applications, where vortex shedding causes oscillations to a flexible diaphragm and converts flow energy to electrical by the piezoelectric phenomenon. Different bluff body shapes and configurations located in a very small flow channel were numerically simulated for a fixed Reynolds number and the achieved vortex shedding severity was assessed in terms of the unsteady pressure fluctuation.

The main conclusions of the present study are that the shape of the bluff body is very important to achieve significant pressure variations, as well as that the design of the whole configuration (how many bodies to use, in what arrangement, with what blockage ratio, where to install the flexible diaphragm) is very crucial in order to harvest significant amounts of energy.

Although, the enhancement of pressure fluctuations constitutes a qualitative criterion for potentially generating electric power, quantitative results are required. So, future work steps refer to:

- implementation of a model for the piezoelectric phenomenon that will provide the ability to correlate
pressure fluctuations with the electric power and essentially model and assess the actual performance of such a device,

– simulations for a range of different Reynolds numbers,

– more detailed parametric studies in terms of different geometric configurations (distance between bodies, different bodies in tandem, asymmetric installation in the channel),

– statement and solution of general design optimization problems concerning the bluff body shape and location, as well as the diaphragm location,

– consideration of fluid structure interaction in the simulations (that requires modelling of the simultaneous diaphragm motion).

The aim in the long-term is to develop a simulation tool, appropriate for the study and design flow energy harvesting micro-devices.

### Table 4: Comparison of pressure variation amplitudes cases A2 and F2 at points upstream of S.

<table>
<thead>
<tr>
<th>Point</th>
<th>p0</th>
<th>p1</th>
<th>p2</th>
<th>p3</th>
<th>p4</th>
<th>p5</th>
<th>p6</th>
</tr>
</thead>
<tbody>
<tr>
<td>∆p[Pa], A2</td>
<td>235</td>
<td>250</td>
<td>245</td>
<td>355</td>
<td>280</td>
<td>350</td>
<td>240</td>
</tr>
<tr>
<td>∆p[Pa], F2</td>
<td>325</td>
<td>390</td>
<td>385</td>
<td>500</td>
<td>470</td>
<td>525</td>
<td>335</td>
</tr>
<tr>
<td>% increase</td>
<td>38%</td>
<td>56%</td>
<td>57%</td>
<td>41%</td>
<td>68%</td>
<td>50%</td>
<td>40%</td>
</tr>
</tbody>
</table>

The references are as follows:


