Investigation on the relation between incoming wind characteristics and surface pressure distribution

Daniele Rocchi [a], Paolo Schito [a*], Alberto Zasso [a]

[a]*Department of Mechanical Engineering– Politecnico di Milano, via La Masa 1, 20156 Milano (MI) Italy*

ABSTRACT

A comparison between Large Eddy Simulation (LES) and Wind Tunnel tests (WT) results is proposed to investigate the relationship between the turbulent characteristics of the incoming wind and the fluctuations of the pressures on a building façade. CFD simulations are performed to compare an atmospheric boundary layer flow condition with an uniform turbulence condition in order to highlight the effects induced by the low-frequency large turbulent scales on the genesis of the peak pressure values. Simulations consider the wind-structure interaction of a medium rise building and are performed in the same geometrical scale of the WT tests allowing for a direct numerical-experimental comparison. By the comparison of the time histories of the surface pressure and by the statistical analysis of the distribution of the pressure fluctuations around the mean a physical correlation of the peak values occurrences with the lack or presence of turbulent structures is proposed.

Keywords: LES; wind tunnel test; medium rise building; peak pressure; PDE

1 INTRODUCTION

Wind loads on buildings are an important issue for structural engineers to ensure the safety of the inhabitants and of the people in the surrounding. In case of strong winds, the structure of the building must be able to survive to the pressure loads on the façades without detaching elements in the air. The mechanism that leads to the structural failure depends on the local pressure, on its statistical distribution and on the peak maximum absolute value. A lot of work has been conducted on simple geometries to predict the design pressure values, but the problem is still open on buildings with a non-conventional shape (Ono et al. 2008). This paper aims to investigate the relation between the incoming wind properties and the statistics of the surface pressure distribution on several points on a non-conventional 50m high medium rise building. To this purpose wind tunnel test results will be compared to CFD simulations results focusing the attention on the time histories of the surface pressure on different interesting points and on the statistical distribution of peak values. Wind tunnel tests were performed reproducing realistic wind condition with an atmospheric boundary layer profile and turbulence properties (turbulence intensity, length scales and correlation) scaled in the model dimension through passive turbulence generators and surface roughness (Lund et al. 1998, Kataoka et al. 2002, Kondo et al. 2002, Xie et Castro 2008, Thomas et Williams 1999, Lignarolo et al. 2011). The pressure measurements in 129 positions allow to highlight how the interaction between the wind and the building may lead to extremely different pressure values on different points on the building, not only in terms of mean values but mainly in terms of peak values and statistical distribution. While mean values are basically related to the mean wind profile

* Corresponding author: phone +39 02 2399 8434, fax +39 02 2399 8492, mail paolo.schito@mail.polimi.it

and to the presence or absence of flow separation from the building surface, the peak values are related to the turbulence that is present in the incoming wind and to its modification when the flow impacts on the building. Data post-processing of the wind tunnel (WT) tests results shows how the probabilistic distribution of the fluctuating pressure values around the mean value may be very different from point to point and in particular from region where positive peaks or negative peaks occur (Zasso et al. 2009). This experimental evidence is not explainable considering only the mean flow field modification, due to wind-structure interaction, but it may be related to the interaction of the larger turbulence structures, present in the atmospheric boundary layer flow, with those parts of the building that are effective in modifying and stretching the vortexes (edges driving flow separations). This phenomenon is present in the experimental results, since the wind tunnel common practice is to scale the real wind and structure interaction by reproducing a wind field that contains the turbulence structures and by simulating similar statistical properties of the real wind in the site where the structure will be built. The same approach is not usually followed by those CFD numerical simulations that model the incoming turbulence through URANS (Unsteady Reynolds Average Navier Stokes) approaches able to account only for averaged turbulence properties (k, ε, ω) or in those simulations where not realistic uniform turbulence conditions are used as boundary conditions for the incoming wind. In this paper a LES approach will be used to study how the fluctuating part of the surface pressure distribution is reproduced and in particular to investigate its relation to different incoming wind properties and turbulence content. As a first step two different incoming wind fields were generated through preliminary CFD simulations: an atmospheric boundary layer (ABL) scenario and an uniform turbulence (UT) one. In the former case, the preliminary simulation is performed to develop a flow field that matches the WT one in terms of mean profile, turbulence intensity and length scale profiles by reproducing the flow interaction with passive turbulence generators and floor roughness elements. The aim of the preliminary simulation is to develop large and small turbulence scales in a similar way of what is performed in the wind tunnel. In a similar manner, for the second situation the preliminary simulation was performed by simulating the flow interaction with a uniform solid grid in order to develop a turbulent incoming wind spectrum with a lack of the larger turbulence scales. The aim of these two preliminary simulations is to generate two databases of a time-space field of velocity and pressure to be used as time dependent incoming flow conditions that satisfy the Navier-Stokes equations and that will be assigned to the following numerical simulation of the wind-structure interaction. Simulations of the flow around the building are performed by using the same geometrical scale of the wind tunnel tests allowing for a direct comparison between wind tunnel data simulation results. The building represents a medium-rise building whose aerodynamic peculiarity and wind tunnel results were already presented in a previous paper (Zasso et al. 2009), highlighting the already mentioned features of the pressure distribution. ABL flow characteristics reproduced by CFD simulations are similar to those adopted during wind tunnel tests while UT conditions are investigated only by CFD simulations and they don’t have correspondence with equivalent WT tests. In the paper, an analysis on how the peak values of the fluctuating surface pressure are produced by the two different incoming wind scenarios will be presented and compared to the WT results with the aim to have a deeper insight on the flow mechanism that leads to have, in some parts of the structure, large negative peak pressure values whose probability distribution shows large asymmetric skewness. To the authors knowledge also the most recent works dealing with numerical-experimental investigation of the pressure distribution on buildings are used to compare results in terms of mean values and rms values (Lu et al. 2011). In the present paper the attention is mainly focused on the peak values representing the most critical parameter for the building designer and, in particular, to their distribution around the mean value that is not always predictable using the standard deviation and a gust-factor, especially when a large skewness of the PDE of the pressure variation is present. To this purpose results will be presented analyzing the time histories, the power spectral densities and the probability density estimate of the pressure fluctuations (Lim et al. 2009).

