Doctoral School on Engineering Sciences
Università Politecnica delle Marche

Extended summary

Microscale Analysis of wind resource: experimental approach and numerical modelling

Curriculum: Energy Sciences

Author

Daniele Di Benedetto

Tutor

Prof. Eng. Renato Ricci

February 15th, 2014

Abstract. The aim of this work is to evaluate the potential of the wind tunnel tests and the numerical modeling, which represent the alternative solutions to in-situ measurements of environmental flows. In this work the experimental approach was conducted inside the environmental wind tunnel of the *Università Politecnica delle Marche*; the test chamber is 9 m long and so there is not enough length to naturally recreate a fully developed environmental boundary layer at the end of the test section. In order to simulate the Atmospheric Boundary Layer (ABL), special devices as a fetch of roughness elements, Counihan elliptic wedge generators and a castellated barrier wall are used. The reliability of the ABL simulation has been verified with anemometer measurements, pressure taps and PIV investigation. The performance of several Reynolds Averaged Navier-Stokes (RANS) turbulence models have been also evaluated with several CFD codes on 2D and 3D environmental test cases. The obtained results highlights the extreme sensitivity on turbulence parameterization. Lastly a district of Ancona, characterized by a complex terrain topography, was chosen as a case study. Experimental measurements in the wind tunnel on the scale model and numerical simulations have shown the interaction of the wind both with the terrain topography and with the built environment. This interaction modifies substantially the flow field respect to the mesoscale flow. The comparison between numerical and experimental results shows the importance of the scale parameters.

Keywords. CFD, Microscale Analysis, Wind Resource, Wind Tunnel.

1 Introduction

The study of environmental flows[1], relative to environmental and energy issues, is currently a very important topic in engineering and in the coming years it will assume an increasing interest.

The environmental parameters could be evaluated directly in situ through the use of instrumented towers; but this direct method can be too expensive and in most cases difficult to achieve. For this reason, more and more frequently, the techniques to obtain this information involve the use of scale models in wind tunnel or the numerical modeling of the problem.

The environmental wind tunnel of the "*UNIVPM*" requires, in order to ensure a correct simulation of the wind distribution in the test chamber, the installation of particular devices to recreate the ABL, such as the Counihan spires [2][3][4].

In this work some numerical solutions are presented and the applicability of CFD codes, both commercial and open source, have been evaluated through typical environmental test cases characterized by the presence of complex fluid dynamic phenomena [5][6].

To evaluate the interaction between the wind resource and the urban environment, an area, characterized by a complex terrain topography and also a not negligible buildings, was chosen as a case study. The area is a zone in the district of Ancona destined to a neighborhood of new construction. The analysis was conducted both in wind tunnel on a scale model 1:500 and numerically.

The comparison with the experimental results allowed evaluating the problem of the Reynolds analogy, which is the main limit arising in the experimental tests on scale models and can hardly be reached.

2 Analysis and discussion of main results

2.1 Experimental analysis of the simulated ABL

The experimental approach was conducted inside the environmental wind tunnel of *UNIVPM;* the test chamber, that is 9 m long, does not allow the natural growth of the boundary layer and requires special devices to simulate the ABL.

2.1.1 Anemometric measurements

The anemometric measurements have allowed to evaluate the efficiency of a configuration with a fetch of roughness elements (RE), elliptic vortex generators or Counihan spires (EWG) and a castellated barrier wall (CB).

The system is composed specifically by three Counihan spires with an height of 994 mm and a base angle of 12°, a 6 m fetch of two different roughness elements, a first staggered disposition of 40 mm wooden cubes and a second staggered disposition of 20 mm wooden cubes.

In order to evaluate the uniformity of the flow in the central section of the test chamber, the anemometry measurements were obtained scanning the whole section. In the Figures 1 and 2 the dimensionless velocity and the turbulence intensity profiles are shown, the profiles are measured in y=0 and the bars are the maximum variance of the quantity in y-direction at different heights. For each velocity profile the main parameters which represent the simulated boundary are collected in Table 1.

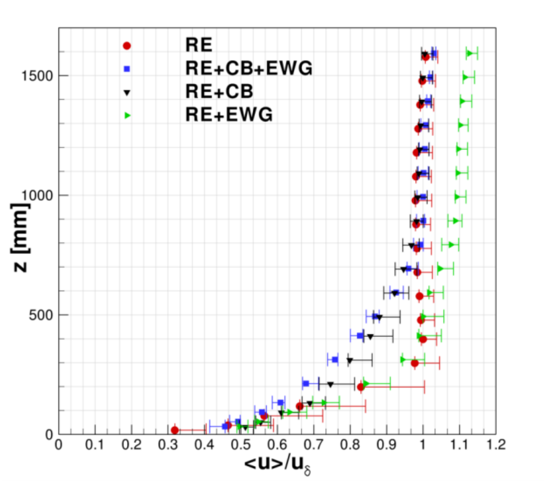

Figure 1. Dimensionless mean velocity profiles of different arrangements in y=0, the bars represent the maximum variances of the quantity in y-direction.

