Available online at www.sciencedirect.com

ScienceDirect

Procedia Engineering 109 (2015) 162 – 170

Procedia Engineering

www.elsevier.com/locate/procedia

XXIII Italian Group of Fracture Meeting, IGFXXIII

Dynamic effects of wind loads on a gravity damper

L. Scappaticci[a,*], F. Mariani[b], N. Bartolini[b], F. Risi[b], A. Garinei[a]

[a]DMII, University Guglielmo Marconi, Roma 00193, Italy
[b]Department of Engineering, University of Perugia, Perugia 06125, Italy

---

Abstract

The gravity damper is safety device used for the air treatment that prevent overpressure inside the unit through the opening. It is a normally closed valve under the effect of the gravity force, which, under the action of the incident air flow, allows to manage any excess mass. Clearly, although the device is rather simple and therefore reliable, the operating conditions may prove burdensome, especially if the gravity dampers are applied to installations of energy transformation, such as the gas turbines; this is mainly due to the need to develop large masses of air at speeds rather incurred. This article describes an experiment carried out on a gravity damper designed to be installed in a gas turbine. The characterization has been performed in numerical (CFD-FEM), considering both the mode shapes and the natural frequencies of the device in working condition as well as any phenomenon of detachment of the fluid that can trigger vortex shedding and subsequently validated in the wind tunnel facilities of the University of Perugia. In particular, what is wanted to be highlighted is the fact that, after a preliminary analysis, it has been clearly evident that, under the operating conditions, the structure would be affected by phenomena of vortex shedding. The shedding frequency is next to some natural frequencies of the structure, with obvious repercussions on the integrity of the structure. An experimental vibration analysis performed in the wind tunnel at flow regime has in fact allowed to identify the phenomenon of lock-in.

Peer-review under responsibility of the Gruppo Italiano Frattura (IGF)

Keywords: VIV, vortex vhedding, lock-in, CFD, wind tunnel testing, FEM, fluid-structure interaction;

---

1. Introduction

The gravity dampers are safety devices used in the treatment of the air that allow you to prevent overpressure inside the main unit by a one-sided opening. It is normally closed valve under the effect of the gravity force, which allow to dispose of any excess of air mass when the flow rate reaches a certain value. From the constructive point of view, a gravity damper is constituted by a square or rectangular frame, which allows the housing inside the pipe that needs to be adjusted, and by a number of sections, which, in conditions of closure, occlude the area of the main duct. In the heaviest applications, the closing sections are interconnected by means of a four-bar linkage mechanism that allows the synchronous movement of all the elements. These, consisting in rectangular shaped AISI sheets, connected to a square cylinder shaft (200 *mm* each side) by means of bolts and nuts (Figure 1). Each section is linked to the main frame by means of bearings to enable the opening mechanism. The structure of the quadrilateral can

---

* Corresponding author. Tel.: +39-075-585-3945.
E-mail address: l.scappaticci@unimarconi.it

Peer-review under responsibility of the Gruppo Italiano Frattura (IGF)
doi:10.1016/j.proeng.2015.06.228

