Excerpt from the Proceedings of the 2011 COMSOL Conference in Stuttgart

Computational Fluid Dynamics Simulations of an Innovative System of Wind Power Generation

S. Carcangiu ^{*,1} A. Fanni^1 and A. Montisci ^{1}
^{1} Department of Electric and Electronic Engineering, University of Cagliari,
Piazza d'Armi, 09123 Cagliari, Italy
*Corresponding author: Piazza d'Armi, 09123 Cagliari, Italy; s.carcangiu@diee.unica.it

Abstract: In this work an innovative system for wind power generation in urban areas is proposed. The generation system substitutes the roof of the building and it consists of a static part, the stator, and a moving part, called impeller or rotor, which is a centripetal turbine with vertical axis of rotation. The rotary motion of the rotor is transferred through a shaft to a generator that produces electricity. Since the objective is to conveying the air flow in the stator in order to avoid the formation of vortices, the logarithmic spiral has been chosen as the shape for the stator blades. A 2D and a 3D model of the system are analyzed in order to evaluate the efficiency of the new system proposed considering both the influence of the profile of the building and the effect of impedance of the duct on the available power to the impeller.

Keywords: Computational Fluid Dynamics (CFD), laminar flow, stator, wind power generation.

1. Introduction

As it is well known, wind and sun are the most promising renewable energy sources, both for the concentration and availability [1]. Typically, one privileges solar energy in urban areas, because usually the roof surface is not devoted to other employments, and wind energy is preferred in extra-urban areas, because of the better ratio between produced energy and used land. Do not miss cases of wide solar energy power plants, whereas the depletion of wind energy in urban areas is almost absent. This is due to several reasons, for example a turbine with some meters of length sails is unacceptable from an urban point of view, the wind flows is extremely turbulent in the urban areas, due to the presence of buildings and furthermore re-orienting the generator on the basis of the wind direction could be troublesome [2, 3]. In spite of these problems, one could be interested to exploit wind energy in urban areas, both to integrate the energy produced by the sun, and to increase the continuity of production, so that the energy to store or to exchange with the electric network can be reduced.

The system presented in this paper aims to solve the cited drawbacks of wind generation in order to make possible its extensive use in urban areas. The generation system substitutes the roof of the building carrying out the same covering function. It consists of an external static structure, the stator, whose task is conveying the flow toward the centre of the roof, where a vertical axis centripetal turbine transforms the wind energy in mechanical energy. Such a layout of the system could entail several advantages. Firstly, the visual impact and safety problems of the turbine could be definitively avoided, because it is hidden inside the structure. Secondly, the wind direction would not be a problem, because the vertical axis of the rotor would allow it to capture the wind no matter which is its provenance direction. Thirdly, the stator would allow one to intercept a wide section of flow, even if the dimension of the turbine is very small, depending on the ratio between the inlet and the outlet cross section of the stator. This would allow one to exploit a wider range of wind velocities with respect to common wind generators, because of a lower cut-in and an unlimited cut-off velocities. Finally, the stator would allow one to control the direction of the flow that arrives to the turbine, allowing to maximize the efficiency of the turbine. In this way it could be possible to obtain good performance also in the case of turbulent flows, which are caused by other buildings or similar apparatus in the surroundings.

The present paper is organized as follows. In Section 2 a 2D FEM model of the system with the chosen mesh are given. In Section 3 the equations describing the physical phenomenon are given. In Section 4 a parametric study is performed in order to determine the best ratio between the inlet and the outlet area of the stator. In Section 5 the 3D model of the optimized structure is given, together with the description of results in terms of wind power concentration, which have been obtained in presence of light wind.

2. 2D Model Overview: Geometry and Mesh

In Fig. 1 a cross section of the system is shown. As it can be noted, the stator is a scaled-up version of the rotor. In this way, there are two or maximum three stator ducts which convey the wind, and the flux in the inlet of the rotor has a direction tangent to the blades. The spiral shape of the blades in the stator allows one to obtain a great ratio between inlet and outlet section of the duct.

