Tarım Makinaları Bilimi Dergisi (Journal of Agricultural Machinery Science)
2008, 4 (2), 247 - 252

Comparative Analysis of a Wind Turbine's Performances by Means of CFD Simulations

Edmond MAICAN, Sorin-Ştefan BIRIŞ
"Politehnica" University of Bucharest, Faculty of Biotechnical Systems Engineering,
Spl. Independenţei 313, sector 6, 060042, Bucharest, Romania
e.maican@gmail.com

Abstract: It is now possible to virtually assess performances and optimize the new wind turbines designs, due to the computer systems' increased power and permanently development of the computational fluid analysis (CFD) software. There are various analysis methods, each with benefits and disadvantages. Generally, very accurate results mean a high consumption of computer resources and a significantly long period of calculus. This paper presents the results of the CFD simulation in case of a Savonius wind turbine. Two analyses are made by means of a commercial CFD software: first on a 2D model, and second on a 3D model. Even it is more pretentious from a computing resources point of view, the Shared Stress Transport theoretical model is used as it is more precise. All the results are compared with those from a real model tested in the wind tunnel. Final conclusions take into consideration the simulation time for both analysis and the closeness between the simulated and real results.
Key words: CFD, wind turbine, Savonius, static torque, Shear Stress Transport

INTRODUCTION

The permanent scientific concern over global climate change, as well as the aggressive goals for renewable power deployment in response to strong public and political support for clean energy, have fuelled in the last years a remarkable rapid development and implementation of "green" energy technologies. From a financial point of view, the aim is to reduce this kind of power generation cost in order to make it competitive on the energy market. In this respect, wind power generation systems prove to be suitable, as wind is one of the most abundant pollution free sources.

An additional way to cut costs supposes to permanently improve the existing designs, finding the most efficient way of wind-energy conversion. Researchers are now assisted by modern specialized software, which has the ability to simulate more realistically then ever the structure of the turbulent flow. Computational fluid dynamics (CFD) provides a cost-effective and accurate alternative to scale testing, offering the possibility to perform quick variations on the simulation. However, there are numerous factors with high impact on the simulations results, such as the chosen turbulence model, grid refinement, type of simulation (2D or 3D), similitude criteria used in scaled simulations etc. In the past few years, similar researches have been developed in order to relieve the results accuracy. G. Clauss and W. Heisen (2005) performed a CFD simulation in order to asses a yacht sail aerodynamics. Using the Shear Stress Transport Model (SST), they found that 2D-cross section simulation gave substantially different air flow than in case of 3D one.

C. Ferreira et al. (2007) demonstrated the unsuitability of using a single turbulence model scheme for the numerical simulation of a 2D Darrieus turbine in dynamic stall at low tip speed ratios. The resulting flow fields of the three models they used (*Laminar*, *Spalart-Allmaras* and *k-ε*) differed totally in the spatial distribution of vorticity.

W. Haupt (2005) applied CFD in architecture, trying to validate simulation results for an indoor buoyant convection. He concluded that 2D simulation often seems the only viable way to simulate huge halls or similar volumes, but the results are at least dubious. 3D simulations are more precise, but frequently with poor convergence. He also argued that near-wall treatment in commercial CFD codes still lacks a lot of accuracy improvement.

This study aims at simulating the experimental well-known work from Sandia laboratories, performed in a wind tunnel on a Savonius turbine (Blackwell et al., 1977). Results are then compared with experimental ones in order to investigate the degree of accuracy of the 2D and 3D simulations in case of this drag based wind turbine.

MODEL DESCRIPTION

The physical model

The Savonius rotor prepared for experimentations by Sandia Laboratories and used in the present simulation had the following main geometrical parameters (fig. 1):

- Number of buckets: 2;
- Rotor height *H*: 1 m;
- Bucket diameter *d*: 0.5 m;
- Gap spacing (*s*/*d*): 0.2.