be equipped with masses arranged so as to facilitate the opening or closing of the mechanism. The mechanism, as reliable due to its robustness and agility, may be subjected to severe working conditions, as happens in cases where gravity dampers are applied for the air treatment in energy conversion systems, such as gas inlet turbine engines pipes. Here, in fact, the need to develop large masses of air at rather high speeds suggests that some structural problems, particularly of dynamic nature, must be taken into account. Not surprisingly, the UNI EN 1751:2003 (lapsed in March 2014) in this context suggests the general guidelines for the design and experimental validation. Due to the shape of the single section, the structural problems that may occur in operating conditions are those of flattening and/or fluttering that vortex shedding can trigger. The vortex shedding is an aeroelastic phenomenon for which, if a body is immersed in a subsonic fluid current, under certain conditions related to the Reynolds number (Re), it forms a wake of alternating vortices that detach from the body itself. This separation generates aerodynamic forces that vary over time, even if the speed of the incident flow is constant in intensity and direction. If the shedding frequency of the vortices is close to a resonance frequency of the structure, it generates a synchronism (lock-in) that involves the oscillation growth. Referring to the scientific literature, the lock-in is a well-known phenomenon related to the vortex shedding [1]. For a bluff body immersed in a fluid stream and forced to vibrate, the vortices shed in coupling with the excitation frequency. When the body is not forced to vibrate from external causes (as the case of gravity damper is), the vortex shedding phenomenon occurs in a particular frequency, mainly dependent on the velocity of the flow and on the dimensions of the body (Strouhal frequency) [2]. When the vortex shedding frequency is sufficiently near to one of the body frequencies [3], it occurs the lock-in phenomenon, where the shed of vortices excite the natural frequency involved. This can lead to a considerable increasing of the amplitude of the vibration that could damage the structure. Vortex shedding and wakes behind blunt bodies have been subject of numerous articles since 1960 [4]. From an experimental point of view, at earliest stages, hot wire anemometer and pressure trasducers, together with drag and lift relief, have been the most common methods used to characterize the vortex shedding phenomenon. Thanks to PIV (Particle-Image-Velocimetry) and CFD many of the earlyer doubts have been answered in literature. Around 1990 were pubblished the first applications of PIV to the study of the vortex shedding [5] [6]. In the same time appear the first applications of CFD to the analysis of the flow behind a bluff body [7], but it is only since the earliest 1990s that this type of numerical solutions of the problem began to be accured. The occurrence of vortex shedding phenomenon and the prediction of induced vibrations on the structure (VIV) is one of the issues of greatest interest in the numeric field. The great complexity of the phenomenon, coupled with the large computational cost, focus the research on simple bodies. In [8] Murakami et al. analyzed the vortex shedding phenomenon of a square section cylinder through URANS and LES simulations for low values of Re number, and validated the results with experimental acquisitions. The results show that the URANS model doesnt reproduce correctly the vortex shedding which, instead, is best modeled by the LES at the expense of a greater use of hardware resources. The same results were achieved in [9] by Catalano et al. and in [10] by Bouris et al. In [11], Rocchi et al., analyzed the influence of two wires helically wrapped on a cylindrical body on the vortex shedding phenomenon. The simulations are being carried out using 2D and 3D LES model and the results have been validated by experimental values. Also in this case the results confirm the need to achieve a LES 3D simulation to better model the phenomenon of vortex shedding. In [12] Liu et al, characterized, by numerical analysis, the influence of the leading and trailing edge and the length of a 2D bluff body, on the value of the chord-based Strouhal number (Stc). The characterization of vortex shedding phenomenon, around bluff body, was also developed by Nakayama et al. in [13]. The influence of vortex shedding on vibrations of a box girder bridge section was realized in [14] by Sarwar et al. by LES 3D analysis. The adoption of countermeasures aerodynamic allowed to reduce the amplitude of vortex induced vibration. When dealing with structures and wind action, wind induced vibration are referred as vortex-induced vibration (VIV). Many experimental studies were conducted to investigate the vibrational behaviour of structures caused by the acting of aerodynamic forces and vortex shedding phenomena and different measuring systems are presented to evaluate the amplitude of forces and vibrations. In [15] wind tunnel tests are conducted on a model of a bridge section and the displacements are measured by linear and rotational encoders. In [16] a wind tunnel study is presented on a body of rectangular section; to investigate the correlation between fluctuation of forces acting on the body and the pressures on its surface, a pressure scanner is connected by silicon pipes to arrays of holes on the external area of the body. Load cells are interposed between the model and the supports to measure the forces. The same kind of pressure measurement is described in [17] for a model of bridge. In [18] the behaviour of the wake behind a squared cylider is analyzed in a wind tunnel. The attention was focused on the vortex shedding phenomenon and different measurement techniques are used, as PIV with smoke-wires pattern, hot wire anemometer and pressure mapping of the surface by a pressure scanner. In [19] a Spanwise-segmented dielectric-barrier discharge plasma actuators were mounted on the cylinder in a square wave pattern to locate forcing regions. The possibility of detecting the vortex shedding phenomenon throught the use of accelerometers has been widely exploited over years; in [20] and [21] and the signal of accelerometers is used detect vortex shedding events on a large bridge. In [22] 15 accelerometers are used to monitor the modal behaviuor of a steel wind turbine tower exposed to wind excitation. Mukundan et al. [23] used data collected by strain gauges and accelrometers to developed a fatigue damage estimation method on marine risers. Marcollo et al [24] built an experimental facility to replicate and investigate the VIV occurring on a long flexible pipeline exposed to ocean currents. Accelerometers were used and a lock-in was experienced. This paper concerning the VIV acting on the closure elements of a gravity damper. It is divided into 2 main sections. The first part is focused on numerical analysis; in particular a CFD preliminary analysis was performed to understand the vorticosity; since the idea was to characterize the shedding through an indirect meausure, it was found useful for the phenomenon identification, the support of an accurate CFD analysis. A modal analysis was then implemented to obtain modal shapes and natural frequencies. In the second section an experimental test in the wind tunnel is described.

