Available online at www.sciencedirect.com

ScienceDirect

Energy Procedia 126 (201709) 628–635

Energy Procedia

www.elsevier.com/locate/procedia

72nd Conference of the Italian Thermal Machines Engineering Association, ATI2017, 6-8 September 2017, Lecce, Italy

Design of a novel open space test rig for small scale wind turbine

Amirante R.[a*], De Palma P.[a], Distaso E.[a], M. La Scala[b], Tamburrano P.[a]

[a]*Department of Mechanics, Mathematics and Management (DMMM) Politytechinc University of Bari, Italy*
[b]*Department of Electrical and Information Engineering Politytechinc University of Bari, Italy*

Abstract

In the present paper, an innovative and cost-effective open test rig for small and medium wind turbines is proposed. The main aim is to develop a valid alternative to wind tunnels, which present unresolved problems such as the unmatched Reynolds numbers for downscaled wind turbine tests. The proposed test bench concept is an open field, subsonic facility for horizontal and vertical axis wind turbines. The core of the test bench is a cluster of axial fans, positioned at a given height from the ground, which generate an air flow suitable for testing a wind turbine placed in front of the fans. The present work aims at investigating the feasibility of this novel concept of test rig for small wind turbines having a rotor diameter smaller than 5 m. A thorough CFD analysis is performed in this paper in order to assess the characteristics of the wind generated by the fans in terms of uniformity and intensity, even in case of atmospheric disturbances. The developed CFD modelling is also instrumental in both determining the maximum rotor diameter that can be tested and selecting the correct position for a wind turbine in the proposed open test rig.

Peer-review under responsibility of the scientific committee of the 72nd Conference of the Italian Thermal Machines Engineering Association

Keywords: Small scale wind turbines; Test rig in open field; CFD Simulations; Energy microgeneration.

---

1. Introduction

Energy production is changing in the world because of the need to reduce the greenhouse gas emissions, and the dependence on carbon/fossil sources [1,2]. Renewable energy sources have become one of the most important focus for the development of new energy concepts [3].

The reasons that can explain the increasing attention to renewables are the depletion of fossil fuels and the necessity of offering a best future to 4,5 billion of people who today have limited access to energy resources, and this is also in line with the international environment treaties aimed at reducing world pollution and global warming, i.e. Kyoto

---

[*] *Corresponding Author: Riccardo Amirante,*
e-mail address: amirante@poliba.it

1876-6102
Peer-review under responsibility of the scientific committee of the 72nd Conference of the Italian Thermal Machines Engineering Association
10.1016/j.egypro.2017.08.249

Protocol and EU 2020 [4]. In the next future, it is expected that most industries will produce energy for their own use by means of renewable sources, coupling the energy production with storage systems [4] to overcome the discontinuity in the availability of renewable sources.

One of the most promising and useful strategy consists in the employment of small size wind turbines, thus installing decentralized grid systems [5,6]. Small-scale wind turbines can be used as a reliable source of energy if they are properly sized. They are suitable for some autonomous applications that require a very high level of reliability; moreover, they can also become a generation source of socio-economically valuable energy for most of the developing Countries [5]. In addition, thanks to their small dimensions and attractive geometries, small/medium scale wind turbines are suitable for an urban environment because of their cut-in wind speed, which is lower than that of large wind turbines. This peculiarity shows good compatibility with different scenarios of application.

Most of the development of the design concepts for modern wind turbines is achieved by wind tunnel testing. Wind tunnel investigations present important features, such as the reproduction of a broad range of well-characterized wind conditions. However, in many cases this experimental approach has been hindered by the large size of blade models. As an example, a turbine generating power of the order of some MWs, has blades with a diameter larger than 50 m; on the other hand, the conventional wind tunnel facilities have diameters smaller than 10 m. Therefore, a scaled down model for the experimental studies of wind turbine blades is needed. This leads to the scaled effect, which comes from the fact that the Reynolds number in the experiment is significantly smaller than the real value [7][8][9]. In addition, the natural variability in the Atmospheric Boundary Layer flow, such as variations in wind velocity and direction, turbulence intensity, spatial heterogeneity and thermal stability, is difficult to be entirely reproduced through wind tunnel experiments [10]. Moreover, the wind tunnel walls do not allow one to reproduce the characteristic conical shape of the flow around the turbine rotor disk. The walls around turbines are geometrical constraints forcing the stream lines direction. As a result, the test section should be designed much larger than the turbine diameter to reproduce an open field condition.