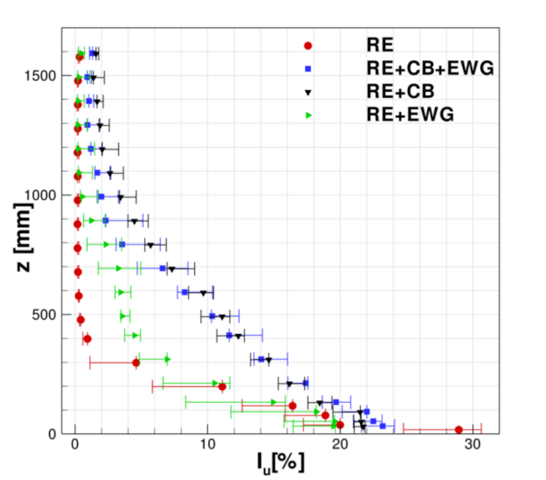

Figure 2. Turbulence intensity profiles of different arrangements in y=0, the bars represent the maximum variances of the quantity in y-direction.

The last solution, with all the devices (RE+CB+EWG), appears as the best tested arrangement in both uniformity level and the reproduced boundary layer height δ and power-law exponent α (1013 mm and 0,26).

Table 1. Main velocity profile parameters which represent the simulated boundary layer and the corresponding correlation coefficients.

ABL devices	δ [mm]	α	R_a^2	u_*[m/s]	R_{u*}^2	z_0[mm]	d [mm]	$R_{z0,d}^2$
RE+CB+EWG	1013	0,26	0,995	0,587	0,998	1,258	0	0,973
RE	398	0,39	0,993	0,654	0,997	2,520	0	0,979
RE+CB	991	0,20	0,996	0,480	0,992	0,279	0	0,944
RE+EWG	493	0,28	0,992	0,497	0,994	1,040	0	0,901

The Figures 3-4 show the power spectral density R_N at different heights (inside and outside the boundary layer) for the RE+CB+EWG arrangement, compared to the reference curves of Von Karman and Eurocode. The Figure 3 shows a good presence of a zone with the typical slope of -2/3 due to the effect of the spires, outside the boundary layer, in Figure 4, the behavior is completely different.

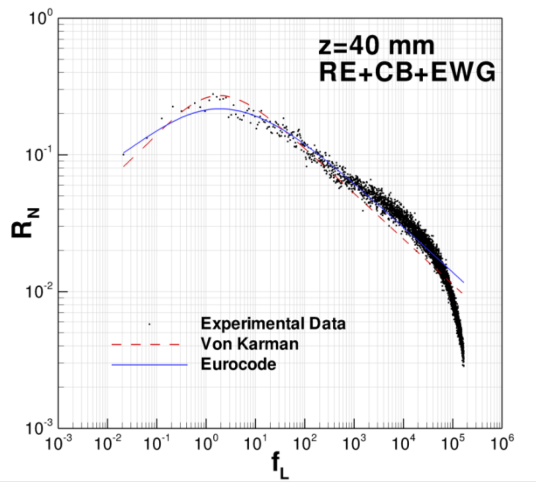

Figure 3. The power spectrum in RE+CB+EWG configuration compared with reference curves of Von Karman and Eurocode at z=40 mm.

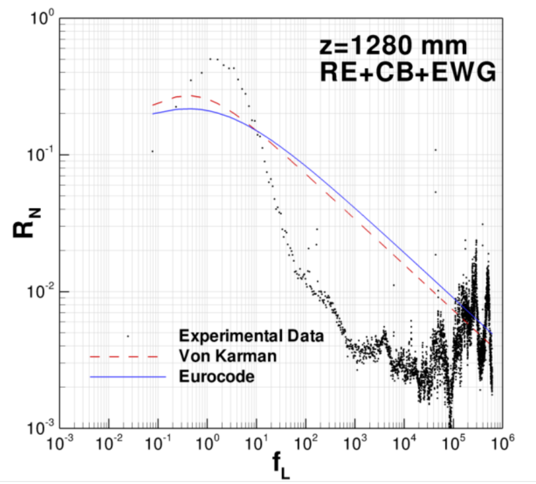

Figure 4. The power spectrum in RE+CB+EWG configuration compared with reference curves of Von Karman and Eurocode at z=1280 mm.

2.1.2 Analysis of the Counihan elliptic vortex generators

The elliptic vortex generators were designed by Counihan following observations and studies relating to the turbulent energy production inside the environmental boundary layer. The turbulent energy ε in the region near a surface is produced by the action of the Reynolds stresses τ_R and by the velocity gradient (du/dz), according to the following relation:

$$\varepsilon = \frac{\tau_R}{\rho}\frac{du}{dz} \tag{1}$$

Figure 5.The instrumented spire.