The objective of the paper is to give a numerical and experimental identification of the vortex shedding acting on a simple structure that has an effective application, putting in evidence the amplitude that accelerations can reach in the case of lock-in at high Re number. In figure 1 is reported the geometry of the closure section under analisys.

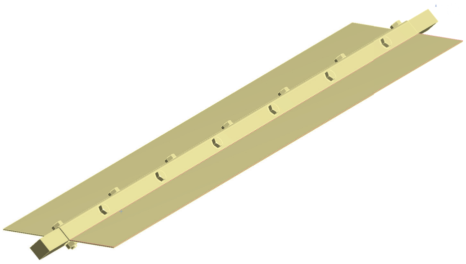


Fig. 1. Closure section, 1.5 *x* 155 *x* 998 *mm*

2. CFD Analysis

A preliminary numerical analysis was performed; it has been addressed using an approach that try to evaluate the interaction between the structure and the air with the main goal consisting to find out, by CFD-3D simulations, the occurrence of the phenomenon of vortex shedding. The physical quantity evaluated in the simulations and used to evaluate the vortex shedding is the lift coefficient (C_L); in particular the fast Fourier transform executed on the signal representing the trend of the average lift, exerted on the structure under analysis.

2.1. Simulation setup parameters

To contain the computational time it has been defined a quasi-2D geometry; this was possible thanks to the fact that the phenomenon develops mainly in one direction while along the third dimension can be exploited the attribute of symmetry thus limiting the depth and thus the number of cells. A segregated incompressible flow was considered. The total simulation physical time is equal to 1 s. Transient analysis with a time step of 10^{-4} s with a consequent total number of time-steps equal to 10000. The simulation was made on 12 Pentium cores, with an average duration for each time-step equal to 16.5 s/time-step and a consequent elapsed time equal to 46 h. In the model there are 1600000 polyhedral cells with a size that ranges from 0.5mm (in the core flow) to 2 cm (in the external flow). The mesh boundary layer (near the skin of the wing) is made by 10 layers spreaded on 1 mm with a ”prism layer stretching” set to ”geometric progression” equal to 1.5 and the thickness of the first layer equal to 8.8 10^{-3} mm. Three different situations were analyzed in order to consider the three different configurations of the section as: bolt turned back, bolt facing forward, without bolt. The boundary conditions at the inlet were defined with a velocity equal to 45 m/s, that is the 150% of the nominal operative condition as the UNI EN 1751 : 2003 suggests. The considered air density is $1.238kg/m^3$ and the temperature is $300K$; the outlet boundary is defined as a pressure boundary with atmospheric value of $1bar$. Together with the mesh and the time-step value, for this type of problem fundamental is the choice of the turbulent model. To obtain a very refined description of the shedding vortex phenomenon that occurs into the core of the flow, the LES (Large Eddy Simulation) model was chosen.

2.2. The Large Eddy Simulation model

Large-eddy simulation relies on the idea that some scales of the full turbulent solutions are discarded to obtain a desired reduction in the range of scales required for numerical simulation. More precisely, small scales of the flow are supposed to be more universal (according to the celebrated local isotropy hypothesis by Kolmogorov) and less determined by boundary conditions than the large ones in most engineering applications [25]. The LES is a mathematical model [26] [27] used in computational fluid dynamics (CFD) for the study of turbulent phenomena. It stands as a happy medium between a Reynolds averaged Navier-Stokes (RANS) modeling (faster but more approximate) and a direct numerical simulation (DNS, more accurate but prohibitive in terms of computational time). In the phenomena of turbulence within the fluid will generate large vortical structures, which transfer their energy to gradually ever smaller structures, up to the smaller structures that dissipate this energy. This complex phenomenon is known as the Richardson cascade. Unlike RANS models (those currently in use), with the LES it is possible directly simulate the larger vortex structures, and rely on modeling only the smaller vortex structures, those that dissipate energy, called Kolmogorovs. It is therefore a method more accurate than the RANS.