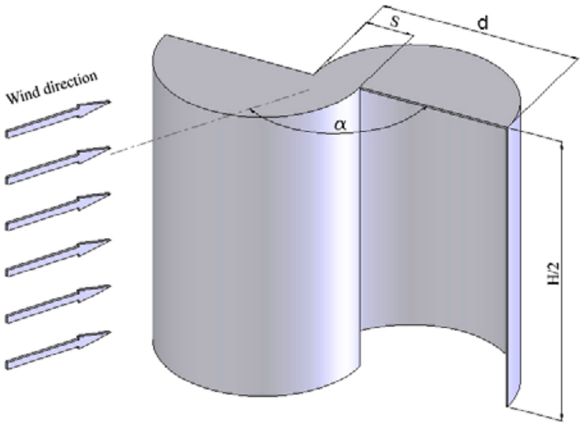


Figure 1. Geometrical parameters of the rotor

This configuration was tested at a nominal freestream velocity of 7 m/s, corresponding to nominal Re/m of $4.32 \cdot 10^5$ (for a length scale of 1 m, which is nominally the diameter of the turbine). The reason the Reynolds number was presented as a per-meter value is that there was no universally accepted length scale with which to calculate a Reynolds number for a Savonius rotor. Even there were conducted both static and dynamic tests, with different turbine configurations, only the static tests were considered for the purpose of this article. Experimentations consisted of measurements of the static torque produced when the rotor was locked at different angles ($\alpha - \pi/2$) relative to the flow (fig. 1). Then, static torque coefficients C_Q (eq. 1) were plotted as a function of rotor angular position.

$$C_q = \frac{Q}{q_\infty \cdot R \cdot A_S} \qquad (1)$$

In this equation, the torque *Q* includes both the measured torque and the tare torque, which is of the order of 0.68 Nm. The other parameters are:

R – rotor radius of rotation (R = 0.4512 m);

A_S – turbine swept area (As = 0.9023 m^2);

q_∞ – freestream dynamic pressure.

This pressure is calculated by means of the following formula:

$$q_\infty = 0.5 \cdot \rho_\infty \cdot v_\infty^2 \qquad (2)$$

where:

ρ_∞ - freestream density (ρ_∞ = 1.185 kg/m^3)

v_∞ - freestream velocity (eq. 3).

$$v_\infty = v_{\infty_u}(1+\varepsilon) \qquad (3)$$

In equation 3, $v_{\infty u}$ is the freestream velocity uncorrected for wind tunnel blockage ($v_{\infty u}$ = 7 m/s), and ε is the wind tunnel blockage factor (ε = 0.0162). So,

$$v_\infty = 7(1+0.0162) = 7.1134 \text{ m/s} \qquad (4)$$

and

$$q_\infty = 29.98 \text{ kg/(ms}^2\text{)} \qquad (5)$$

Formula (1) becomes:

$$C_q = \frac{Q}{29.98 \cdot 0.4512 \cdot 0.9023} = \frac{Q}{12.20539} \qquad (6)$$

The CFD Model

There are two CFD models: one for 2D simulations and the other for 3D simulations. Both models replicate the physical one at a scale of 1:1. There are also placed walls at the same distances like the walls of the wind turbine. However, related hardware like rotor's frame, fixing cables, stand, was not modeled as there are no data regarding the exact placement and dimensions of these components. It is also true that their influence was included in the wind tunnel blockage factor ε which was used to calculate the freestream velocity v_∞ which intervene in the static torque coefficient calculus C_Q.

A commercial CFD software was used, which comprises various viscosity turbulence models. In the RANS (Reynolds Averaged Navier-Stokes) equations, the Reynolds stress tensor is:

$$\overline{v'_i v'_j} = \frac{2}{3} k \cdot \delta_{ij} - \nu_t \left(\frac{\partial \overline{v_i}}{\partial x_i} + \frac{\partial \overline{v_j}}{\partial x_i} \right) \qquad (7)$$

In this equation, the prime denotes the variable part of the speed, and the overline is the averaged value.