2 INCOMING WIND GENERATION

As already mentioned, 2 databases of wind fields were created, by performing preliminary CFD simulations, to be used as incoming flow condition for the numerical simulation of the wind-building interaction. An ABL turbulent flow field and an UT flow field were reproduced. To this purpose a numerical domain, corresponding to an empty wind tunnel test chamber is considered. The databases consist in the collection of time histories of the wind velocity components and pressure in a section of the domain, positioned downstream the passive turbulence generators (spires and floor roughness for the ABL and uniform grid for the UT), and close to the building model location during the wind tunnel tests.
The CFD simulations were performed using a Large Eddy Simulations (LES) approach with a dynamic Smagorinsky-Lilly Subgrid Scale model. Concerning the ABL flow simulation, the domain accurately reproduces the geometry of the wind tunnel test section equipped with the passive turbulence generators and floor roughness elements, in order to develop a boundary layer flow similar to what was present in the wind tunnel, with similar time and length scales.
Figure 1 reports a comparison among the wind tunnel test section equipped with passive turbulence generators and floor roughness elements adopted during the experimental campaign (a) and the CFD domain used for the generation of the ABL turbulent flow (b). The domain dimensions are: 45m in the longitudinal dimension, 14m in the lateral direction and it has a height of 3.8m (corresponding to the dimensions of the Boundary Layer test section of the Polimi Wind Tunnel (CIRIVE)). The floor roughness is geometrically reproduced in the CFD domain helping to slow down the lower part of the wind profile, in order to have high turbulence near the ground with the correct length scales. Uniform turbulence conditions are reproduced by simulating only the presence of a bar screen close to the inlet section of the numerical domain with a uniform square mesh with thickness of 0.2m and edges of 1m (Figure 1c). Constant total pressure boundary conditions are imposed on the inlet, constant static pressure is assigned to the outlet of the domain to avoid unwanted pressure fluctuations as discussed in a previous works (Zasso et al. 2009, Zasso et al. 2010). No-slip conditions are considered on all the other surfaces of the domain, i.e the spires at the beginning of the test chamber or the square mesh grid, the floor roughness on the bottom surface and all the vertical and horizontal surfaces. The mesh is mainly structured, the total number of cells, for the ABL simulation, is 11’650’000 (most of them are used to describe the floor roughness). The maximum element of the grid has a length of 0.1m. A similar discretization has been used also for the UT simulation.

(a) (b) (c)

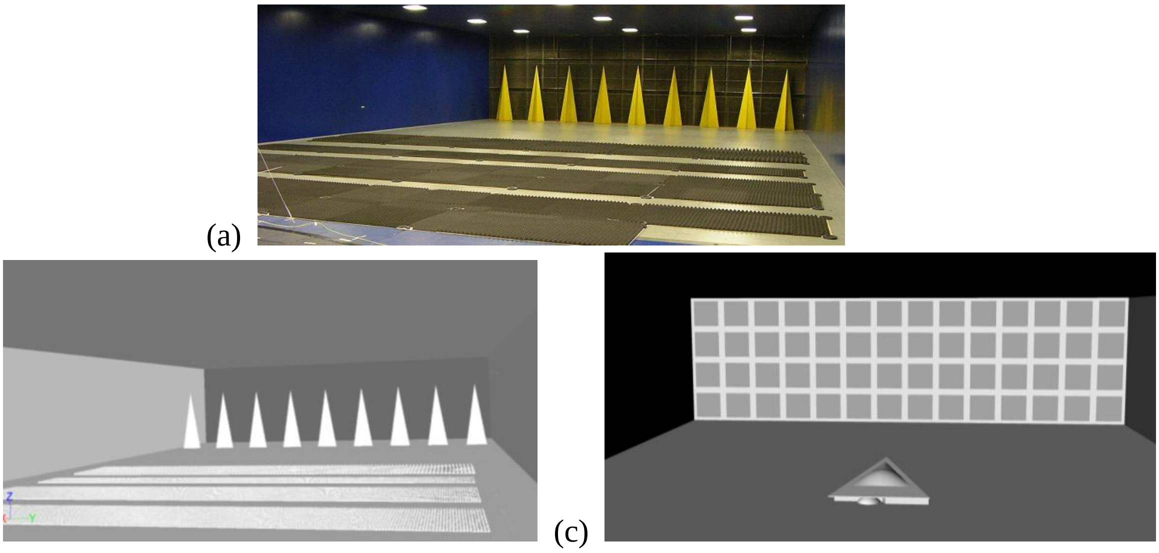


Figure 1. Wind Tunnel test section (a). CFD domains for the generation of the turbulent boundary layer flow (b) and of the uniform turbulence (c).

Simulations are performed for 200 s with a time step of 0.01 s using a second order numerical scheme according to what will be used for the simulations on the building. The pressure and velocity distribution on all the cells at a section 5 m upwind the model position where the building model were positioned during the wind tunnel tests is recorded at each time step and will be later used for the wind-building interaction simulation. Figure 2 shows an analysis of the flow produced by the CFD simulations with ABL reproduction (“LES profile”) and with uniform turbulence (“LES uniform”) compared to the wind tunnel flow (“Exp”) in terms of vertical non dimensional profile of mean wind speed, turbulence intensity of the three wind velocity components and corresponding integral length scales along the middle vertical line of the considered section. “LES profiles” and “Exp” results are in good agreement in terms of mean wind speed profiles, while “LES uniform” flow has a higher wind speed close to the floor since it doesn’t reproduce the slowing down effect induced by the floor roughness that is present in the other two cases. CFD mean wind speed profiles are reported in non-dimensional form using the velocity at the reference height of 0.4 m, representing the reference height of the building. Uniform turbulence simulation has an almost block profile at higher levels. On the contrary the ABL simulation is able to reproduce the vertical gradient of the experimental mean wind velocity, denoting to well simulate the contemporary effect of both the large mixing due to the spires at higher quotes and the small scale turbulence generated by the floor roughness in the lower part of the flow. In particular not only the mean value is well captured, but also the vertical profiles of turbulence intensity of the different wind velocity components are similar between the experimental and the simulated ABL conditions, especially close to the reference height (0.4 m): I_u=0.13; I_v=0.11; I_w=0.09.

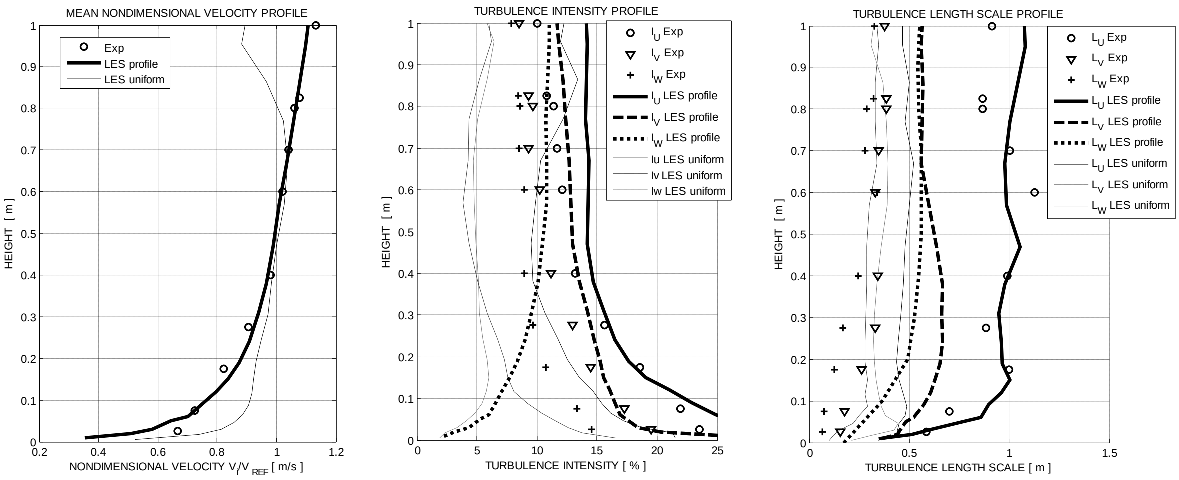


Figure 2. Mean flow speed (left), turbulence intensity (center) and length scale (right) vertical profiles.