2.3. Numerical vortex shedding identification

The main results of the simulations are represented in Figures 2, 3, 4. These show the scalar velocity magnitude in a plane that contains the axis of the bolt. It is easy to individuate the formation of the vortices around the object and their evolution into the control volume. In particular, as expected in all three cases, it is easy to observe the vortices propagation after their detachment from the wake.

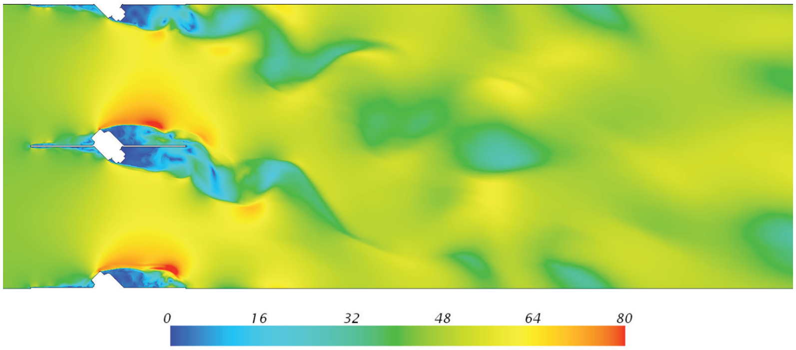


Fig. 2. Velocity Magnitude (m/s), bolt facing forward

3. The FEM model

A numerical modal analysis was performed to identificate both the modal shapes and the natural frequencies of the system. The pressed steel part of the wing was implemented as a surface model (SHELL 181 structural element), while for the axle, bolts and nuts, $SOLID187$ elements where chosen . The contacts between all elements were modeled as welded. Fixed support was considered near the ends of the axis. All components are in $AISI$ 316L. The modal model

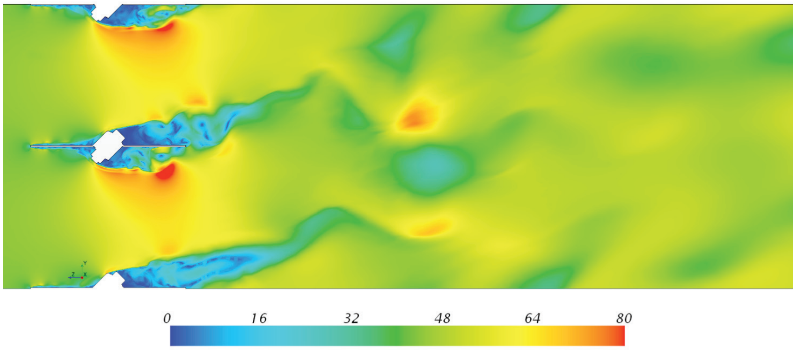


Fig. 3. Velocity Magnitude (m/s), bolt turned back

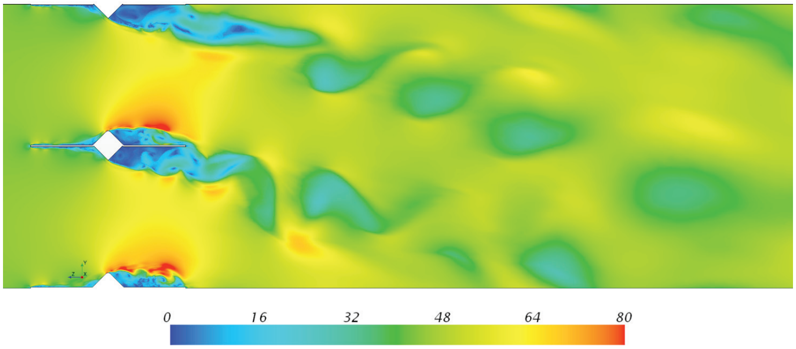
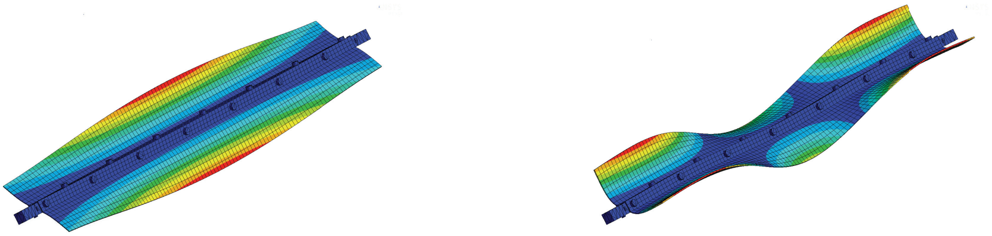