Counihan studied a device able to generate a high turbulence intensity using a high shear stress in the form of Reynolds stress. The lack of complete and easy to find information about their aerodynamic effects in a wind tunnel has motivated a measurement campaign. The analysis has been realized by using two different measurement techniques: pressure taps around the EWG surface and particle image velocimetry (PIV) to investigate the flow in the first centimeters downstream. In order to evaluate the flow field around a Counihan spire it was necessary to realize a particular spire (Figure 5) which could contain many pressure taps. The spire behaves as an aerodynamic body, the flow is attached on the side surfaces where gradually loses pressure and accelerates. On the back the spire behaves as a drag body and the flow separation produces an extended negative pressure zone (negative C_p), in particular on the top the high extent of the negative pressure is caused by strong edge vortices. By using a castellated barrier wall the spire can work more efficiently, because the integration of trapezoidal components improves the behavior of the turbulent flow that interacts mostly with the side surfaces of the spire (Fig.6-7).

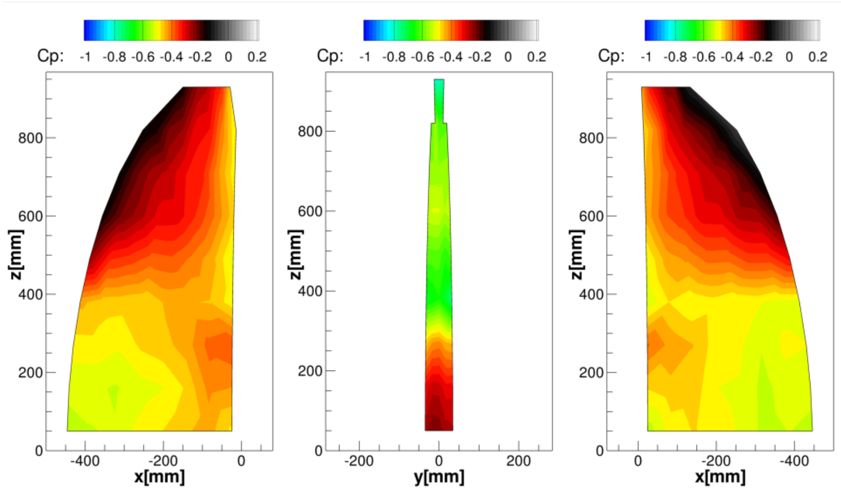

Figure 6. Two-dimensional representation of pressure distribution on the central spire in absence of lateral ones and in configuration with the barrier wall (simple profile).

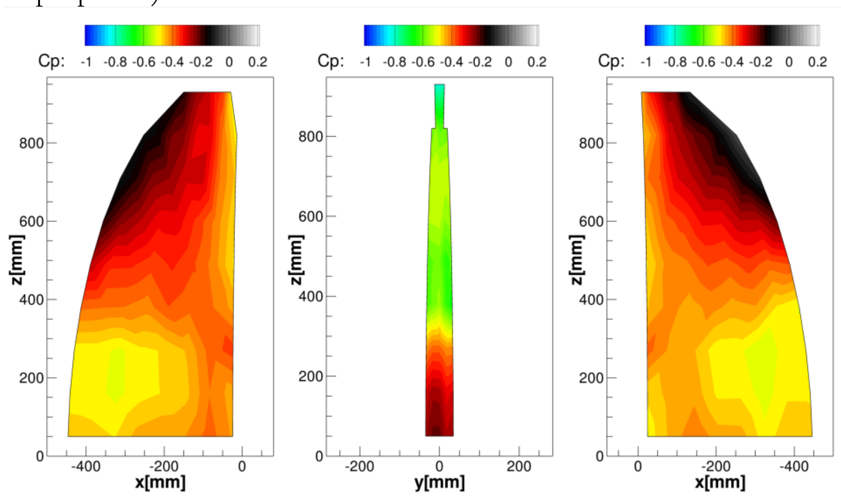

Figure 7. Two-dimensional representation of pressure distribution on the central spire in absence of lateral ones and in configuration with the castellated barrier wall.

The 2D and 3D PIV measurements confirm the alternating vortex separation (Fig.9), which becomes more non-stationary in the high area due to the presence of an edge downwash (Fig. 8), on the back it shows the presence of two counter-rotating stationary vortices (Fig. 10), the presence of the barrier wall modifies completely the flow around the spire, the fall of momentum gives a wake slowdown and distances the two eddies from the back, so the barrier causes also a great increasing of the turbulence intensity. In the lower part, the flow field is totally different because the spire is inside the separation zone caused by the barrier, the flow changes the motion direction moving slowly to the spire , without significant vertical components and climbing on its back(Fig. 11).

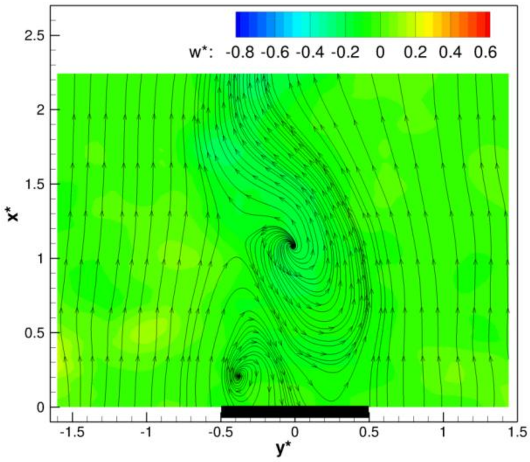

Figure 8. The 3D PIV result on a plane at z=720 mm without the barrier wall (Karman vortex street): the w* colored map with the x*-y* streamlines.