The isotropic part of the tensor is $2k/3$, where k is the turbulent kinetic energy. The anisotropic part is a function of the turbulent kinetic viscosity ν_t and the mean strain rate tensor.

In the *k-ε* model, ν_t depends on the turbulent energy dissipation ε and the turbulent kinetic energy k. Even it is characterized by robustness and reasonable accuracy, when faced with non-equilibrium boundary layers this model tends to predict too late the onset of separation and to under-predict the amount of separation. This can result in an optimistic machine performance prediction.

In the *k-ω* model of Wilcox (2002) ν_t is calculated as a function of the turbulence frequency ω and turbulence kinetic energy k. This model is well behaved in the near-wall regions, where low Reynolds number corrections are not required. On the other hand, it is sensitive to the freestream values of ω.

Being one of the most effective, the Shear Stress Transport (SST) model of Menter (1994) activates the *k-ω* model in the near-wall region, and the *k-ε* model in the outer wake region and in free shear layers. Moreover, the definition of eddy viscosity is modified to account for the transport of the principal turbulent shear stress.

Because of its presumed accurate predictions of the onset and the amount of flow separation under adverse pressure gradients, the SST turbulence model was used for the present computations. The inflation process is highly recommended in simulations involving lift, drag or pressure drop in the model. By means of inflation, for the 3D model prismatic elements were generated by "inflating" triangular elements starting from the blade surface towards the fluid. As a result, the grid near the turbine's walls was formed of 20 layers of flat prismatic elements, which provide a smaller grid length scale in the direction perpendicular to the wall (fig. 2).

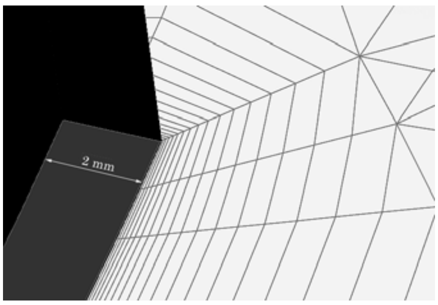
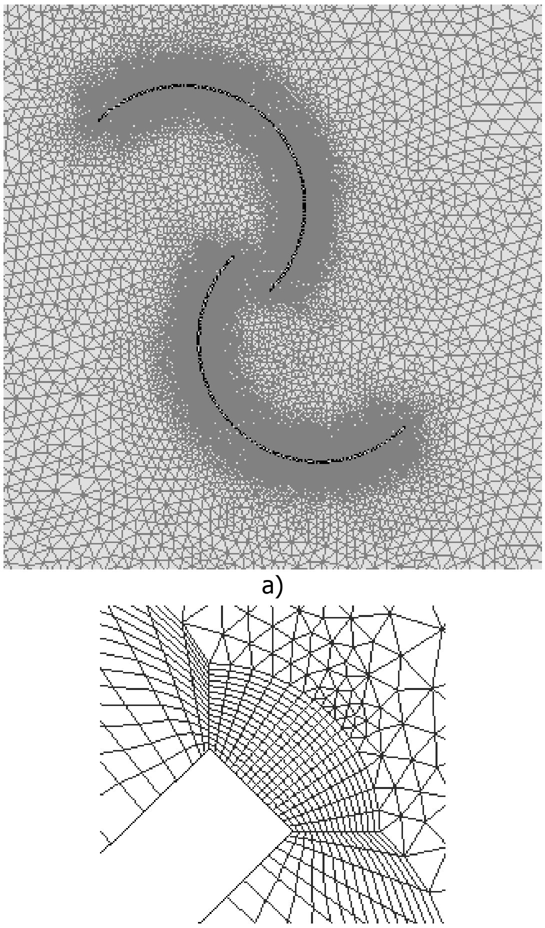


Figure 2. Grid on a plane in the vicinity of the rotor

For a good resolution of the solution it is important to have at least 10 nodes in the boundary layer. Inflation was applied with the condition to have a y+ number below 2. The dimensionless wall distance y+, which indicates the fineness of the grid near walls, is based on the distance *y* from the wall to the first node and the shear stress *τω*:

$$y^{+} = \mathrm{v}_{\tau}\frac{y}{\nu} = \sqrt{\frac{\tau_{\omega}}{\rho}} \cdot \frac{y}{\nu} \tag{8}$$