Fig. 4. Velocity Magnitude (m/s), without bolt

consisted in 25967 elements and 6336 nodes. Since we were interested to such phenomenon occurring in the frequency band $200 - 400\ Hz$ the modal analysis was performed to obtain the first 10 modes; the 10^{th} mode occurred at $343\ Hz$. If we exclude the first and the third mode, whose deformed regards the bending axis on mutually perpendicular planes, all other vibration modes relate to the flatter of the metal sheet; clearly the growth of the frequency produces a growth of the number of nodes. The second mode of vibration regards the inflection to 3 nodes of the axis and also involves the sheet.



(a) (b)

Fig. 5. 4^{th} modal shape, $244\ Hz$ (a), 6^{th} modal shape, $303\ Hz$ (b)

Table 1. Numerical natural frequencies

Mode	Frequency (*Hz*)
1	84
2	217
3	241
4	244
5	293
6	303
7	311
8	320
9	329
10	343

4. Wind tunnel testing

A wind tunnel investigation was carried out on the gravity damper in the wind tunnel facilities of the University of Perugia. The objective of the investigation was to evaluate the effect of forcing due to the wind load on the device.

4.1. The wind tunnel of the University of Perugia

The facilities at the University of Perugia also include a Wind Tunnel in closed loop configuration. An impeller with 11 blades in carbon warped shell, can accelerate the air to a maximum speed of 55 m/s. In each curve there are fixed blades to enhance the air flow pattern. The impeller is driven by a 375 kW electric motor. Just prior to the test room (approximately 4 m^2 of effective section), the turbulence of the flow is conditioned and reduced by means of a honeycomb structure. The flow is further accelerated by a converging duct into the testing room. Both the static and dynamic air conditions are monitored and correlated with the aerodynamic forces; two Pitot tubes monitor air speed that runs over the tested body, while a control station positioned in the testing room provides data about static air pressure, temperature and relative humidity.

4.2. Experimental setup

The gravity damper analysed in this work is composed of four full scale closure sections (Figure 1). The body was placed on the aerodynamic scales respecting its opening orientation. The wind tunnel tests were carried out in 70 seconds each, which included start up, data logging at regime, and ramp down. The tests were divided into three steps:

- rising speed ramp (20 s);
- regime (40 s);
- decreasing speed ramp.

The tests were carried out at a regime speed value of 45 m/s and were thrice replicated. One of the middle closure section was instrumented using 5 piezoelettric accelerometers. Signals were acquired at 5 kHz and then low pass filtered at 2 kHz.

5. Results

Figure 6 shows the time history of the accelerometer placed in the middle of the closure section. Figure 7 shows the