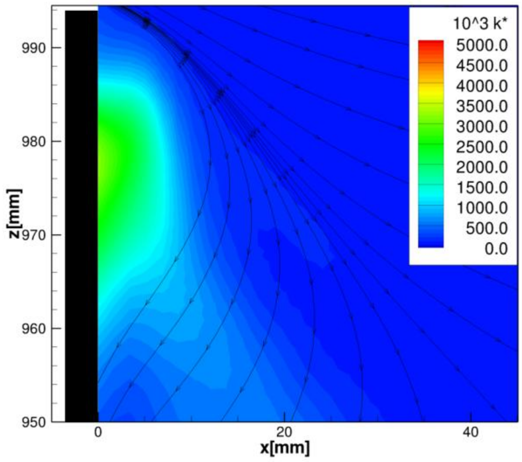

Figure 9. The 2D PIV statistical results on a vertical plane at the top of the spire without the barrier: the dimensionless turbulent kinetic energy k* colored map with the x*-z* streamlines.

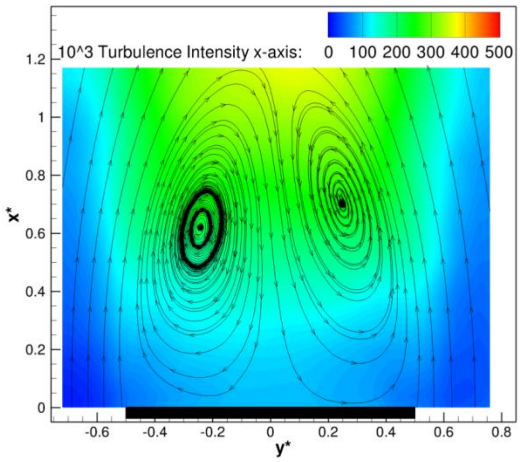

Figure 10. The 2D PIV statistical results on a plane at z=870 mm without the barrier wall: the turbulence intensity I_u colored map with the x*-y* streamlines.

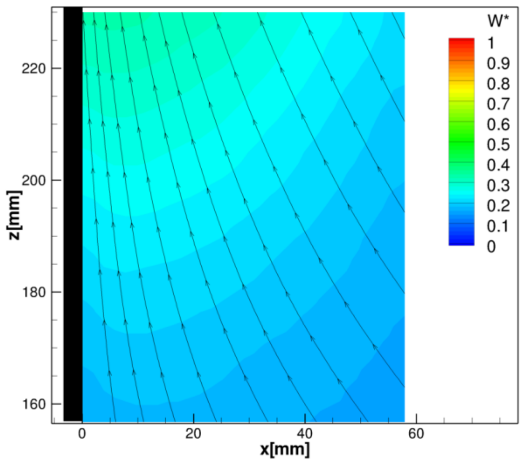

Figure 11. The 2D PIV statistical results on a vertical plane with the barrier wall: the W*colored map with the x*-z* streamlines.

2.2 Numerical approach in environmental applications

The application of the Computational Fluid Dynamics (CFD) to the study of environmental flows brings undoubted benefits. In the RANS approach the choice of a turbulence model rather than another can only be justified through experimental test cases. In this work, different commercial CFD codes (general purpose type) as Fluent and Star CCM+, and the open-source code OpenFoam, are presented in two typical test cases (2D and 3D) for environmental applications.

2.2.1 Test case 2D

The periodic two-dimensional hills of Almeida [5] is one of the most popular test case, considered as a reference for the analysis of separated flow.

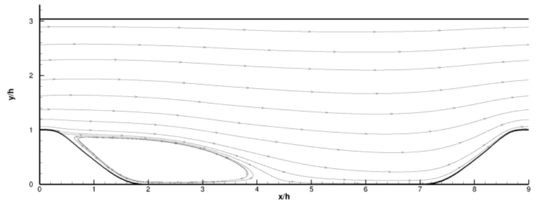

Figure 12. The streamlines corresponding to the flow calculated in OpenFOAM with the realizable k−ε turbulence model.

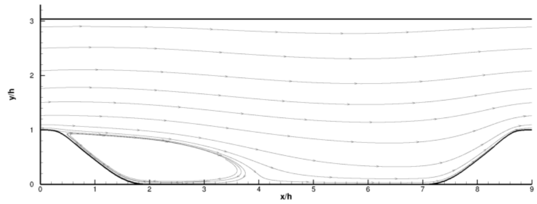

Figure 13. The streamlines corresponding to the flow calculated in OpenFOAM with the RSM turbulence model.