Several research studies have been carried out to date in order to solve the problem related to the unmatched Reynolds numbers for downscaled wind turbine tests in wind tunnels. Makita and Sassa [11] proposed to employ a turbulence generator capable of inducing a certain degree of turbulence in the flow. An active grid system was used in [12] as a turbulence generator in an attempt to minimize the impact of the unmatched Reynolds number [12]. Turbulence grids were also used in [13] to adjust the turbulence intensity.

Rather than focusing on improving wind tunnels to overcome the above-mentioned limits, this paper proposes an innovative way of testing wind turbines, which consists on using cost-effective, open field test rigs, very different from conventional ones. As to date, the scientific literature has not highlighted effective wind tunnel tests capable of reproducing an open bound test rig, the task is very challenging and, at the same time, can represent a great innovation in wind turbine tests.

Comparing conventional systems (wind tunnels) with the proposed one (open test rig), it is clear that the latter can eliminate a lot of elements that induce a major usage of resources in term of both costs of installation and time for designing the facilities. In fact, a typical wind tunnel mainly consists of a convergent duct followed by a divergent one, which are usually equipped fans, honeycombs, and other channels to guide the air. Furthermore, typical wind tunnels are usually characterized by large lengths which can be as high as 30 m [14,15]. The main advantage of the proposed solution over common wind tunnels is the possibility of analysing the real effect of open field surroundings and all the associated phenomena, such us the real wind speed, turbulence effects and atmospheric conditions. Moreover, another important advantage is the absence of the scale effect because rotors can be tested with their real geometry. In addition, as confirmed by [16,17], the blockage effect would not be encountered, therefore the turbine power output would not be affected by any increase in velocity due to the wind tunnel walls.

The realization of such a system will be part of a net of laboratories called “Zero Emission Research Option” (ZERO), which the Polytechnic University of Bari and Apulia Region decided to found in order to study the conversion and management of energy obtained from renewable sources. In addition, the laboratory will work to promote the development of the smart grid technology in a smart city context. In this regard, lab-ZERO will be integrated with a micro co-generative heat and power plant [18–21] fuelled with carbon-neutral biomass (such as pruning residues [22]) to provide energy to all of the facilities.

In the present work, a thorough 3-D Computational Fluid Dynamics (CFD) approach has been employed with the aim of accurately predicting the air flow generated by the fans and its interaction with the external environment. This analysis aims at determining the correct distance at which the wind turbine must be placed in the proposed open test rig and the maximum diameter of the rotor that can be tested.

2. Methodology

2.1. Open space test rig description

Fig. 1-a and b show a 3-D representation and a picture of the recent innovative test bench, built in October 2016, respectively. The location of experiments is a 40 m^2 site in the area of Polytechnic University of Bari. As shown Fig. 1-b, the test rig is placed in an open field surroundings without any kind of obstacles and urban impediment.

The proposed test bench is suitable for testing both horizontal and vertical axis wind turbines. The test bench concept consists of a wind tower equipped with a cluster of axial fans, positioned 7 m above the ground, which can provide an adjustable airflow rate to a wind turbine to be placed in front of the cluster. The wind tower holds the rotor and the nacelle, where all the power transmission components (gearbox, bearings, alternator, and shaft) are arranged. The position of the turbine can be adjusted by moving the turbine on a rail.

The general concept can be applied to different situations, provided that the characteristics of the cluster of the axial fans (number of fans and dimensions) are adapted to the size of the turbines to be tested. The test rig will allow the flow rate (and hence the air velocity) to be adjusted finely thanks to an electrical drive connection for each fan, providing great accuracy in the control of the flow-rate blowing out. This control could be valuable for future experimentations when different rotor geometries will be tested along with novel control techniques [23].

The velocity of the wind produced by the fans could be not uniform and affected by vorticity and thus might not be suitable to test the turbine. This undesired phenomenon will occur either if the turbine is placed too close to the fans matrix or if the turbine is very far from the fans matrix (see Fig. 2). Moreover, since the system works in an open-field domain, it is crucial to take into account the effect of an atmospheric wind, which could blow orthogonally to the airflow generated by the fans. Therefore, 3-D CFD analyses were performed with the aim of characterizing the velocity profile of the jet, as well as of evaluating the minimum and the maximum distance from the fans at which the wind turbine should be placed and its maximum diameter. Therefore, the flow blowing from a single rotating fan was firstly analysed, in order to study the tangential components generated by the fan motion; then, the influence of a lateral wind blowing orthogonally with respect to the axial direction was investigated by simulating the entire fan matrix in steady-flow conditions, for simplicity. The CFD analysis was performed by means of the commercial code ANSYS FLUENT-17.1. For the simulations, the incompressible Navier-Stokes equations were solved and the RANS k-ε turbulence closure model was employed; a pressure-based solver and a second-order accuracy in space and time were also adopted.