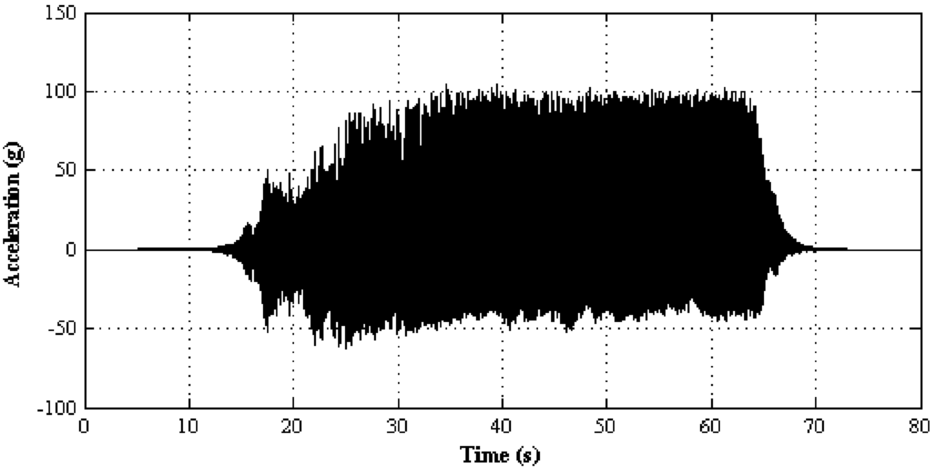


Fig. 6. Time History of the accelerometer

FFT of the of the time histories of C_L coefficients evaluated in the three section configrations by the CFD model: the peak of frequency lies in the frequency band 235 – 264 Hz. The Fast Fourier Trasform of the measured acceleration, represented in the waterfall diagram of figure 8, reveals that, at regime speed, the vibration has three contributions (239 Hz, 270 Hz and 298 Hz).