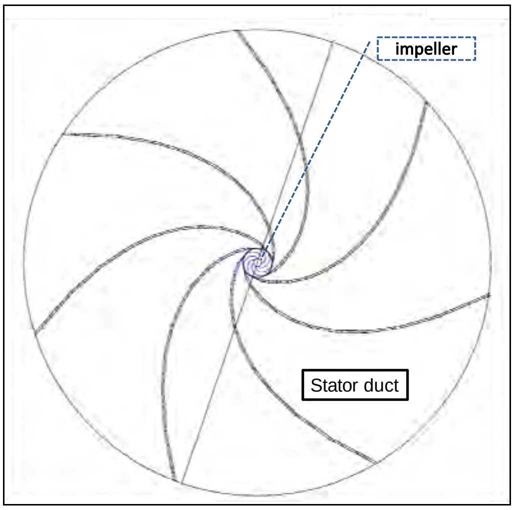

Figure 1. 2D cross section of the stator-rotor system.

Fig. 2 represents the building with the semi spherical eolic roof. In our analyses, only one stator duct is considered, together with the divergent duct where the flow downstream of the turbine is convoyed. In Fig. 3 a 2D geometry of the system is shown. In the real case, the cross section of the duct is tapered in both horizontal and vertical directions. Such narrowing in the 2D model has been modelled by a simply reduction of the height of the duct. The flow evolving inside the turbine changes its direction from radial in the inlet section to axial in the outlet one. The vertical divergent duct, downstream the turbine, allows the flow to accelerate in correspondence of the turbine itself. Finally, the pressure drop, which takes place upon the building, favors the passage of the air through the internal path instead of the external one.

The aim is making the impedance of the internal path as low as possible, so that a consistent fraction of the air flows approaching the building can be exploited to produce electric energy. To this end, a sensitivity analysis has been performed in order to optimize the shape of the path in order to evaluate how it affects the concentration of wind power at the inlet of the turbine.

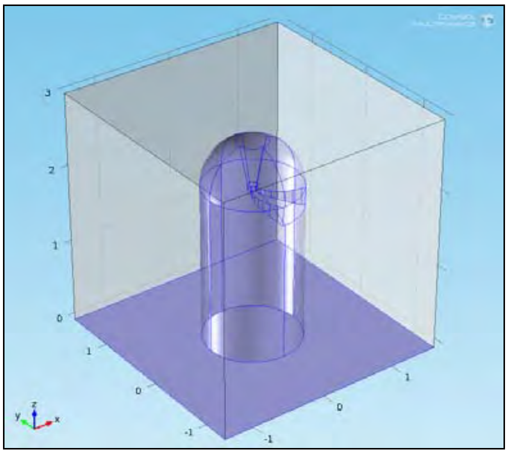

Figure 2. 3D geometry of the system.

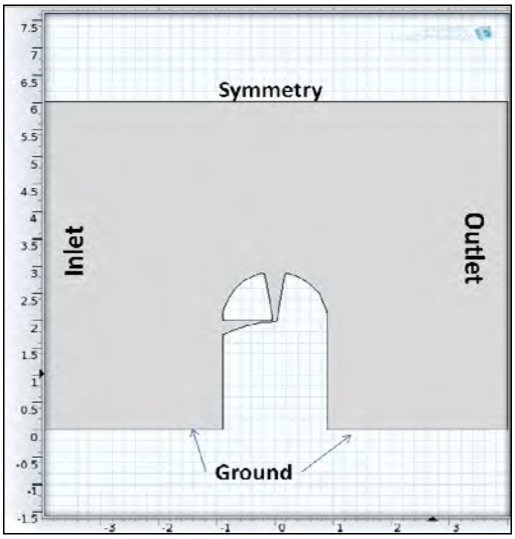

Figure 3. Geometry of the 2D model.

The flow is intrinsically 3D, so that some simplifying hypotheses have been introduced in order to study it with a 2D model. To this end, we can consider that the three-dimensionality is fundamental when we take into account the momentum transfer from the fluid to the blades of the rotor. On the other hand, when one considers only the fluid dynamic aspects, the real skew shape of the duct can be substituted with a planar one. As a further simplification, a unique static duct is considered, without considering the rotating part of the impeller.

This allows one to separate the study of the fluid dynamics of the process from the mechanics. The energy available for the process will be calculated taking into account the state of the fluid in the narrowest section of the duct and the nominal efficiency of the turbine.

In Fig. 4 a detail of the 2D model with the corresponding triangular mesh is shown. The mesh consists of 2 324 elements. As can be noted, in the region near the walls (ground and walls of the building) the mesh is composed by smaller elements since in such regions the stresses are highest.