The uniform turbulence simulation is characterized by lower levels of turbulence intensity especially in the transversal and vertical components (almost half of the previous ones). It is possible to observe how the ratios between the turbulence intensity values of the different wind velocity components, are similar comparing the experiments with ABL case. In this case the lateral component is almost in the middle between the other two values, while they are different for the uniform turbulence flow where I_v and I_w are similar (and are almost half the value of Iu): this corresponds to a uniform distribution of the energy of the turbulence fluctuations in the lateral and the vertical direction. A comparison of the values of turbulence intensity at the reference height is reported in Table 1. In terms of turbulence length scales, it is possible to appreciate how the ABL numerical flow is characterized, on the along wind direction, by values that are similar to the experimental ones while they are almost double in the other directions, at the reference height.

Table 1 - Reference turbulence intensities at the reference height of 0.4m.

	I_U	I_V	I_W
EXP	13 %	11 %	9 %
LES profile	15 %	12 %	10 %
LES uniform	9 %	5 %	5 %

Wind tunnel values, in terms of ratio between the length scales of the different wind velocity components, are more similar to what prescribed in standards with respect to natural winds (Eurocode: ${}^xL_v=0.25{}^xL_u$ and ${}^xL_w=0.1{}^xL_u$) with a stretching of the values in favor of the along wind component that is much larger than the others. CFD values denote that the turbulence structures are similarly stretched along the wind direction but are less anisotropic: the ratio between the three components tends to be constant with the height in the UT case while in the ABL case lateral and vertical components tend to have similar values for a height larger than 0,6 m as it happens in wind tunnel. ${}^xL_u$ integral length scales of the uniform turbulence simulation are, as expected, sensibly lower than the ABL ones (almost the half) while ${}^xL_v$ and ${}^xL_w$ values are closer the ${}^xL_u$ value presenting a less anisotropic structure of the flow.

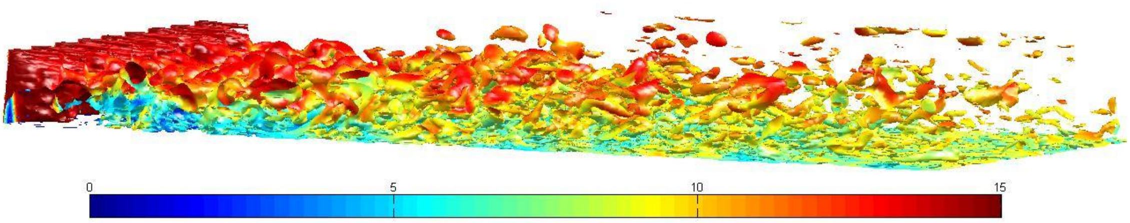


Figure 3. Isosurfaces of vorticity in the ABL CFD flow colored by velocity magnitude

A representation of the flow field for the ABL numerical simulation is reported in Figure 3 where the interaction between the incoming wind and the spires is depicted through a contour plot of the velocity magnitude on isosurfaces of equal vorticity. It is possible to observe how large turbulent scales are generated and convected in the flow domain being stretched in the flow direction and strongly mixed in the bottom part.

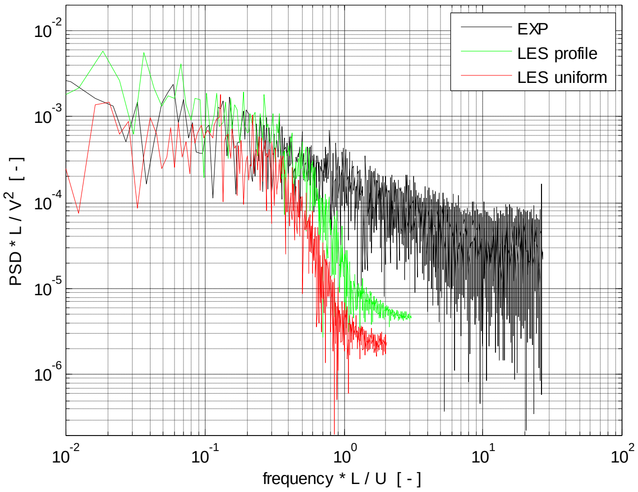


Figure 4. Power Spectrum Density of longitudinal velocity ad 0.5 m height.

A further comparison is reported in Figure 4 in terms of power spectral density distribution of the longitudinal wind velocity component at 0.4 m height on the middle vertical line of the considered section. PSD values are presented in non-dimensional form by simply dividing them by the square of the mean wind speed at the same location and multiplying them by the reference height of the building model (L=0.5 m) as far as the frequency that is divided by the mean wind speed and multiplied by L in order to compare the different situations characterized by different mean wind speeds.

In this way, considering the experimental data as reference, it is possible to appreciate that:

- at high frequency, the numerical spectra are limited by the LES numerical mesh filtering;
- at middle-low frequency, the energy content of the ABL numerical simulations is very similar to the experimental one both considering the absolute value and the distribution on the different harmonic turbulent components;
- at low frequencies, the uniform turbulence results highlight the lack of low frequency (large dimensional scale) components;
- the energy content of the uniform turbulence flow is lower than what is present in the other situations

3 WIND LOADS ON BUILDING

Simulations of the wind-building interaction are performed on a numerical domain that reproduces the part of wind tunnel chamber where the building model was positioned during the wind tunnel tests. The inlet section of the domain corresponds to the section (5 m upwind the model) where the flow data were obtained by the previous simulations. The numerical grid has 13’700’000 cells and it is mainly structured, apart from the region around the building, where tetrahedral cells are used to improve the mesh refinement close to the walls. The maximum cell size is 0.05m. Two unsteady simulations have been conducted on the same domain, using the same mesh discretization and simulation set-up: one using as inlet boundary condition the turbulent boundary layer wind profile obtained from the ABL simulation and the other with the uniform turbulence wind profile generated with the UT simulation. The inlet boundary condition is imposed by using the time-space field of velocity and pressure calculated in the previous simulations, while all other boundary conditions and simulations settings are the same as those reported for the flow simulations. The same Sub-Grid Scale model and discretization order of the preliminary turbulence generation calculation is kept.
The building geometry corresponds to a medium rise building called Cultural Center located in Abuja (Nigeria), that was already studied through wind tunnel tests by the same authors [Zasso et al, 2009, Zasso et al. 2010] measuring the surface pressure distribution on 129 points at a sample frequency of 100 Hz for 200 s. The details of the wind tunnel results are presented in the reported works and they will be not here reported.
Results between numerical simulations and wind tunnel tests results will be presented in terms of pressure coefficients obtained by dividing the differential pressure between the local value and the reference pressure value by the dynamic pressure q defined on the basis of the reference wind speed according to the following formulation:

$$c_p = \frac{p - p_{ref}}{0.5 \rho U_{ref}^2} = \frac{p - p_{ref}}{q} \quad (1)$$

In the experiments the reference pressure value is the average pressure measured on the wind tunnel test section during the tests, while the reference pressure for the numerical simulations is close to the pressure that is been imposed to the outlet boundary condition. The reference wind speed is the wind velocity magnitude measured/simulated at 0.4 m height 5 m upwind the model. In the following, the time fluctuation of the pressure coefficient on the different faces of the building will be analyzed.