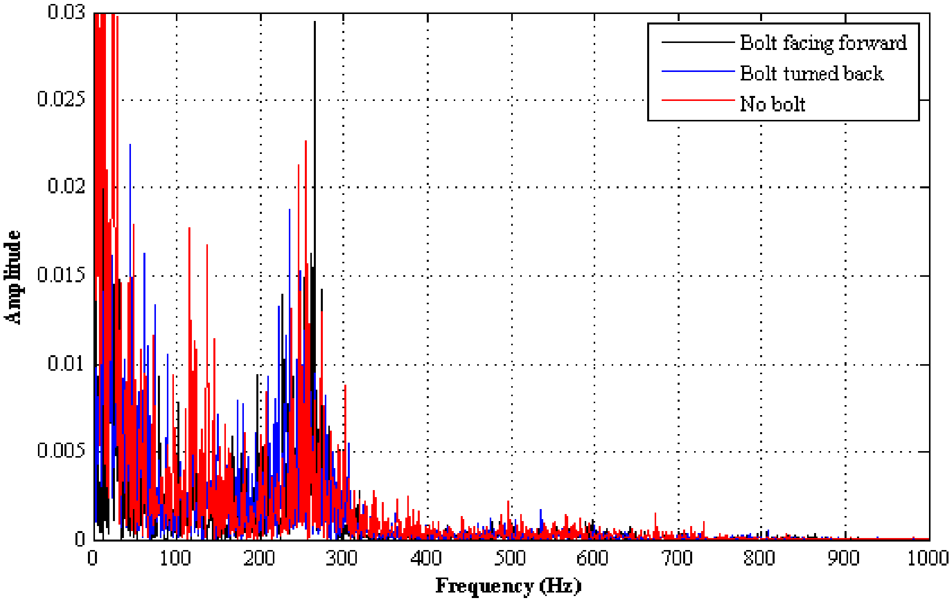


Fig. 7. FFT of the C_L behaviour for the three sections analized in CFD

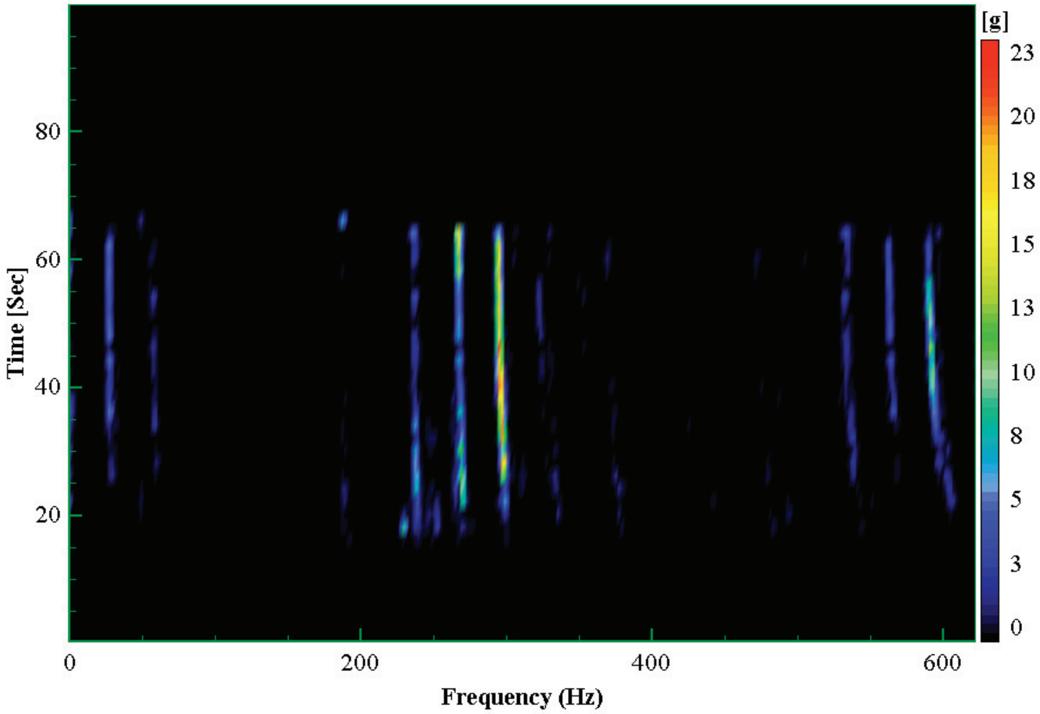


Fig. 8. Watefall of the accelerometer place in the middle of the closure section

6. Conclusions

In this paper an industrial application consisting in a gravity damper that must process large air masses was analysed. The study consisted in the vortex shedding phenomenon identification with numerical approach, supported by a wind tunnel campain; substantial agreement with experiments was found in terms of vortex shedding numerical frequencies. In particular it was noted that the presence and orientation of bolt and nuts do not affect the vortex shedding vibration in a significant way. The study was found to be interesting from a scientific point of view since it represent a simple device which is actually in use in many industrial applications and that, in operating conditions that can actually occur, can experience the lock-in. The paper puts in evidence that, in the case of lock-in at high Reynold's number, the amplitude of the flattering accelerations reach worrying levels that may affect the structural integrity.

The possibility to switch in lock-in was actually averted creating a gap between the flattering frequencies and vortex shedding one. Two different solutions were experimented in the wind tunnel; in the first case an airfoil profile was subsituted to the actual closure section and in the second case the cross section of the whole device was maintained but the longitudinal dimension of the closure section was halved. In both cases, a 90% reduction of the vertical acceleration was observed according to the same measurement conditions.

References

[1] O. Griffin, R. Skop, G. Koopmann, The vortex-excited resonant vibrations of circular cylinders, Journal of Sound and Vibration 31 (2) (1973) 235–IN3.

[2] C. Barbi, D. Favier, C. Maresca, D. Telionis, Vortex shedding and lock-on of a circular cylinder in oscillatory flow, Journal of Fluid Mechanics 170 (1986) 527–544.

[3] O. M. Griffin, A universal strouhal number for the locking-onof vortex shedding to the vibrations of bluff cylinders, J. Fluid Mech 85 (part 3) (1978) 591–606.

[4] A. Roshko, Experiments on the flow past a circular cylinder at very high reynolds number, Journal of Fluid Mechanics 10 (03) (1961) 345–356.

[5] W. Arnold, H. Hinrichs, K. Hinsch, Particle image velocimetry (piv) in the study of vortex shedding mechanisms from finite cylinders, in: Flow Visualization V; Proceedings of the 5th International Symposium, Vol. 1, 1990, pp. 414–421.

[6] L. Lourenco, L. Vandommelen, C. Shib, A. Krothapalli, Unsteady flow past an airfoil pitched at constant rate.

[7] J. Boris, O. Griffin, J. Baum, J. Gardner, F. Grinstein, K. Kailasanath, E. Oran, T. Young, Vortex shedding and base pressure behind a wide-splitter plate, Pressure Vessels and Piping 154, Flow-Induced Vibration (1989) 147–154.

[8] S. Murakami, A. Mochida, On turbulent vortex shedding flow past 2d square cylinder predicted by cfd, Journal of Wind Engineering and Industrial Aerodynamics 54 (1995) 191–211.