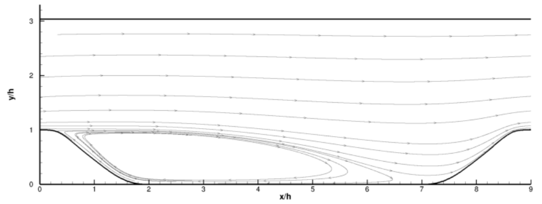

Figure 14. The streamlines corresponding to the flow calculated in OpenFOAM with the SST k−ω turbulence model.

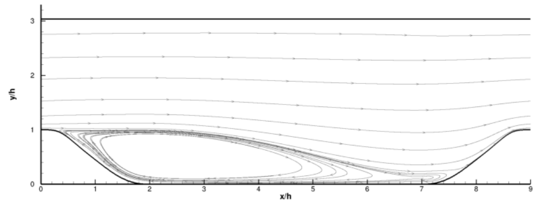

Figure 15. The streamlines corresponding to the flow calculated in OpenFOAM with Spalart-Allmaras turbulence model.

The SST k−ω, the Spalart-Allmaras model, RSM (Reynolds Stress Model) and the realizable k−ε were applied in this test case. All the models have shown a detachment of the flow but, while the last two models have correctly estimated the reattachment point (Fig.12-13), the SA model and the SST k−ω have provided a too large recirculation zone overestimating very significantly the separation (Fig. 14-15).

The results for the commercial codes and the open source code have been very close to each other.

2.2.2 Test case 3D

The second phase has provided the 3D test case of the hill of Askervein in Scotland [6]. In this case turbulence models were the RSM and the cubic k−ε for Star CCM+, while for OpenFoam the RSM and the standard k−ε.

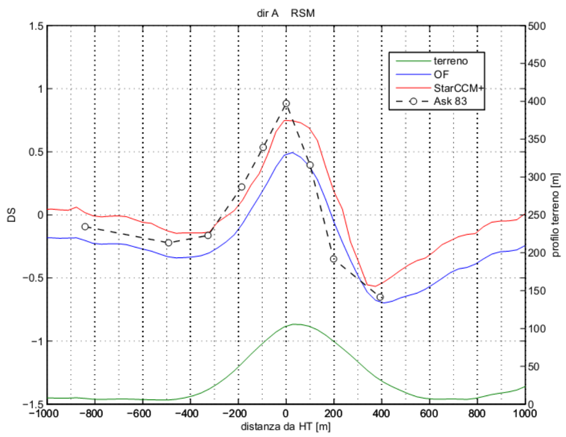

Figure 16. Speed up at 10 m along the A direction (RSM simulation).

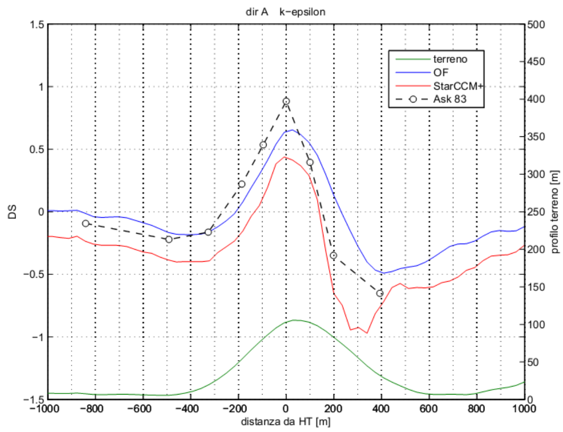

Figure 17. Speed up at 10 m along the A direction (k−ε simulation).

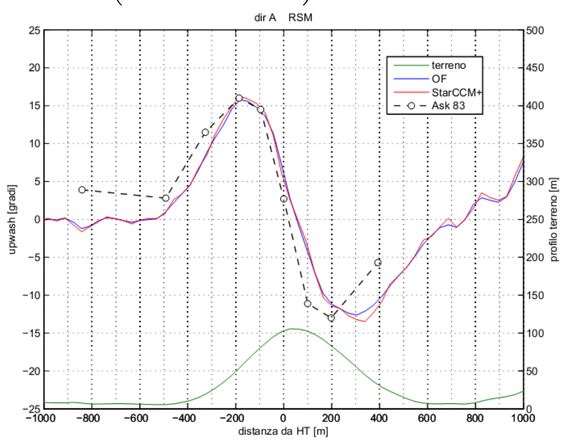

Figure 18. Upwash at 10 m along the A direction (RSM simulation).

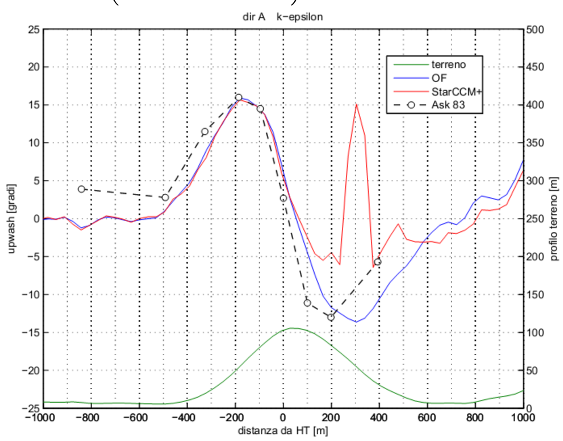

Figure 19. Upwash at 10 m along the A direction (k−ε simulation).