2.2. CFD set-up

The computational domain employed for evaluating the flow field generated by a single fan is shown in Fig. 3-a. Fig. 3-b offers a detail of the unstructured computational mesh close to the fan. The grid was composed of about 1,5 million elements. The outlet boundary is located 6 meters from the outlet section of the fan. A rotating mesh was used to perform transient simulations of the fan 1400 rpm. A pressure equal to 1 bar was set on each boundary of the computational domain. A time-step of 0,0043 s was sufficient to guarantee a stable and accurate solution. In order to reach the steady-state conditions, 400 time-steps were needed (500 iteration were performed in each of them).


(a) (b)

Fig. 1. Test bench rendering (a) and test bench, under construction (b).

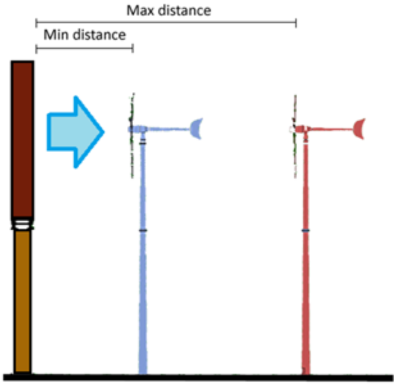


Fig. 2.: The minimum and maximum distance between wind turbine and fans.

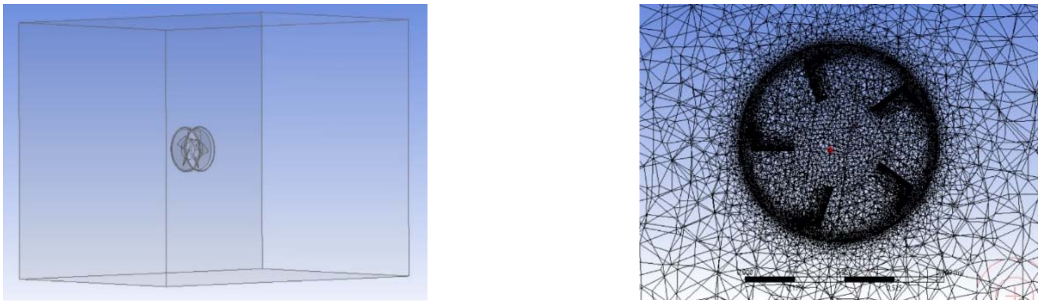
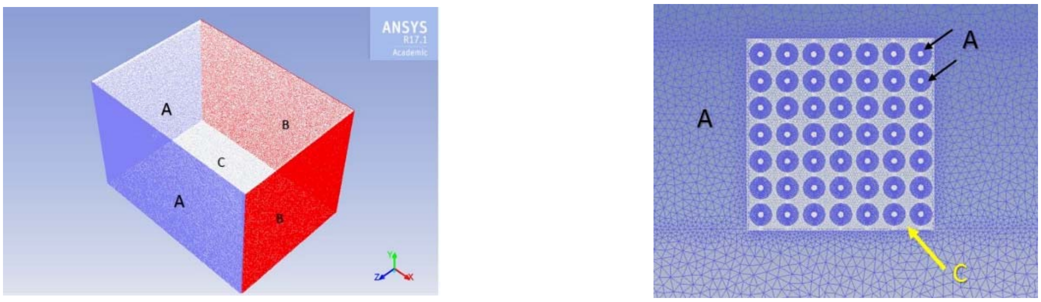

(a) (b)

Fig. 3. Geometrical Domain employed in the CFD analysis of the flow generated by a single fan (a). Detail of the mesh around the fan, belonging to a cross-section-plane of the computational domain.



(a) (b)

Fig. 4. Computational domain and boundary conditions adopted in the simulations investigating the influence of a lateral wind. A: inlet boundaries; B: Outlet boundaries; C: Walls.