[9] P. Catalano, M. Wang, G. Iaccarino, P. Moin, Numerical simulation of the flow around a circular cylinder at high reynolds numbers, International Journal of Heat and Fluid Flow 24 (4) (2003) 463–469.

[10] D. Bouris, G. Bergeles, 2d les of vortex shedding from a square cylinder, Journal of Wind Engineering and Industrial Aerodynamics 80 (1) (1999) 31–46.

[11] D. Rocchi, A. Zasso, Vortex shedding from a circular cylinder in a smooth and wired configuration: comparison between 3d les simulation and experimental analysis, Journal of Wind Engineering and Industrial Aerodynamics 90 (4) (2002) 475–489.

[12] Z. Liu, G. A. Kopp, A numerical study of geometric effects on vortex shedding from elongated bluff bodies, Journal of Wind Engineering and Industrial Aerodynamics 101 (2012) 1–11.

[13] A. Nakayama, H. Noda, Les simulation of flow around a bluff body fitted with a splitter plate, Journal of Wind Engineering and Industrial Aerodynamics 85 (1) (2000) 85–96.

[14] M. W. Sarwar, T. Ishihara, Numerical study on suppression of vortex-induced vibrations of box girder bridge section by aerodynamic countermeasures, Journal of Wind Engineering and Industrial Aerodynamics 98 (12) (2010) 701–711.

[15] R. Körlin, U. Starossek, Wind tunnel test of an active mass damper for bridge decks, Journal of wind engineering and industrial aerodynamics 95 (4) (2007) 267–277.

[16] F. Ricciardelli, Effects of the vibration regime on the spanwise correlation of the aerodynamic forces on a 5: 1 rectangular cylinder, Journal of Wind Engineering and Industrial Aerodynamics 98 (4) (2010) 215–225.

[17] K. Kwok, X. Qin, C. Fok, P. Hitchcock, Wind-induced pressures around a sectional twin-deck bridge model: Effects of gap-width on the aerodynamic forces and vortex shedding mechanisms, Journal of Wind Engineering and Industrial Aerodynamics 110 (2012) 50–61.

[18] S. C. Yen, C. W. Yang, Flow patterns and vortex shedding behavior behind a square cylinder, Journal of Wind Engineering and Industrial Aerodynamics 99 (8) (2011) 868–878.

[19] S. Bhattacharya, J. W. Gregory, Investigation of the cylinder wake under spanwise periodic forcing with a segmented plasma actuator, Physics of Fluids (1994-present) 27 (1) (2015) 014102.

[20] O. Flamand, F. De Oliveira, A. Stathopoulos-Vlamis, P. Papanikolas, Conditions for occurrence of vortex shedding on a large cable stayed bridge: Full scale data from monitoring system, Journal of Wind Engineering and Industrial Aerodynamics 135 (2014) 163–169.

[21] H. Li, S. Laima, J. Ou, X. Zhao, W. Zhou, Y. Yu, N. Li, Z. Liu, Investigation of vortex-induced vibration of a suspension bridge with two separated steel box girders based on field measurements, Engineering Structures 33 (6) (2011) 1894–1907.

[22] M. Molinari, M. Pozzi, D. Zonta, L. Battisti, In-field testing of a steel wind turbine tower, in: Structural Dynamics and Renewable Energy, Volume 1, Springer, 2011, pp. 103–112.

[23] H. Mukundan, Y. Modarres-Sadeghi, J. M. Dahl, F. S. Hover, M. S. Triantafyllou, Monitoring viv fatigue damage on marine risers, Journal of Fluids and structures 25 (4) (2009) 617–628.

[24] H. Marcollo, J. Hinwood, On shear flow single mode lock-in with both cross-flow and in-line lock-in mechanisms, Journal of Fluids and structures 22 (2) (2006) 197–211.

[25] E. Garnier, N. Adams, P. Sagaut, Large eddy simulation for compressible flows, Springer Science & Business Media, 2009.

[26] J. Kleissl, Field experimental study of the Smagorinsky model and application to large eddy simulation, 2004.

[27] T. Kobayashi, Large eddy simulation for engineering applications, Fluid dynamics research 38 (2) (2006) 84–107.