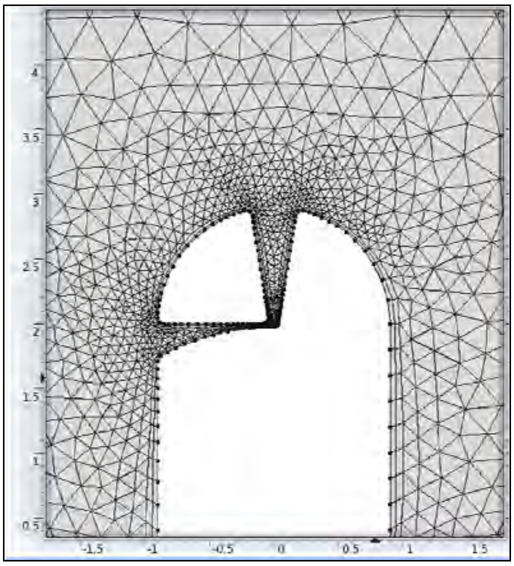

Figure 4. Mesh detail of the 2D model.

3. Governing Equations

The wind flow around and inside the building model is described by the Navier-Stokes equations [4], that have been solved for the velocity field and the pressure. Considering an incompressible flow and a stationary study, the nonlinear system of equations is the following:

$$\rho \mathbf{u} \cdot \nabla \mathbf{u} - \nabla \cdot \left(\mu \left(\nabla \mathbf{u} + (\nabla \mathbf{u})^T \right) \right) + \nabla p = 0$$
$$\nabla \cdot \mathbf{u} = \mathbf{0}$$

where:

μ denotes the dynamic viscosity, **u** the velocity vector, $\boldsymbol{\rho}$ the density of the fluid and p is the pressure.

The modeled fluid is air with viscosity 10^{-5} Ns/m^2 and density 1.2 kg/m^3. The first equation is the *momentum balance*, and the second is the *equation of continuity* for incompressible fluids.

As shown in Fig. 3, the following boundary conditions are applied in the computational domain:

- the *Inlet* boundary condition sets the velocity vector normal to the boundary and it is specified by $\mathbf{u} \cdot \mathbf{n} = 5 \ [m/s]$, that is the mean value of the wind in most of Italian territory [5];
- the *Outlet* boundary condition sets the pressure to a specific value $p_0 = 0 \ [Pa]$;
- the *No-slip* boundary condition, which eliminates all components of the velocity vector ($\mathbf{u} = 0$), is set on the ground and on the walls of the building;
- the *Symmetry* condition, applied to the upper boundary, states that the component of velocity perpendicular to the boundary is zero.

Several analyses have been performed, with different convergence rate of the duct upstream of the turbine. Among the examined solutions, the best one is chosen, which represents the most suitable solution between maximum duct narrowing and maximum available power. In fact, a great convergence rate of the duct corresponds to a high impedance, so that a few flow passes through the duct, but at the same time it allows one to employ a smaller turbine.

The available power P of an air flow crossing the section A (measured in m^2) is equal to:

$$P = \frac{1}{2} \int_A \rho \cdot v^3 dA$$

where:

ρ is the density of the air,

v is the velocity of the wind in *m/s*.

As the power is proportional to the cube of velocity, the reduction of flow due to a decrease of the minimal section can be partially compensated if velocity increases.

4. 2D Simulation results

Diagram reported in Fig. 5 shows the dependency of available wind energy from the ratio between the areas of the cross section in correspondence of the inlet of turbine and the section in the inlet of the duct. As can be seen, there is a wide range of ratio values where the slope of the curve is low, so that even if we reduce the inlet section of the turbine, the power loss is limited or negligible. This represents an advantage, because we can use smaller and then cheaper turbines to exploit the same quantity of power.

The choice of the design narrowing of the duct is taken on the basis of the required energy taking into account the efficiency of the turbine. The diagram in Fig. 5 provides, for each ratio between minimal and maximal area of the duct cross section, the value of the energy per square meter of inlet section of the stator that the system statistically produces in one year. The values of energy depend on the cube of the wind velocity, so that the diagram is deduced on the basis of the statistical distribution of the wind velocities during the year [5]. For a given energy demand per year, and an available section area at the inlet of the building, diagram in Fig. 5 allows one to calculate the narrowing of the statoric duct, then the section area at the inlet of the turbine and finally the dimension of the turbine.