Concerning the second simulation, the computational domain was a box with dimensions 70x45x70 in x, y and z directions, respectively (see Fig. 4). With the purpose of respecting the design architecture of the real test bench, the cluster of fans was positioned at 7 meters from the ground. The surfaces labelled with A in Fig. 4-a indicate the two inlet boundaries, including the annular fan sections. One contains the outlet surface of the cluster and the surrounding inlet region, as shown in Fig. 4-b; the other is the lateral inlet boundary of the domain. The surfaces labelled with B represent outlet boundaries. The surfaces indicated with C are solid walls, whereas the top boundary was modelled using a free-shear wall condition. An axial velocity of 12 m/s was set in the annular sections of Fig. 4, whereas a velocity of 0.5 m/s was imposed at the inlet points surrounding the cluster. A historical study about the wind intensity in the Apulia region led to the consideration that an average wind velocity within the range of 3.5÷5 m/s can represent a reasonable assumption. In particular, a maximum speed of 4 m/s, aligned with the z-direction, was imposed at the lateral boundary of the domain.

3. Numerical results

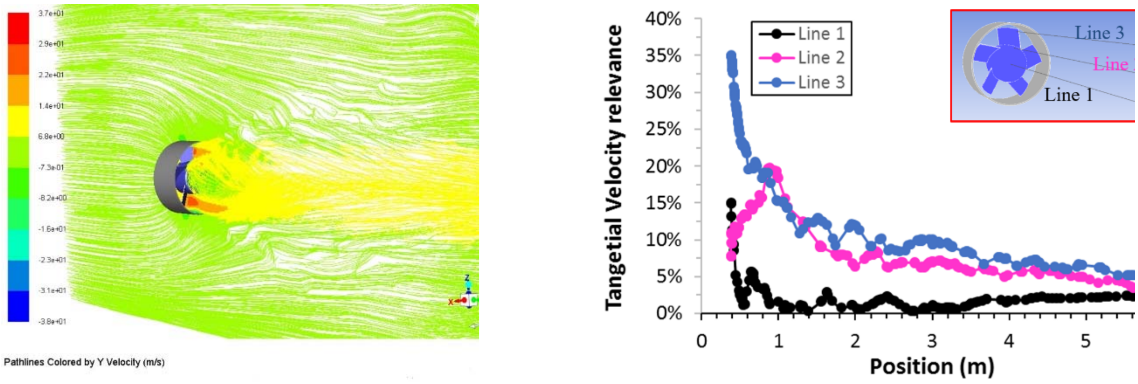
3.1. Analysis of the flow generated by a single fan

Fig. 5-a provides the stream-lines of the flow generated by the single fan rotation, coloured by the axial velocity magnitude, as obtained by the numerical simulation. The flow appears highly turbulent; the velocity is very low near the central axis of the rotor, while near the tips of the blades it has a relevant tangential component. Considering the velocity distributions along the three control lines shown in Fig. 5-b, it is possible to observe that the percentage fraction of the tangential velocity, computed with respect to the axial component of the velocity, is large at the outlet section of the fan, but then it gradually decreases, to a value of 10% after about 2.5 m. This preliminary analysis leads to the conclusion that the wind velocity core reaches an almost one-dimensional profile at a distance of 2.5 m from the fans.

3.2. Analysis of the flow generated by the cluster

The distribution of the velocity components has been analyzed along seven horizontal control lines starting from the inlet sections of the cluster, as shown in Fig. 6. Fig. 7 shows that after almost 6 m from the inlet, it is possible to observe a progressive decay of the axial component of the velocity (x-velocity). Lines 1 and 7 are the most external ones and, as a result, the velocity magnitude decrease is larger. These results highlight the crucial role played by the estimation of the maximum distance at which the turbine can be placed.

Fig. 8-a and b show the spatial evolution of the y- and z-component of the velocity, respectively. The y-component, measured along each of the seven control lines, is smaller than the x-component. The z-component starts to be relevant at about 7.5 meters from the cluster and the more the distance from it increases, the more the presence of a lateral wind influences the velocity field.



(a) (b)

Fig. 5. Stream-line of the flow generated by a single fan after 400 time-steps (a). Percentage fraction of the tangential with respect to the axial component of the velocity, computed on 3 lines, starting from the outlet section of the fan (b).