Along the main directions, the RSM model has shown a good matching with the experimental data (Fig. 16-18), both in the calculation of topographic multipliers and in the wind direction. The cubic k-ε model has shown, on the other hand, an excessive dissipation with reduced flow accelerations, a low turbulent kinetic energy, and also the presence of a separation zone downstream of the hill, which most likely does not exist (Fig.17-18). The OpenFoam code has highlighted results concurring with the experimental data, in this case with some differences with respect to the results calculated by Star CCM+. The numerical results from the RSM model of the commercial code are closer to the experimental data, but the most important difference is in the k−ε simulation: the standard formulation for high Re of OpenFoam has shown acceptable results with respect to the cubic k−ε results of Star CCM+, without the separation zone downstream of the hill.

2.3 Numerical – experimental comparison on a case study

The last part of the work was focused on an application case; an area of the district of Ancona destined to a neighborhood of new construction was chosen. The chosen area presents a complex terrain topography which, combined with the built environment, generates a significant interaction thus representing an excellent case study. Preliminarily, a mesoscale weather prediction was carried out to estimate the prevalent wind directions.
The prediction was obtained using the MM5v3 code, improved by our research group; it allows to reach a very detailed resolution with horizontal spatial grids up to 200 m.
The results showed that the prevalent wind direction is West. It does not show strong seasonal characteristics, being equally present in both summer and winter months.

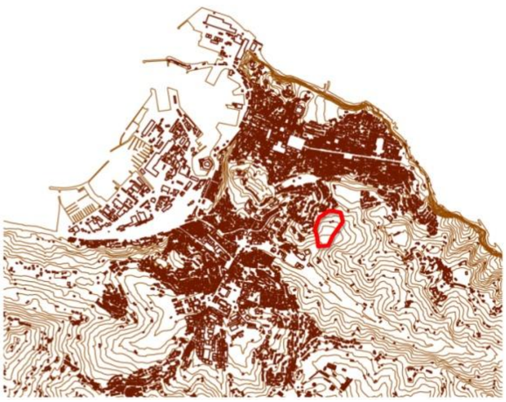

Figure 20. The investigated area.

Figure 21. The prevalent winds calculated using MM5 code.

The experimental analysis was carried out in the environmental wind tunnel on a scale model 1:500. The study was performed by using anemometric probes which allow measuring the velocity components on both vertical and horizontal planes. In order to carefully evaluate all interactions, the test was carried out first on the model with the only topography terrain and then performed with the constructions appropriately scaled.

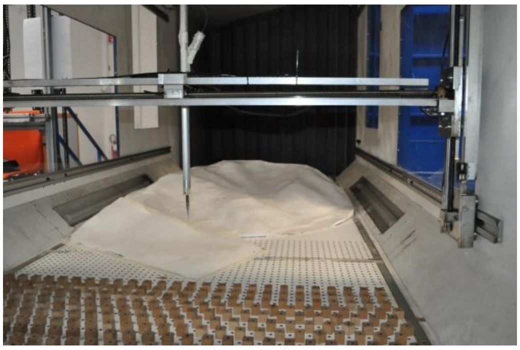

Figure 22. The model positioned inside the test chamber.

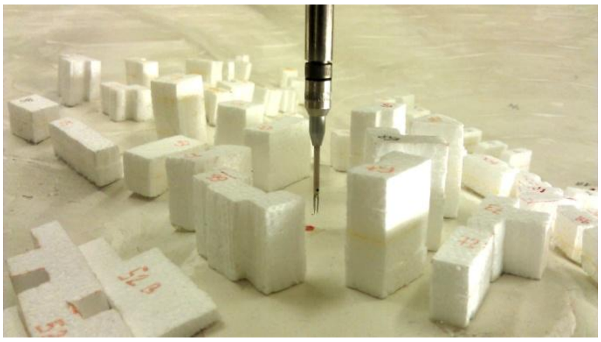

Figure 23. The anemometric probe during a measurement.

From the observation of the real terrain it was decided to set up a suburban – rural wind profile, characterized by a power-law exponent α of 0,2.

In Figure 25 the results of the only terrain model (with reference to Figure 24) have shown, in the neighborhood zone, an inclination on the horizontal plane of the main flow; therefore, it does not appear aligned with the West direction anymore. Also an acceleration due to the hill that precedes the studied area is highlighted.