A first comparison, between experimental and numerical results, is reported in Figure 5 in terms of time histories of the measured/simulated pressure coefficient on a point located on the building roof close to the upper windward edge. The position of the considered point is highlighted with a white circle on a rendering of the building that is reported in the same figure.

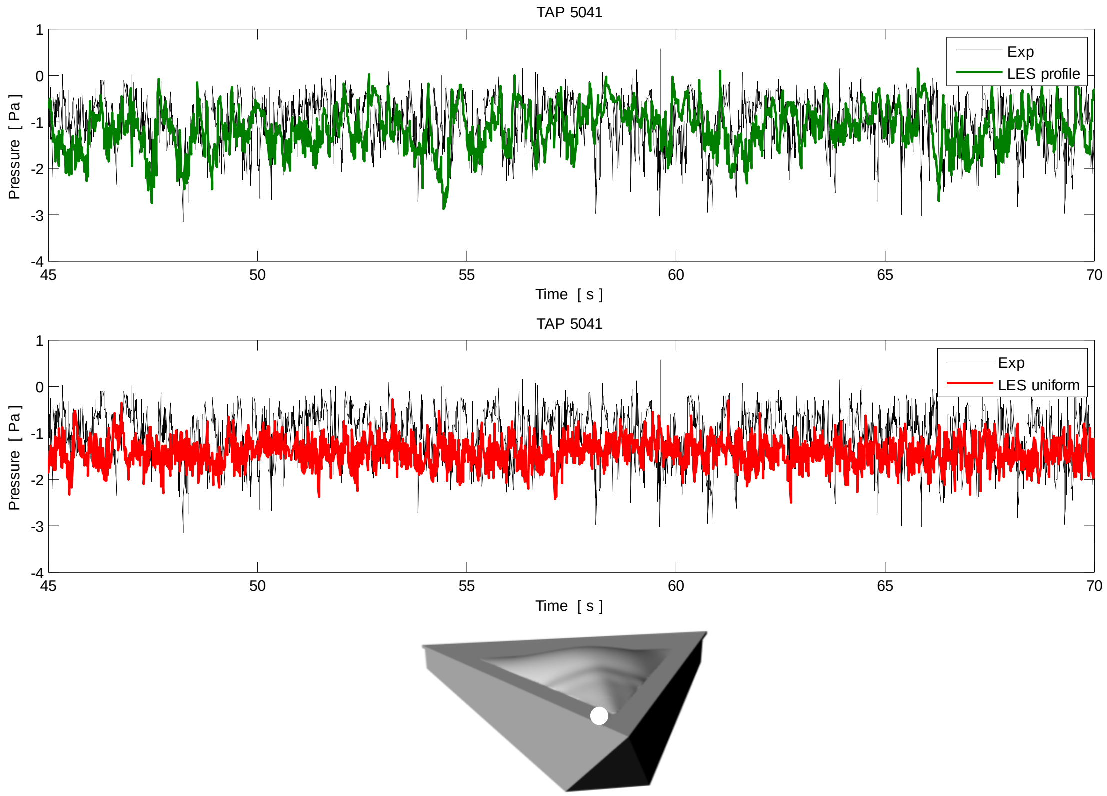


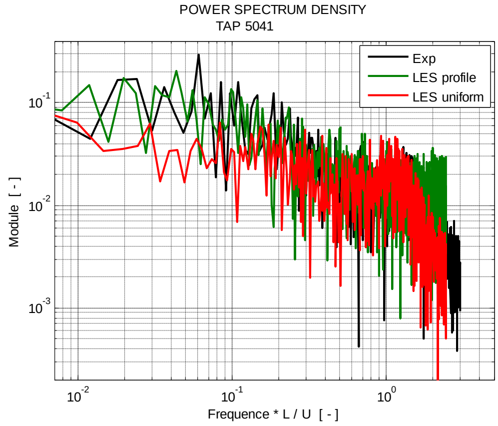
Figure 5. Pressure coefficients time histories numerical vs experimental data for the considered point.

The LES simulation with uniform turbulence shows smaller fluctuations of the pressure coefficient compared to the experimental ones while a better agreement is present if we consider the LES simulation with ABL and the experimental data.
Table 2 reports a comparison between the mean, the maximum and the minimum values of pressure coefficients time histories.
As already visible in the time histories, the mean value of the LES uniform turbulence simulation is higher in magnitude (larger negative value) than the others but, nevertheless, the maximum negative peak values are reached by the experimental results and by the numerical results in presence of ABL flow characteristics.

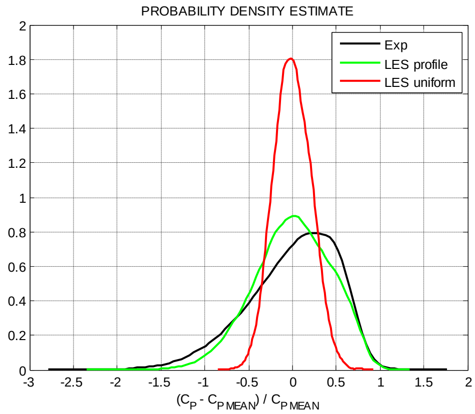
Table 2 – Pressure coefficients statistical values: experimental wind tunnel tests vs CFD simulations results

	$C_{P\ MEAN}$	$C_{P\ MAX}$	$C_{P\ MIN}$	Std deviation	skewness
Exp	-1.07	0.56	-3.81	0.55	-0.73
LES profile	-1.16	0.45	-3.70	0.53	-0.20
LES uniform	-1.45	-0.30	-2.51	0.31	0.08

The explanation of such a difference is not only related to the lower turbulence intensity level, that is present in the uniform turbulence flow. In fact, the pressure fluctuations on the considered position is not only related to the buffeting effect of the turbulence contained in the incoming flow, but they are also related to the way the flow interacts with the roof edges. The point under consideration is located downstream a flow separation that is induced by the trailing edge of the roof and therefore the pressure fluctuations acting on that point depend on what happens in the separation region when the different flow turbulent structures interact with the building geometry. In order to better analyze this aspect, the power spectral densities of all the signals are overlapped in Figure 6 and the probability density estimates (PDEs) of the ratio between the variations of the pressure coefficient and the mean value are reported.