A similar approach was implemented for the 2D model (fig. 3), but with a finer length scale.



a)

b)

Figure 3. (a) 2D grid;(b) detail at the tip of the rotor

The size of the energy containing eddies is specified by means of integral length scale. In the absence of experimental reports, it is recommended to use a length scale based on the size of the object over which flow is moving. Taking into consideration other estimated values mentioned in similar problems and examples (Garg, 2002), it was assumed to be 5%

of the swept area diameter, which leads to a rounded value of 4.5 cm.

The residence time for the fluid in the domain is approximately 1 s, which was chosen as the maximum simulation timescale. The CFD solver will start with a conservative time scale that gradually increases towards the fluid residence time as the residuals decrease.

SIMULATION

The 2D simulation was performed on an Intel Core 2 CPU, with 1.66 GHz and 1.0 GB of RAM. In order to establish a very good degree of convergence of the solution, a tight target Root Mean Square (RMS) of 5e-5 was established. The CFD software calculates the RMS residual by taking all of the residuals throughout the domain, squaring them, taking the mean, and then taking the square root of the mean.

In case of Sandia experimentations, the measurements were made for 19 angles of attack ($\alpha - \pi/2$), from 10^o to 10^o, starting with -15^o and up to 175^o. The 2D simulations were made however for 15 angles of attack, from 5^o to 145^o, a sufficiently large range to estimate the conformity between the experimental results and the simulation.

Even the solver uses a robust formulation that allows accelerated convergence with relatively large timesteps, in case of 5 angles of attack (25^o, 55^o, 75^o, 115^o, 135^o, the resulting convergence behavior was "bouncy". Up to 4 simulation for each of them were made, with higher grid densities (fig. 3*a*), variations of timescales and of the RMS residuals (up to 1e-5), until a smooth numerical stability was obtained. For all of them the calculation time and number of iterations were significantly longer than in the other cases, as it can be seen in table 1.

Based on the computed torque, equation 6 was used to calculate the torque coefficient. Comparing the simulations with experimental results (fig. 4), it is obvious that the trend of the CFD values is very similar to the experimental ones. However, it can be observed a clear overestimation of the simulation on the torque coefficient.

Moreover, if the maximum value in case of experimental data corresponds to an angle of attack of 115^o, in case of CFD simulation the maximum was calculated at 125^o.

Table 1. Calculation time and number of iterations for 2D simulations

Angle (deg)	Cq CFD	Simulation time (s)	No. of iterations
5	0.30924	163	49
15	0.26531	178	52
25	0.23916	450	47
35	0.24512	189	60
45	0.28492	173	52
55	0.32919	1973	101
65	0.31896	173	52
75	0.42124	3627	198
85	0.61545	288	89
95	0.63829	215	64
105	0.82571	169	53
115	0.94473	1704	192
125	1.05630	189	59
135	0.87449	2985	171
145	0.79212	260	81

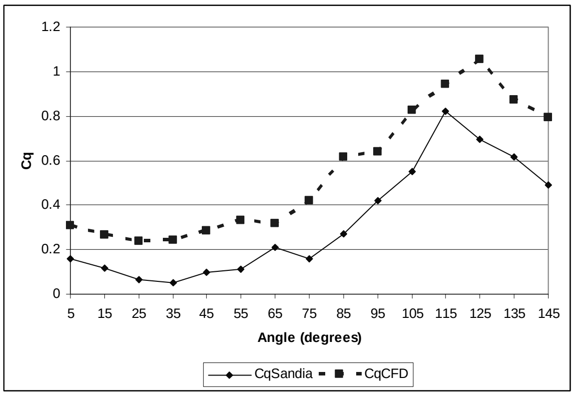


Figure 4. Experimental vs. simulation results

In order to find the conformity between simulations and experimental tests, polynomial regression is used. Figure 5 shows the polynomial functions that properly fits the two sets of data points.