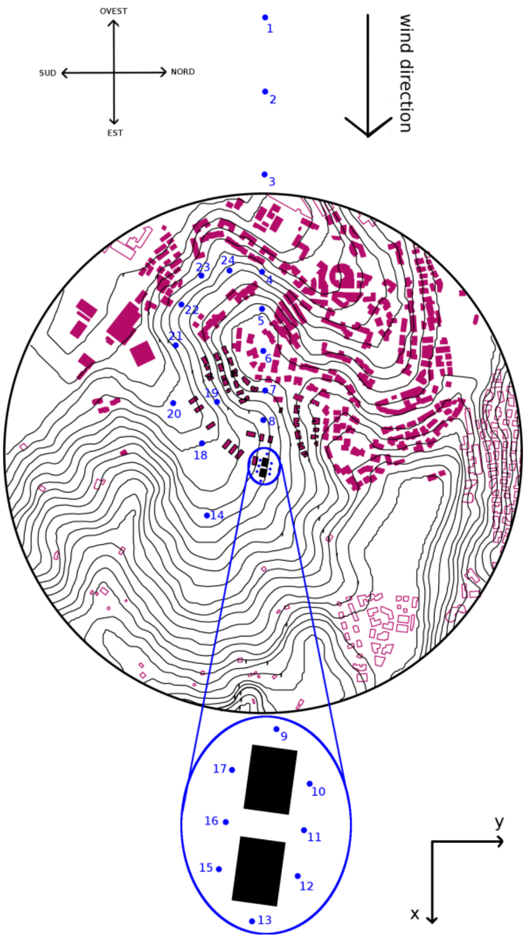

Figure 24. The analysis points.

In Figure 26 the presence of the buildings modifies significantly the flow field. It produces, in addition to an expected loss of momentum at the low altitudes, also greater inclinations of the main flow that arrive at angles up to 20°.

The modeling of the problem, unlike the experimental test, did not provide only the area in the domain of interest, but a wider area, which encloses the entire topography of the city of Ancona, including the port area. The simulations were performed using the commercial code Fluent and setting RSM as turbulence model, which had worked well in the previous test cases .The calculations have confirmed the experimental results in the wind tunnel although all the phenomena related to the presence of buildings, such as the inclination of the main flow in the area of interest, have appeared much more restrained (Fig. 27).

The comparison between the experimental and numerical data has highlighted one of the limits, probably the main one, of the experimental approach, that is the absence of

Reynolds analogy with the real problem. In order to better analyze this matter and the consequences of this approximation, a numerical simulation was carried out in the Reynolds analogy with the wind tunnel setting a viscosity 500 times greater. The results (Fig. 28-29) show trends in excellent agreement with the experimental data, and evidence as in the wind tunnel the aerodynamic disturbance phenomena, that an obstacle (in this case the buildings) gives to the incident flow, are accentuated (Fig. 30-31).

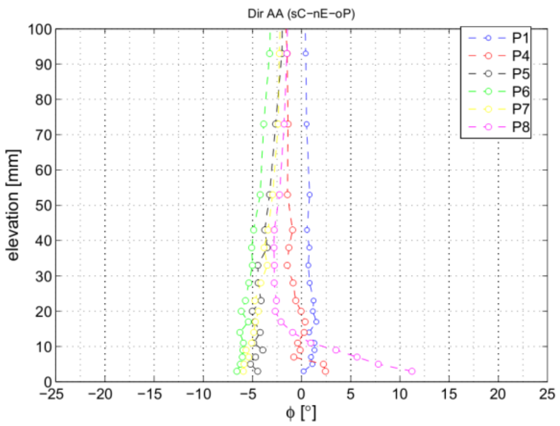

Figure 25. Wind profile in the main flow direction (with the only terrain topography).

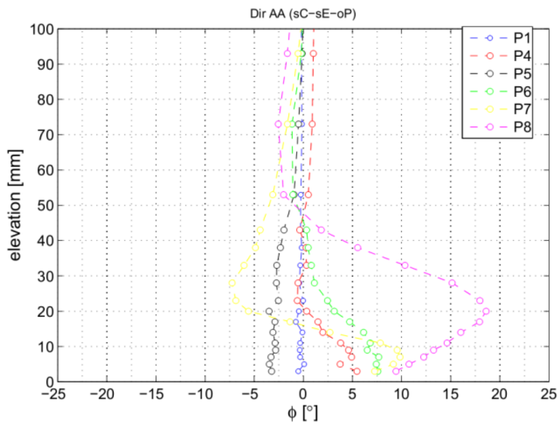

Figure 26. Wind profile in the main flow direction (with all the buildings).