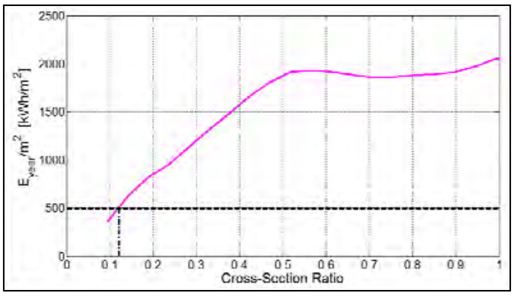

Figure 5. Energy vs. cross-sections ratio diagram.

In Fig. 6 a detail of the map of velocities is given, which corresponds to the chosen cross-sections ratio case. In this paper, the FEM analyses are performed using COMSOL Multiphysics. As can be seen, the shape of the duct allows one to have a meaningful increase of velocity in the minimal section.

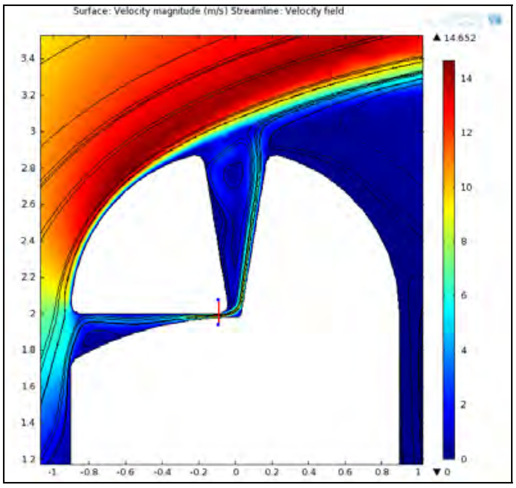

Figure 6. Velocity map of the 2D model.

From the map in Fig. 6, COMSOL Multiphysics allows one to obtain the velocities profile in the minimal section. To this end, a line is drawn in the map (see red line terminated by two blue points, near to the minimal section, in Fig. 6), and the velocity profile along this arc can be obtained (see Fig. 7). As can be noted, a parabolic velocity distribution with the fastest velocity down the center and zero velocity (no-slip condition) at the outside of the section has been obtained.

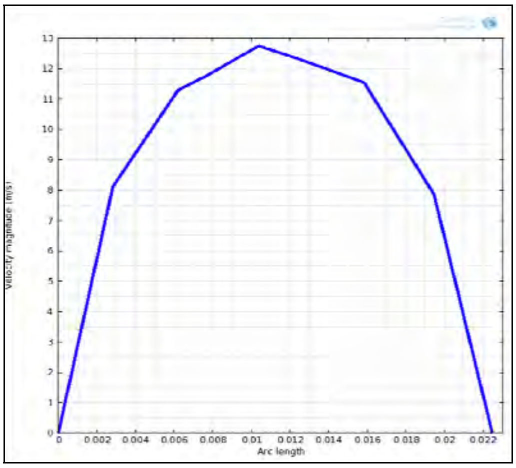

Figure 7. Velocity profile of the minimal section.

5. 3D Model and Results

A three-dimensional analysis of the model has been performed, taking into account the results of the above described two-dimensional analysis. In particular, the same ratio between sections is kept, as well as the wind velocity and the boundary conditions. In Fig. 2 and Fig. 8 the 3D model is reported, while Fig. 9 shows a detail of the mesh. The *inlet* and the *outlet* conditions are set on the opposite planes XZ, whereas the *no-slip* conditions are set in the ground and the walls of the building (blue surfaces in Fig. 2). Finally the symmetry conditions are set on the remaining boundary surfaces.

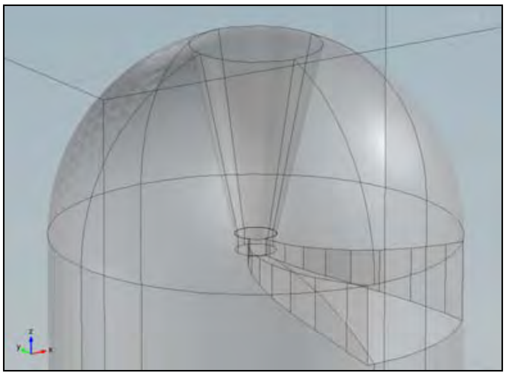

Figure 8. A detail of 3D geometry of the system.