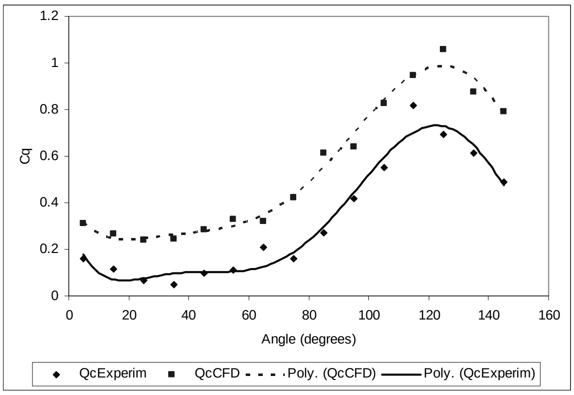


Figure 5. Polynomial regression of data sets

Both functions are polynomials of 6-th degree, for a better R-squared value:

$$f_{CFD}(x) = 5e\text{-}12x^6 - 2e\text{-}09x^5 + 4e\text{-}07x^4 - 3e\text{-}05x^3 + 0.0013x^2 - 0.0245x + 0.41 \quad (9)$$

$$f_{exp}(x) = 8e\text{-}12x^6 - 4e\text{-}09x^5 + 7E\text{-}07x^4 - 6e\text{-}05x^3 + 0.0022x^2 - 0.0403x + 0.328 \quad (10)$$

where $f_{CFD}(x)$ corresponds to the simulation results and $f_{exp}(x)$ – to the experimental data.

The integral of each of the above functions represent the work done by the turbine if it would rotate from 5 to 145 degrees. This work is defined by the area between the curve and x-axis:

$$I_{\exp} = \int_{5}^{145} f_{\exp}(x)dx = 45.028 \quad (11)$$

$$I_{CFD} = \int_{5}^{145} f_{CFD}(x)dx = 76.2 \quad (12)$$

So, the 2D CFD simulation predicted a mechanical work which is 69% greater than the real case:

$$\frac{I_{CFD} - I_{\exp}}{I_{\exp}} \cdot 100 = 69.2\% \quad (12)$$

Taking advantage of the parallel processing capability of the software, the 3D simulations were carried out on a network of 8 computers with AMD Athlon XP 1700+ processor and 512 MB of RAM each. Due to the long computing time for each simulation (more than 9 hours), only 3 analysis were made: for 25°, 45° and 75°. Table 2 and figure 6 present the torque coefficients both for experimental and simulation results. Table 2 also includes the deviation of the CFD results from the experimental ones.

Table 2. Torque coefficients for experimental tests and 3D simulations

Angle (deg)	Cq exp.	Cq CFD	Deviation (%)
25	0.63829	0.07265	11.7642
45	0.82571	0.10924	9.2398
75	0.94473	0.19141	19.6302

As one can notice, this time the results are substantially closer to the experimental ones than in case of 2D simulations. However, due to large amounts of necessary computation resources, the grid used in spatial simulations was almost 3.5 times coarser than in case of 2D analysis. Inflation was used for wall treatment, so layers of prismatic elements were generated close to the walls, allowing much larger aspect ratios. This way, the created mesh was finely resolved normal to the wall, but coarser parallel to it. The remaining areas of the domain were formed of tetrahedral elements. At this time, there was also no time to make supplementary simulations in order to test and validate the grid size.