With the purpose of understanding how the lateral wind influences the flow field, a dimensionless performance coefficient was considered. This parameter is called *"deviation angle"* (Dev%) and is calculated as the percentage ratio of the z- to the x-velocity components. Fig. 9 reports the values of the *deviation angle* for all the considered control lines. Considering a threshold value of 10% for *Dev%*, it is noticeable that, except for the control line 1 (which represents a zone widely exposed to the external wind), this threshold was not overtaken along the remaining lines for a distance lower than 4.7 m. for the cluster, this value should be regarded as the maximum distance at which the turbine should be placed in the case of lateral wind blowing at 4 m/s. Beyond this value, the lateral component of the velocity becomes important and hinder the correct testing of a wind turbine.

By means of this numerical analysis, it was possible to infer that the effect of a wind, transversally blowing with respect to the axial direction of the fans composing the cluster, cannot be neglected and its estimation is needed for selecting the correct space interval in which the turbine can be placed.

Finally, the CFD predictions were used to determine the maximum diameter of the turbine rotor that can be tested for the considered characteristics of the test rig. To this purpose, the parameter δ%, called "*Overlap Factor*", was employed. It is calculated as the ratio of the difference between the maximum axial velocity (v_{max}) and the local axial velocity (v_i) to the maximum axial velocity (v_{max}), namely:

$$\delta\% = \frac{V_{max} - V_i}{V_{max}}.$$

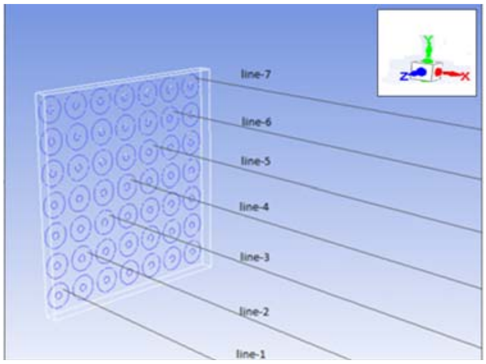
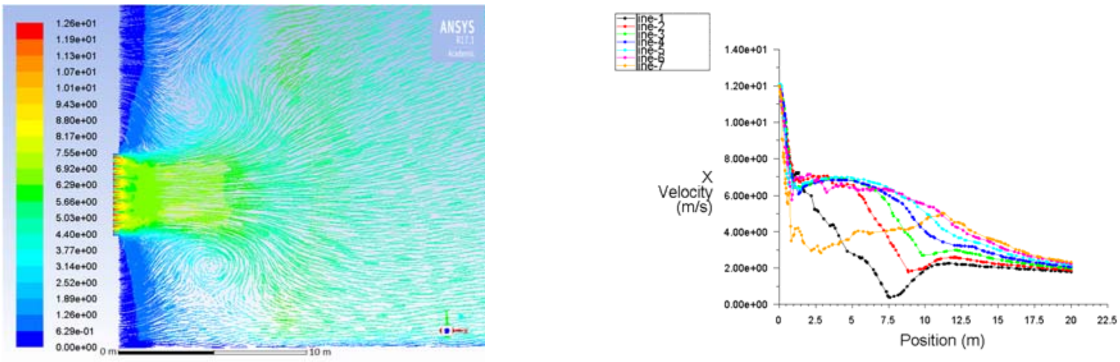


Fig. 6. Cluster space domain with seven control lines.



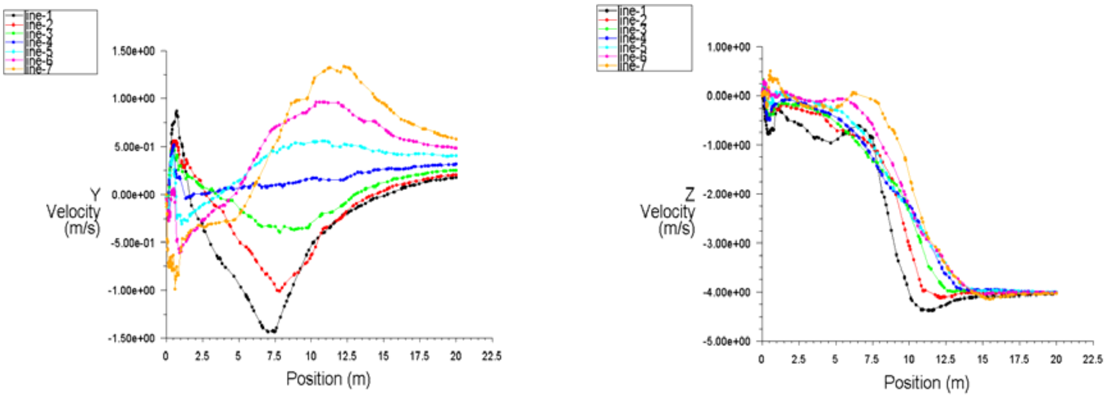
(a) (b)

Fig. 7. Stream-lines coloured by velocity magnitude [m/s], in XY cross plane, in presence of lateral disturbance (a). X-velocity evolution for the seven control lines (b).