(a)



(b)

Figure 6. C_p power spectral density (a) and PDEs of $(C_P-C_{P\ MEAN})/C_{P\ MEAN}$ (b): numerical vs experimental

From the analysis of the power spectral densities, it is evident that there is less energy at low frequencies, in the pressure signal of the uniform turbulence simulation, while the frequency content between "Exp" and "LES profile" results is very similar. Anyway the energy content at high frequency is very similar among all the results.

This is related to the incoming wind characteristics (see Figure 4) and it is due to the wind-building interaction in presence or absence of large turbulent structures (low frequency components). As expected, the lack of low frequency components, in the uniform turbulence data, prevents the pressure coefficient to reach the large negative values that sometimes occur in the experimental time history, visible in Figure 5 and particularly interesting, from an engineering point of view, since they drive the design aspects. What is really interesting in the comparison between the two situations is highlighted in the comparison of the PDEs. The uniform turbulence simulation result exhibits an almost Gaussian distribution of the values that are limited to be lower than the 80% of the mean value while both the experimental result and the ABL simulation result are characterized by a wider and not symmetric distribution. It is possible to observe that the experimental results show a higher probability to have large fluctuations of the negative Cp values (from 0,7 to 2 times the mean value) compared to the CFD profile results, while the large fluctuations of the positive Cp values (between 0,75 to 1,5 of the mean value) have the same probability distribution.

The comparison among the PDEs shows how the effect on the building façade pressure of the small turbulent scales, acting at high frequency, is different from that related to the large turbulent structures acting at low frequency. In fact, the UT simulation that is characterized by an almost equal amount of energy distributed on all the present turbulence scales (see the flat trend of the power spectrum reported in Figure 6) and a limited amount of anisotropy, produces pressure fluctuations that are far from the real physics of the phenomenon measured in the wind tunnel, since they are small and perfectly symmetrically distributed. Probably, negative peak values, of the same order of magnitude of the experimental one, might be reached by simply increasing the incoming uniform turbulence intensity level but they would be anyway symmetrically distributed around the mean.

On the other hand, the LES simulation with ABL flow characteristics appears to better catch the fluid-structure interaction phenomena, suggesting that what was observed in the experimental data (large negative peak values on some points on the building facade) is related to the low frequency turbulence scales.

The different behavior between the LES uniform results and the other ones is also highlighted by the skewness parameter that is largely negative for the experimental results and for the LES profile results while it is almost null for in the LES uniform case as reported in Table 2.

The skewness parameter represents an index of the probability to have a not symmetric distribution of the extreme values around the mean and it helps to highlight critical situations like the one considered in this example where the largest negative peaks are reached with pressure fluctuations that may be amplified up to 2 times the mean values, while positive fluctuations can reach values up to 1.5 times the mean value.
As an additional insight in the fluid-structure interaction phenomenon, Figure 7 reports an instant view of the flow field obtained by plotting the colored contour plot of velocity magnitude on an iso-surface of equal vorticity, when a negative peak pressure occurs. It is possible to highlight the presence of a large turbulence structure in the flow in correspondence of the considered point, positioned on the building roof. In the image it is possible to appreciate how this situation may be related to the interaction between the building and a large turbulent coherent structure impinging on the upper part of the building close to the considered point.
It is worth remembering that the region, the considered point belongs to, is a region where a strong mean C_P gradient occurs because of the speed up effect of the separation phenomenon. The speed up effect becomes an amplification of the suction in the region close to the building edge, since pressure are related to the square of the wind speed. When a large turbulent structure approaches the leading edge of the roof, a large slowing varying fluctuation of the wind speed occurs, that is locally perceived by the local pressure. The large fluctuation is therefore amplified by the wind speed-up in the same direction of what happens for the mean pressure field and it results in a large negative fluctuation of the C_P.

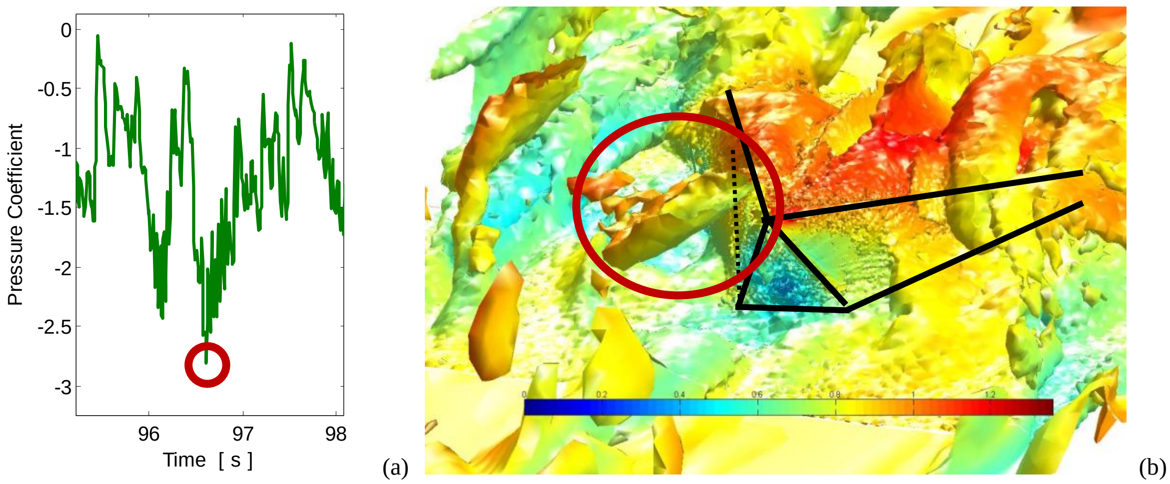


Figure 7. Peak of negative pressure coefficient in the time history corresponding to the flow situation reported in (b) as a contour plot of velocity magnitude plotted on isosurfaces of vorticity.

The residual discrepancy between the numerical LES profile and the experimental values, in the author opinion, may be related, among all the other possible causes, to the presence of strong gradients in the separation region, close to the roof edge, that may be affected by the quality of the experimental realization of the geometry adopted during the wind tunnel tests. Nevertheless, the considerations on the numerical capability to reproduce the fluid dynamic phenomena are anyway valid.

Table 3 – Pressure coefficients statistical values: experimental wind tunnel tests vs. CFD simulations results for the pressure tap reported in Figure 8.

	$C_{P\,MEAN}$	$C_{P\,MAX}$	$C_{P\,MIN}$	Std deviation	skewness
Exp	0.71	1.56	0.25	0.17	0.40
LES profile	0.89	1.77	0.21	0.23	0.43
LES uniform	0.95	1.42	0.46	0.15	0.13

A similar investigation on pressure fluctuations is reported in the following by considering other interesting points located on other faces of the building and characterized by different types of flow-structure interaction already presented in a previous work by Zasso et al. (2009). Table 3 reports the main statistics for the C_P fluctuation on a point located on the frontal vertical face that is shown in Figure 8. It is possible to see that the positive mean value of the pressure coefficient shows small differences that are mainly related to the differences in the incoming wind profile measured/simulated upstream of the building.