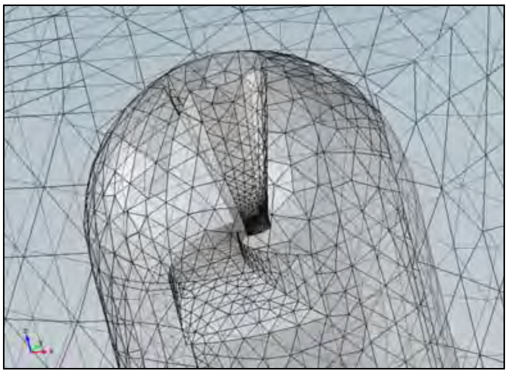

Figure 9. A detail of the mesh.

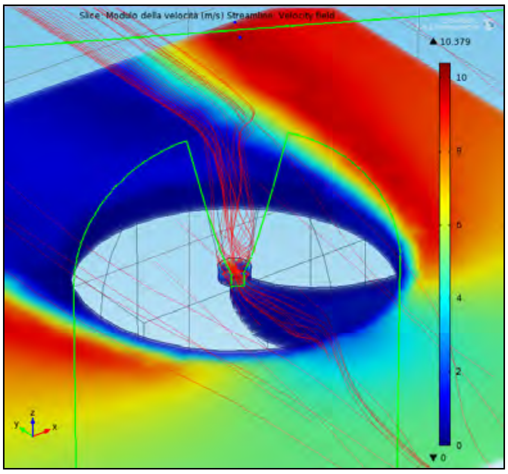

Figure 10. Velocity map and the wind streamlines of the 3D model.

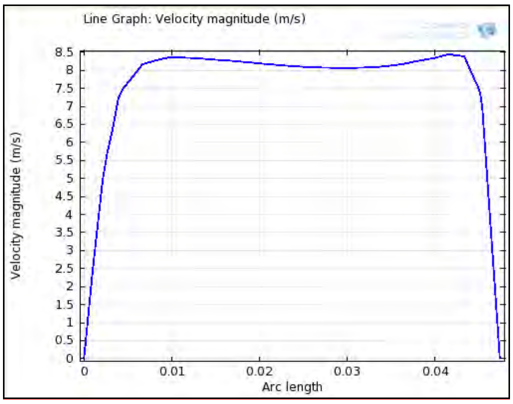

Figure 11. Velocity profile of the minimal section of the 3D duct.

The overall number of elements is 51 110, and a nonlinear solver has been used, with 91 616 degrees of freedom.

Fig. 10 shows the velocities diagram of the solution. By integrating the power along the whole minimal cross-section we obtain the value of the power available at the inlet of the turbine. In Fig. 11 the velocity profile along an intermediate line of cross-section is shown.

By integrating the power in the minimal section of the duct, we obtain the power available at the inlet of the turbine. Such value can be compared with the power of the wind crossing an equal section in a point far from the build, so that the velocity is uniformly equal to 5 m/s. The ratio between the two values of power gives us the gain of power due to the use of the statoric structure. In the analyzed problem such ratio is equal to 3.7.

6. Conclusions

In this paper, a system to exploit wind energy in urban areas is presented and studied. The proposed idea aims to avoid several obstacles to the use of wind energy in such a context, such as town planning constraints, energy density, wind direction, and so on. Firstly a 2D parametric study has been performed in order to obtain the best combination of parametric values. Such optimization gave a set of design parameters that has been evaluated also in a 3D model.

7. References

1. J. F. Manwell, J. G. McGowan, A. L. Rogers, *Wind Energy, Theory, Design and Applications*, Contract NAS2-11665, Muadyne Report 83-2-3, John Wiley and Sons (2006)
2. S. Eriksson, H. Bernhoff, M. Leijon, Evaluation of different turbine concepts for wind power, *Renewable and Sustainable Energy Reviews*, vol. 12, no. 5, pp. 1419–1434 (2008)
3. A.S. Bahaj, L. Myers, P.A.B. James, Urban energy generation: influence of micro-wind turbine output on electricity consumption in buildings, *Energy and Buildings*, 39 2, pp. 154–165 (2007)
4. R. Temam, *Navier-Stokes equations, theory and numerical analysis*, AMS- Chelsea Series, AMS, Providence (2001)
5. L. Baggini, M. Benini, G. Botta, C. Casale, C. Cavicchioli, General Assessment of exploitable Wind Resources in Italy, *Proc. of International Conference on Clean Electrical Power 2007*, pp. 605 - 612, 21-23 May 2007

8. Acknowledgements

This work is supported by the operating program of Regione Sardegna (European Social Fund 2007–2013), L.R.7/2007, “Promotion of scientific research and technological innovation in Sardinia”