Choosing an overlap factor value equal to 10% as the threshold value, the maximum dimension of the rotor is found. For a testing plane placed at 2.5 m from the cluster, the maximum rotor diameter recommended is 4.6 m.

4. Conclusions

The present paper described a novel cost-effective experimental test bench designed to perform the wind turbine testing in an open-field. The innovative idea is to take advantage of a cluster of composed of a matrix of 49 fans, placed 7 m above the ground and producing a jet in open air, which avoids the undesired phenomena occurring in wind tunnels, such as the unmatched Reynolds numbers. A wind turbine is to be placed in front of the fans and its position can be regulated by moving the turbine on a rail. Such testing platforms will give great flexibility to the experimental investigations of wind turbines since it will also be possible to regulate the wind flow-rate, thus allowing wind turbines to be tested for different operating conditions.



(a) (b)

Fig. 8. Y-velocity (a) and Z-velocity (b) evolution for the seven control lines, in presence of lateral disturbance.

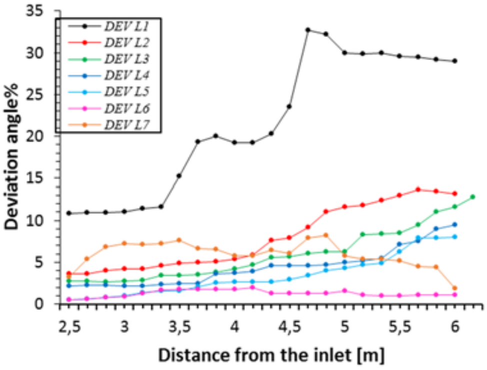


Fig. 9. Deviation angle variation with the distance from the velocity inlet source.

By means of full 3D CFD analyses, the reliability of the system was demonstrated. The flow blowing from a single rotating fan was first analysed by means of unsteady simulations, in order to accurately study the vorticity generated by the fan motion and the results showed that the air flow provided by means of the fans can be considered approximately one-dimensional at a distance larger than 2.5 meters. Then, the influence of a lateral wind blowing orthogonally with respect to the axial direction of the fans was investigated by simulating the entire fan matrix in steady conditions and the results suggested that the estimation of the maximum distance at which the turbine ca be placed plays a crucial role. In particular, it was found that the distance within it is convenient to place the turbine ranges between 2.5 and 4.7 m. These conditions are optimal to test any wind turbine (having either horizontal or vertical axis) having a maximum rotor diameter of 4.6 m for the considered test facility.

Acknowledgements

This work has been supported by "Regione Puglia" under contract "Progetto Reti di laboratori pubblici di ricerca APQ Ricerca Scientifica – II atto integrativo Fase A – (DGR n.92 del 27/11/2007) – titled: Laboratorio per lo sviluppo delle fonti rinnovabili e dell'efficienza nei distretti energetici: progetto zero (Zero Emission Research Option)".

References

[1] Amirante R, Distaso E, Di Iorio S, Sementa P, Tamburrano P, Vaglieco BM, et al. Effects of Natural Gas Composition on Performance and Regulated, Greenhouse Gas and Particulate Emissions in Spark-Ignition Engines. Energy Convers Manag 2017;143:338–47.

[2] Amirante R, Distaso E, Tamburrano P, Reitz RD. Measured and Predicted Soot Particle Emissions from Natural Gas Engines. SAE Tech Pap 2015-24-2518 2015. doi:doi:10.4271/2015-24-2518.

[3] Amirante R, Coratella C, Distaso E, Tamburrano P. A small size combined system for the production of energy from renewable sources and unconventional fuels. Energy Procedia 2015;81:240--248. doi:10.1016/j.egypro.2015.12.090.

[4] Amirante R, Cassone E, Distaso E, Tamburrano P. Overview on recent developments in energy storage: Mechanical, electrochemical and hydrogen technologies. Energy Convers Manag 2017;132:372–87. doi:10.1016/j.enconman.2016.11.046.