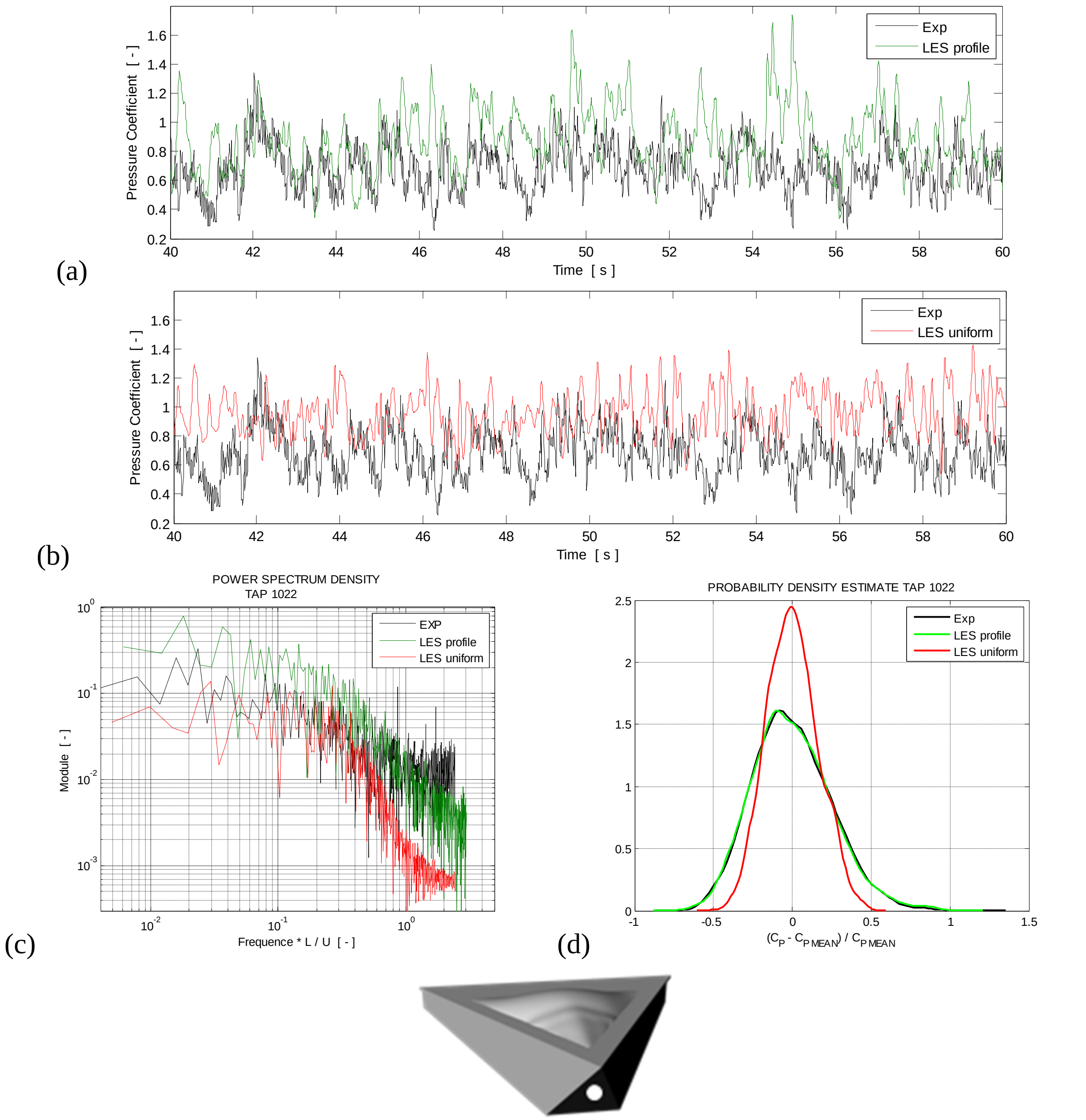


Figure 8. C_p Time history (a-b), power spectral density (c) and probability distribution of $(C_P-C_{P\,MEAN})/C_{P\,MEAN}$ (d): numerical vs experimental

A very good agreement is shown if we consider the PDEs and the skewness values of LES profile results with the experimental ones, showing that the more likely fluctuation of the C_P value are on the positive tail of the distribution (from 0.5 to 1 times the mean value). The "LES uniform" results show, also in this case, a more gaussian distribution with a slightly positive skewness value.

A positive value of the skewness is expected, since the flow velocity has a gaussian distribution around the mean value and the static pressure on a wind-ward face is proportional to the square of the velocity. The standard deviation of "LES uniform" simulation, that is similar to the experimental one, if related to the higher wind speed in the UT wind profile, produces small pressure fluctuations as reported in Figure 8-b and therefore cannot be used as a meaningful index for the correct representation of the interaction phenomenon. Pressure fluctuations closer to the experimental ones are in fact visible in Figure 8-a, where LES profile simulation is considered, despite the fact that standard deviation is higher and imputable to the higher energy content at low frequency Figure 8-c.