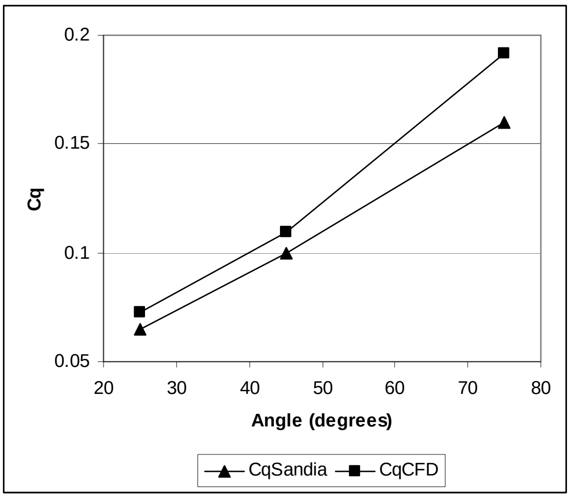


Figure 6. Experimental vs. 2D simulation results

It is important to mention that, for the 3D model, the endplates at the top and bottom sides of the model rotor were also modeled, which was not possible for the 2D simulation. Each endplate drastically modifies the flow aspect, introducing vertical recirculation regions inside and outside of the rotor. Figure 7 presents these regions on a vertical plane that passes through the right side of the turbine.

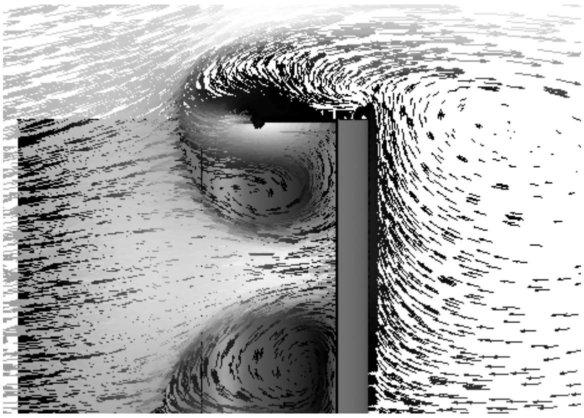


Figure 7. Vertical recirculation regions induced by the top endplate

CONCLUSIONS

The Savonius type wind turbines are drag-based devices, which mean they rotate due to the difference between the velocity of the air impinging on the blade and the velocity immediately downwind of the blade.

The air-flow around the blades is turbulent, so inflation is necessary during the meshing process for boundary layer resolution.

The full scale 2D simulation has proved to be very useful in the assessment of the torque coefficient trend, but it was observed a clear overestimation on its values. Such results can be explained by the development of vortex-like structures in real flow that cannot be modeled based on to a 2D geometry. Especially in turbulent flows, 2D simulations would lead, in the best case, to divergent behavior of the software and, in the worst case, one would get impressive results that are deeply wrong and misleading. Therefore, the necessity for true 3D simulations is obvious.

However, bidimensional simulations can be used as preliminary studies, most of all due to the highly reduced amount of computing time compared to the 3D ones.

REFERENCES

Blackwell, B.F., R.E. Sheldahl, L.V., Feltz, 1977. Wind Tunnel Performance Data for Two- and Three-Bucket Savonius Rotors. Sandia Laboratories, Albuquerque, New Mexico, 87115.

Clauss, G., W., Heisen, 2005. CFD Analysis on the Flying Shape of Modern Yacht Sails. 12-th International Congress of the International Maritime Association of the Mediterranean.

Ferreira, C.S., G., Bussel, G., Kuik, 2007. 2D CFD simulation of dynamic stall on a vertical axis wind turbine: verification and validation with PIV measurements. In: 45th AIAA Aerospace Sciences Meeting and Exhibit. Reno, Nevada, 8-11 January 2007.

Garg, V.K., 2002. Low-Pressure Turbine Separation Control – Comparison with Experimental Data. NASA/CR-2002-211689.

Haupt, W., 2005. Do's and Don'ts of CFD-Simulations on Free Convection. In: 7th Symposium on Building Physics in the Nordic Countries. Reykjavik, Iceland, June 13-15.

Menter, F.R., 1994. Two-Equation Eddy-Viscosity Turbulence models for Engineering Applications. AIAA J., 26: 1299-1310.

Wilcox, D.C., 2002. Turbulence Modeling for CFD. DCW Industries, 2nd Edn. ISBN-10: 192872910X. ISBN-13: 9781928729105.