[5] Tummala A, Velamati RK, Sinha DK, Indraja V, Krishna VH. A review on small scale wind turbines. Renew Sustain Energy Rev 2016;56:1351–71. doi:10.1016/j.rser.2015.12.027.

[6] Wang C, Prinn RG. Potential climatic impacts and reliability of very large-scale wind farms. Atmos Chem Phys Discuss 2009;9:19081–102. doi:10.5194/acpd-9-19081-2009.

[7] Cho T, Kim C. Wind tunnel test for the NREL phase VI rotor with 2m diameter. Renew Energy 2014;65:265–74. doi:10.1016/j.renene.2013.10.009.

[8] Cho T, Kim C. Wind tunnel test results for a 2/4.5 scale MEXICO rotor. Renew Energy 2012;42:152–6. doi:10.1016/j.renene.2011.08.031.

[9] Schepers JG, Boorsma K, Cho T, Gomez-iradi S, Schaffarczyk P, Jeromin A, et al. Final report of IEA Task 29, Mexnext (Phase 1): Analysis of Mexico wind tunnel measurements 2012:312.

[10] Iungo GV. Experimental characterization of wind turbine wakes: Wind tunnel tests and wind LiDAR measurements. J Wind Eng Ind Aerodyn 2016;149:35–9. doi:10.1016/j.jweia.2015.11.009.

[11] Makita H, Sassa K. Active turbulence generation in a laboratory wind tunnel. Adv. Turbul. 3, Springer; 1991, p. 497–505.

[12] Talavera M, Shu F. Experimental study of turbulence intensity influence on wind turbine performance and wake recovery in a low-speed wind tunnel. Renew Energy 2017.

[13] Qing'an L, Kamada Y, Maeda T, Murata J, Nishida Y. Effect of turbulent inflows on airfoil performance for a Horizontal Axis Wind Turbine at low Reynolds numbers (Part II: Dynamic pressure measurement). Energy 2016;112:574–87.

[14] Bottasso CL, Campagnolo F, Petrović V. Wind tunnel testing of scaled wind turbine models: Beyond aerodynamics. J Wind Eng Ind Aerodyn 2014;127:11–28. doi:10.1016/j.jweia.2014.01.009.

[15] Schümann H, Pierella F, Sætran L. Experimental investigation of wind turbine wakes in the wind tunnel. Energy Procedia 2013;35:285–96. doi:10.1016/j.egypro.2013.07.181.

[16] Bai CJ, Wang WC. Review of computational and experimental approaches to analysis of aerodynamic performance in horizontal-axis wind turbines (HAWTs). Renew Sustain Energy Rev 2016;63:506–19. doi:10.1016/j.rser.2016.05.078.

[17] Chen TY, Liou LR. Blockage corrections in wind tunnel tests of small horizontal-axis wind turbines. Exp Therm Fluid Sci 2011;35:565–9. doi:10.1016/j.expthermflusci.2010.12.005.

[18] Amirante R, Tamburrano P. Novel, cost-effective configurations of combined power plants for small-scale cogeneration from biomass: Feasibility study and performance optimization. Energy Convers Manag 2015;97:111–20.

[19] Catalano LA, De Bellis F, Amirante R, Rignanese M. An Immersed Particle Heat Exchanger for Externally Fired and Heat Recovery Gas Turbines. J Eng Gas Turbines Power 2011;133:32301.

[20] Amirante R, Distaso E, Tamburrano P. Novel , cost-effective configurations of combined power plants for small- scale cogeneration from biomass : Design of the immersed particle heat exchanger. Energy Convers Manag 2017;148:876–94. doi:10.1016/j.enconman.2017.06.047.

[21] Amirante R, Tamburrano P. High Temperature Gas-to-Gas Heat Exchanger Based on a Solid Intermediate Medium. Adv Mech Eng 2015;6:353586. doi:10.1155/2014/353586.

[22] Amirante R, Clodoveo ML, Distaso E, Ruggiero F, Tamburrano P. A tri-generation plant fuelled with olive tree pruning residues in Apulia: An energetic and economic analysis. Renew Energy 2016;89:411–21.

[23] Amirante R, Andrea Catalano L, Tamburrano P. Thrust Control of Small Turbojet Engines Using Fuzzy Logic: Design and Experimental Validation. J Eng Gas Turbines Power 2012. doi:10.1115/1.4007372.