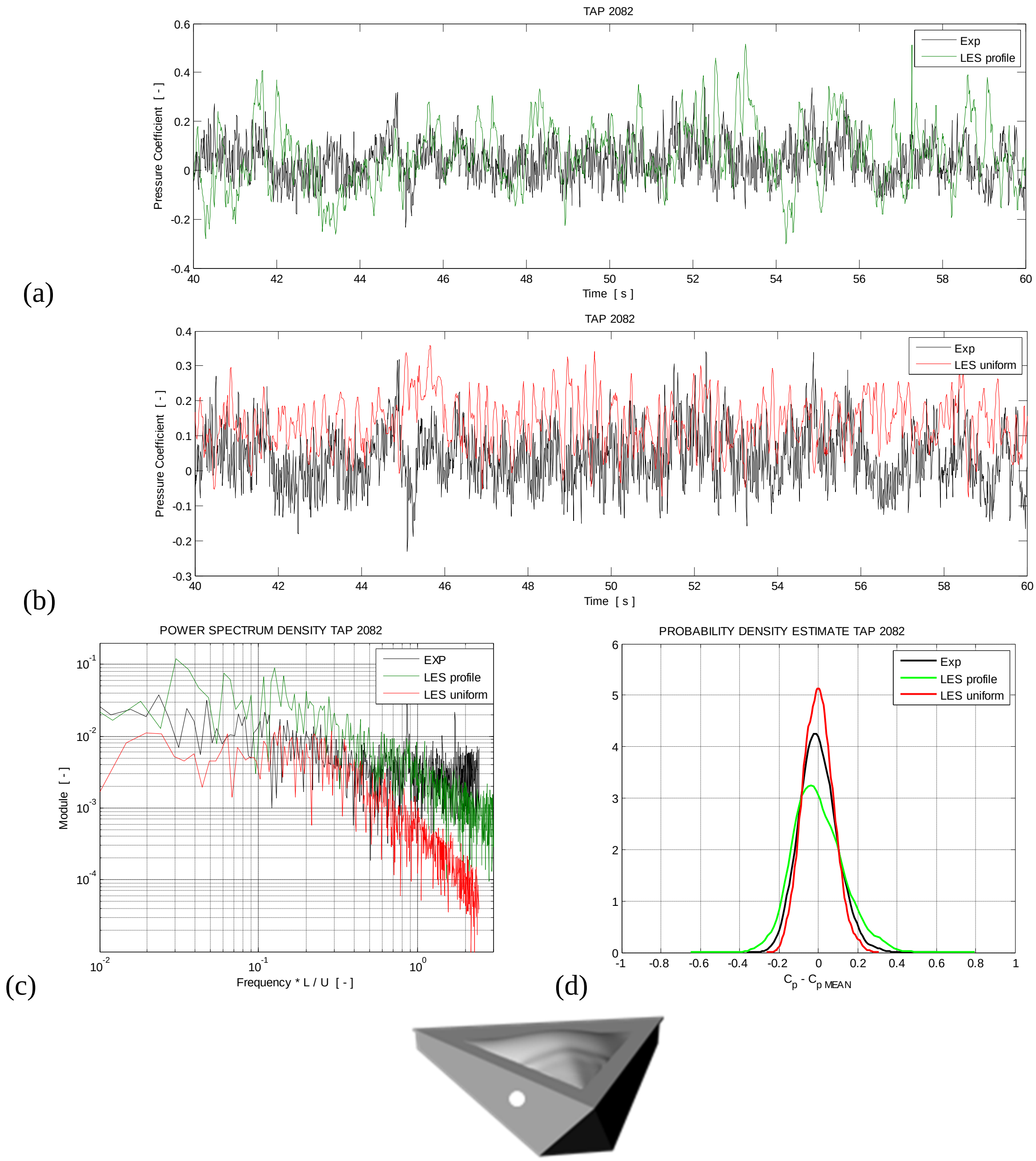


Figure 9. C_p Time history (a-b), power spectral density (c) and probability distribution of C_P-$C_{P\ MEAN}$ (d): numerical vs experimental

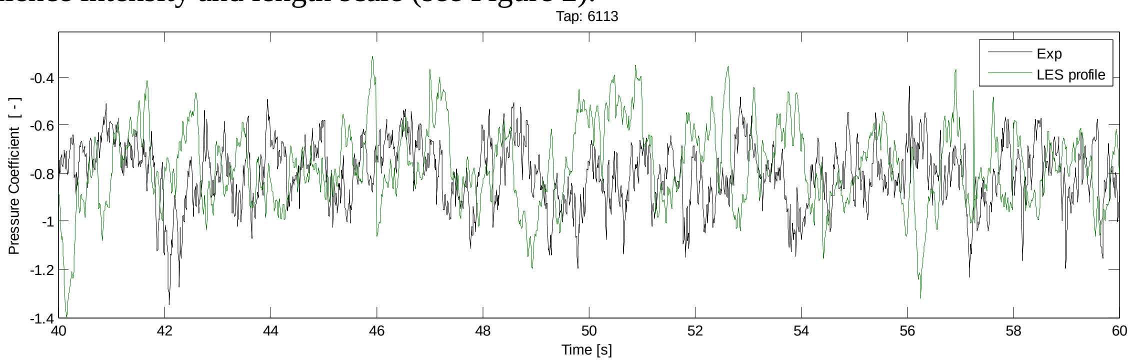
The statistical values of the pressure coefficient measured/simulated on the point positioned on a lateral vertical façade of the building, reported in Figure 9 are summarized in Table 4.

The point belongs to a face where the flow is attached and deflected. The mean pressure coefficient is almost null and it is captured with good accuracy by all the simulations.

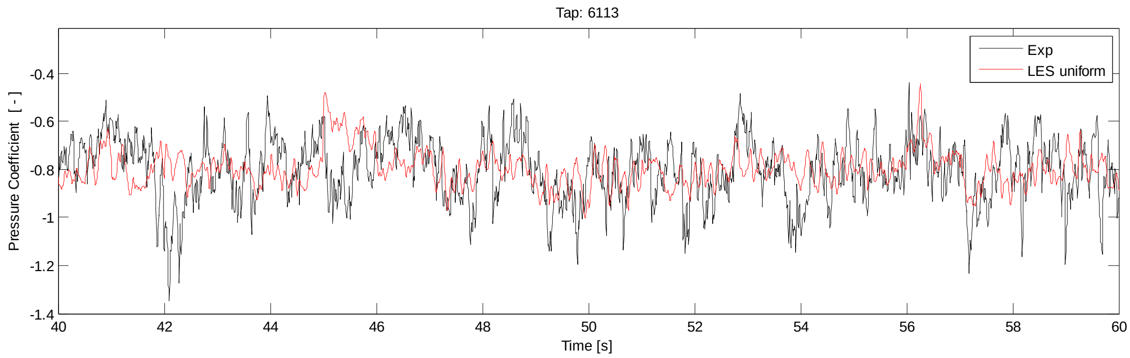
Table 4 – Pressure coefficients statistical values: experimental wind tunnel tests vs. CFD simulations results for the pressure tap reported in Figure 9.

	$C_{P\ MEAN}$	$C_{P\ MAX}$	$C_{P\ MIN}$	Std deviation	skewness
Exp	0.03	0.53	-0.26	0.09	0.35
LES profile	0.07	0.62	-0.52	0.13	0.49
LES uniform	0.13	0.39	-0.09	0.08	0.21

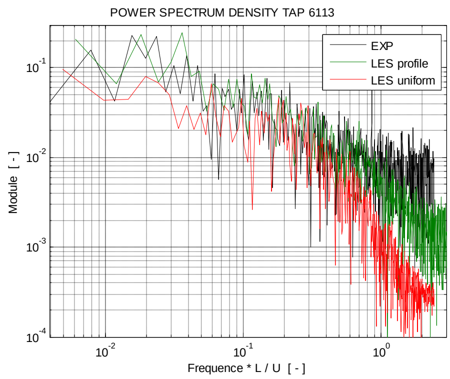
The differences, as already found for the previous pressure tap on the windward face, are in the energy content of the signal, where the result of the LES profile overestimates the fluctuations, having larger extreme events and higher standard deviation, while the LES uniform, has a standard deviation closer to the experimental values but it shows in the time history a different distribution of the pressure coefficient peaks. Considering the PDEs it is possible to appreciate how the "Exp" data are in the middle of the two conditions represented by the narrower distribution of the "LES uniform" simulation, that is lacking of the larger turbulent scale, and of the "LES profile" simulation that tends to overestimate the low frequency part of the PSD as can be seen in Figure 9-c. In this case, the PDEs are reported for the absolute value of the C_P fluctuations instead of its ratio against the mean value, since the mean value is close to zero. The overestimation of the contribution of the large turbulent scale leads also to overestimate the skewness parameter, being the positive peak events more frequent in the "LES profile" simulation, as evident by analyzing the time history of Figure 9-a. A possible explanation of the discrepancy between the "Exp" and "LES profile" results may be searched in the particular face orientation that makes it more sensitive on how the turbulence is distributed/stretched along the lateral direction and therefore more related to the reproduction of the correct flow characteristics that for the "LES profile" case seems to have higher levels of turbulence intensity and length scale (see Figure 2).