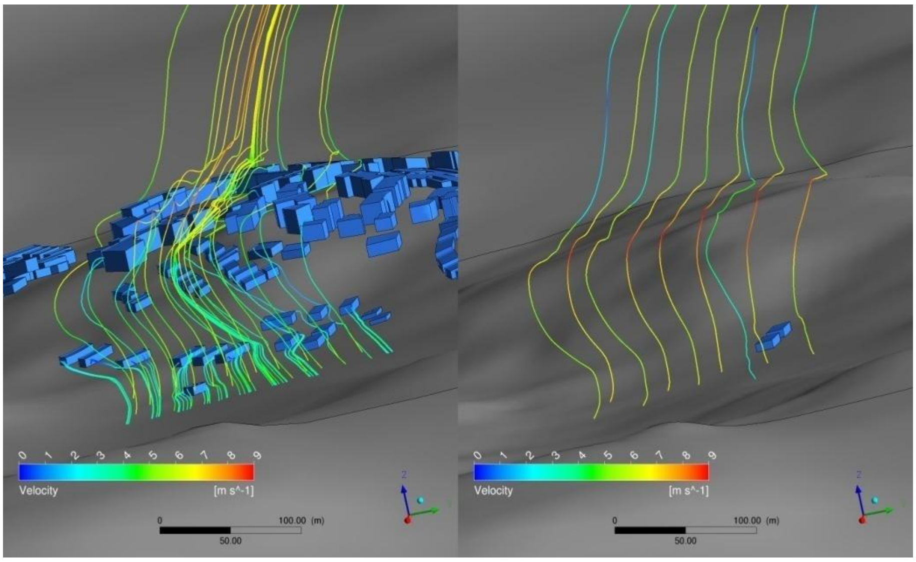

Figure 27. Comparison of streamlines of the area of interest (numerical simulations with and without the buildings).

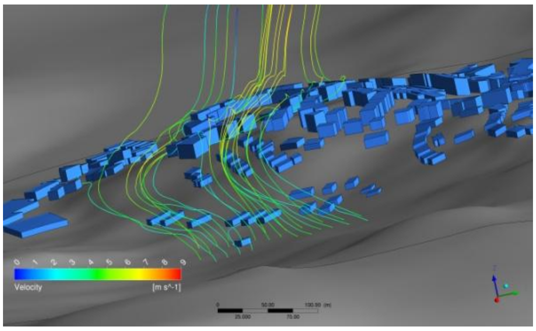

Figure 28. Streamlines in the Reynolds analogy with the wind tunnel.

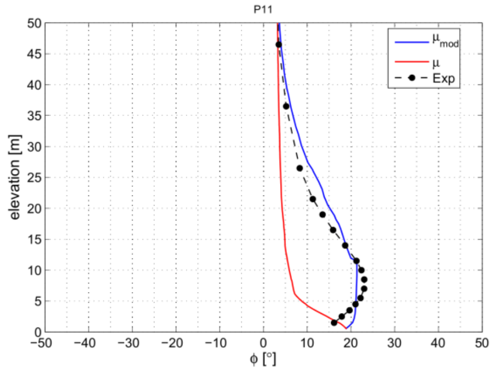

Figure 29. Comparison between the experimental and numerical results of the angle on the horizontal plane (P11 point).

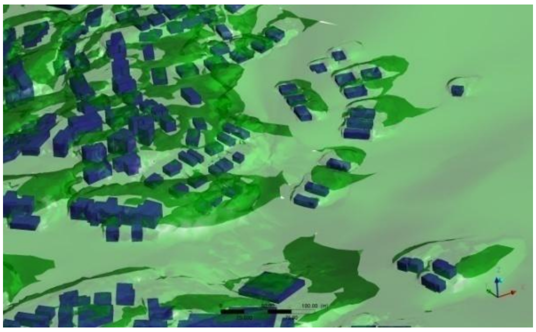

Figure 30. Iso-vorticity layers Ω= 0,01 in the configuration with all the buildings in the simulation with the standard viscosity.

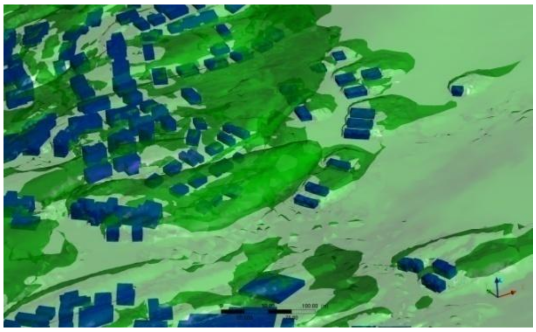

Figure 31. Iso-vorticity layers Ω= 0,01 in the configuration with all the buildings in the simulation with the modified viscosity.

3 Conclusions

The study of microscale wind resource is gaining more and more importance both in energy applications and in the context of sustainable design. In most cases, field measurement activities are not applicable, so it requires experimental tests in the wind tunnel or purely numerical approaches. This work highlights that both approaches can potentially lead to good results, but they are also characterized by significant limits: for example the inaccurate choice of numerical models or the use of scale models may lead to grave errors. The combined approach, however, can bring undoubted benefits.

References

[1]. Garratt, J.R.*The atmospheric boundary layer.* Cambridge : Cambridge University Press, 1992.

[2]. *An improved method of simulating an atmospheric boundary layer in a wind tunnel.* Counihan, J.3:197–214, Atmospheric Environment, 1969.

[3]. *Simulation of an adiabatic urban boundary layer in a wind.*Counihan, J. 7:673–689, Atmospheric Environment. 1973.

[4]. *The simulation of the atmospheric boundary.* Counihan, J. Armitt and J. 2:49–71, Atmospheric Environment, 1968.

[5]. Almeida, G.P. 2d model hill wake flows. ERCOFTAC Classic Database - Exp C-18., 1990.

[6]. Teunissen, P.A. Taylor and H.W. The Askervein hill project. International Energy Agency Programme of R&D on Wind Energy, 1983.