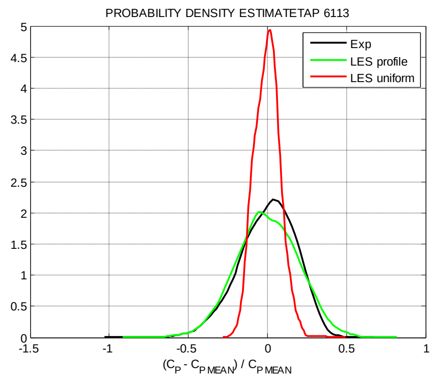


(a)



(b)

(c) 

(d) 

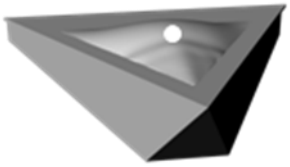


Figure 10. C_p Time history (a-b), power spectral density (c) and probability distribution of $(C_P-C_{P\ MEAN})/C_{P\ MEAN}$ (d): numerical vs experimental

The last consideration are performed for a pressure tap located on the top of the building dome, as can be seen in Figure 10 and summarized in Table 5. The flow in this region is attached to the roof that presents a varying curvature that drive the flow to speed up and the mean pressure to be largely negative as well reproduced by both the simulations. For this particular point, the “Exp” and the “LES profile” simulation are in good agreement, showing good correspondence also in the statistic values and in the PSD. For these two cases, the skewness value, denotes a different distribution of the probability of occurrence of positive pressure fluctuations. In fact, even if the negative part of the two PDE curves is in good agreement, the more likely occurrence of positive values in the “LES profile” simulation, moves the skewness value closer to zero. On the other side LES uniform simulations don't predict accurately the extreme events, as can be inferred also by standard deviation value, and moreover shows a positive skewness value.

Table 5 – Pressure coefficients statistical values: experimental wind tunnel tests vs. CFD simulations results for the pressure tap reported in Figure 10.

	$C_{P\ MEAN}$	$C_{P\ MAX}$	$C_{P\ MIN}$	Std deviation	skewness
Exp	-0.84	-0.43	-1.56	0.15	-0.43
LES profile	-0.77	-0.31	-1.40	0.15	-0.08
LES uniform	-0.80	-0.33	-1.00	0.08	0.30

4 CONCLUSIONS

A correct dimensioning of the structure has to consider the appropriate reproduction of the interaction between the atmospheric boundary layer turbulent flow and the structure, since peak values are not always related just to the mean pressure value and standard deviation.
This is more evident in region of separated flow where usually occurs the dimensioning large negative peaks of pressure that are not symmetrically distributed around the mean.
The skewness parameter helps to identify these situations of asymmetric distribution of the peak values, but it is not sufficient to evaluate the numerical simulation capability to repro-

duce the correct wind-structure interaction phenomenon. On the other hand, the standard deviation, that is usually adopted to compare experimental and numerical results, is a too synthetic statistical parameter to judge the quality of the simulations from a phenomenological point of view and could be also inappropriate for the estimation of peak values. To this purpose, in this paper, an overview of the time histories, of the frequency distribution and of the probability density estimate of the pressure fluctuations is proposed for different points on the building surface. The geometry of the considered medium rise building and the selection of the analyzed locations are chosen to compare different wind-structure interaction conditions.
LES results highlighted, through the comparison between the results obtained with a uniform turbulent wind and with an ABL turbulent wind profile, that the extreme values (located in the far tails of the probability distribution) are related to the turbulence structure of the flow.
In particular the presence of large low-frequency structures is required to reproduce a real-like probability distribution of pressure fluctuation and to simulate the most demanding peak values in region of separated flow.
It is therefore mandatory, both in wind tunnel modeling and in CFD simulations, to correctly reproduce the incoming wind turbulent characteristics in order to define the more appropriate dimensioning peak pressure values. The definition of the incoming wind characteristics becomes therefore a key feature for the modeling of the wind-structure phenomenon.

5 ACKNOWLEDGEMENTS

This work has been supported by Regione Lombardia and CILEA Consortium through a LISA (Laboratory for Interdisciplinary Advanced Simulation) 2010 grant [link:http://lisa.cilea.it]

6 REFERENCES

Zasso A., Rocchi D., Schito P., 2009 “Experimental and numerical study of the flow around a low rise building (CC)”, EACWE5, Firenze – Italy

Zasso A., Rocchi D., Schito P., 2010 “Evaluation of unsteady pressure fluctuations on surfaces using CFD”, CWE2010, Chapel Hill – North Carolina USA

Lund T., Wu X., Squires K., 1998. Generation of turbulent inflow data for spatially-developing boundary layer simulations. Journal of Computational Phisics 140, 233-258

Ono Y., Tamura T., Kataoka H., 2008. LES analysis of un steady characteristics of conical vortex on a flat roof. Journal of Wind Engineering and Industrial Aerodynamics 96, 2007-2018

Kataoka H., Mizuno M., 2002. Numerical flow computation around aeroelastic 3D square cylinder using inflow turbulence. Wind and Structures, 5: 379-392.

Kondo K., Tsuchiya M., Mochida A., Murakami S., 2002. Generation of inflow turbulent boundary layer for LES computation. Wind and Structures, 5: 209-226.

Lu C.L., Li Q.S., Huang S.H., Chen F.B., Fu X.Y., Large eddy simulation of wind effects on a long-span complex roof structure, Journal of Wind Engineering and Industrial Aerodynamics, Volume 100, Issue 1, January 2012, Pages 1-18.

Hee Chang Lim, T.G. Thomas, Ian P. Castro, Flow around a cube in a turbulent boundary layer: LES and experiment, Journal of Wind Engineering and Industrial Aerodynamics, Volume 97, Issue 2, February 2009, Pages 96-109

Xie, Z.T.,Castro,I.P., 2008. Efficient generation of inflow conditions for large-eddy simulations of street-scale flows. J. Flow Turb. Combus. 81,449–470

Thomas T.G., Williams J.J.R., Generating a wind environment for large eddy simulation of bluff body flows, Journal of Wind Engineering and Industrial Aerodynamics, Volume 82, Issues 1-3, August 1999, Pages 189-208

Lignarolo L., Gorlé C., Parente A., Benocci C. Large eddy simulation of the atmospheric boundary layer using OpenFOAM, Proc. ICWE13, Amsterdam, The Netherlands, July 2011

CIRIVE, www.windtunnel.polimi.it