Numerical study of a small H rotor type wind turbine
Diogo Borges Alves Marçal

Abstract
Wind power has been one of the most promising renewable energies and it also presents a better cost-benefit ratio in comparison with most of the other forms of renewable energy. The development of horizontal-axis wind turbines is well known, unlike vertical-axis wind turbines which are still in a developing stage. This is due to the lower efficiency and larger vibrations. The purpose of this paper is to analyze and compare two airfoil sections in two-dimensional and three-dimensional flow conditions.

Keywords: vertical axis wind turbine, unsteady numerical simulation, numerical optimization of blade sections

1. Introduction

Fossil fuels are finite and emit pollutants to the atmosphere when consumed, typically, in the combustion process. This concerns the general public, and the scientific community in particular, to find a sustainable way to get energy. The result of this quest leads to the usage of renewable energy. Renewable energies are important because they are clean and limitless. This means that we can draw power continuously without having polluted the environment.

Lately, the energy production decentralization has been advocated as a viable source for exploitation of electric power supply. This allows to have a lower cost distribution network, requiring less heavier infrastructure compared to the current system that has to accommodate much more power in the grid extensions and more balanced feed of energy in terms of signal, helping to solve the problems arising from:

- power grid fickle and poor quality in remote areas and far from the generation plants,
- the increasing number of electronic devices and their fragility toward grid fickle.

There is also another significant benefit, if one source becomes inoperable, in theory, the whole system remains operational. Nevertheless centralized production will always be need due to contractual requirement of balanced and continuous supply of electricity [1]. Thus, small wind turbines are an increasingly viable option, not only because of its small size for its power but also for its easy installation. A vertical axis wind turbine is very easy to be installed in urban areas, due to their independence of wind direction. In this sense and in order to make better use of wind energy, wind turbines have been the subject of intensive study.

The objective of this paper was to assess the performance of a new optimized airfoil [2], in comparison with the standard NACA0015, for small vertical wind turbines. Firstly, unsteady numerical simulations were performed in two-dimensional flow (infinite span). Subsequently, in order to be able to assess the influence of three dimensional effects in the flow, we performed a study in three dimensions flow (finite span).

The given airfoil optimization took into account various degrees of freedom to exploit a number of parameters in the airfoil geometry. The Differential Evolution method coupled with a two-dimensional unsteady Reynolds Average Navier-Stokes solver, FLUENT 6.3 [3], was used for the airfoil optimization. The objective function was the turbine power output.

The GAMBIT and Vmesh2D [4] codes were used for the generation of geometry and domain discretization. The Vmesh2D code was used to generate the mesh of the optimized airfoil described in [2].
2. **Vertical-axis wind turbine**

A key parameter for the study of fluids dynamic is the Reynolds number. This is defined as:

\[ \text{Re} = \frac{\rho U_{\text{ref}} c}{\mu} \]

where \( \rho \) is the fluid density in kg/m\(^3\), \( U_{\text{ref}} \) the upstream flow velocity in m/s, \( c \) is the blade chord in m and \( \mu \) the dynamic viscosity of the fluid in kg/m s. The fluid that was considered for the study is air.

The wind power available is given by the kinetic energy of the wind for a given area per unit time:

\[ P_{\text{wind}} = \frac{1}{2} \rho A U_\infty^3 \]

being \( P_{\text{wind}} \) the wind power in Watts [W], \( A \) the area swept by the blades projected perpendicularly to the flow in m\(^2\). Note the cubic dependency to the wind speed [5]. The efficiency of a wind turbine is usually characterized by power coefficient \( C_p \):

\[ C_p = \frac{P_{\text{turbine}}}{P_{\text{wind}}} \]

\( P_{\text{turbine}} \) is the power of the turbine. This is, the power coefficient is the fraction of wind power that is extracted by the turbine [5]. Another characteristic value of a turbine is the peripheral velocity or Tip Speed Ratio (TSR), i.e., the ratio between the speed of the blade tip of the turbine and the wind speed:

\[ \text{TSR} = \frac{\omega R}{U_\infty} \]

where \( \omega \) is the angular velocity of the turbine in rad/s and \( R \) is the radius of the turbine in m [5].

In case of a straight blade vertical-axis wind turbine (VAWT) it's equal throw out the entire blade.

In a VAWT, the flow velocity in the blade is variable in magnitude and angle of attack, fig. 1.

![Figure 1 – Typical speed Triangles of a VAWT along the full rotation [6]. Red line: relative velocity or flow approach Paddle; green line: absolute speed or wind and yellow line: rotation speed of the paddle.](image)
3. Optimized airfoil

An optimization study is a problem that seeks to minimize or maximize a function by systematically choosing the values of variables within a pre-defined set. The optimization study [2] was done according to the project TSR and Reynolds number of 3 and $2.9 \times 10^6$, respectively. Since the function in question is not a linear function and not differentiable it is not possible to apply a set of equations to achieve a result or find its maximum. So an algorithm was needed that was capable of doing the optimization iteratively: the method of Differential Evolution. This is accepted for situations where it is necessary a robust code of simple use [7-9], as in the present case.

The airfoil given in [2] was based on two Bezier curves with eight variables. The first curve defines the thickness of the airfoil, and uses eight control points and only six are variable vertically, the leading edge and trailing edge are fixed. The second curve defines the curvature of the camber line, and uses three control points, two fixed at the trailing edge and leading edge and one variable horizontal and vertical. These variables are bound to enable control over the shape of the blade. The comparison can be found in fig. 2.

The optimized airfoil has a slight curvature. This curvature has the opposite curvature of the line described by the blade during the operation of the turbine. The thickness of the optimized airfoil is larger than for the NACA 0015 airfoil, and that the largest difference is found at the lower surface. In the upper surface the difference is only noticeable near the trailing edge.

4. Numerical simulation

In order to simulate the turbine in rotation it is necessary to make the mesh rotate, but as it's an unconfined flow it's also necessary to have the wind speed in a constant direction. To achieve this, the domain was separated into two zones, one outer without rotation and one inner with rotation, fig. 3.
The inner zone was subdivided into two parts for domain discretization reasons. In order to simulate the unconfined flow outside the outer boundary line of the area is divided in half where the left is inlet flow zone and the right represents the outlet flow zone. The programs chosen for the geometry and mesh generation were the GAMBIT and Vmesh2D. The second was released by the authors of work [2], and used in it. For the numerical simulations was used the program FLUENT 6.3, with the coupled model. The chosen spatial discretization scheme was the second order for 2D and first order for 3D. In 2D it was used second order scheme for the pressure, and was used Quick scheme for the momentum, \( k \) (turbulent kinetic energy) and \( \omega \) (specific dissipation ratio). In 3D the first order method was used for the standard pressure, momentum, \( k \) and \( \omega \). The turbulence model used was the \( k-\omega \) SST (shear stress transport), with modification for the transitional flows zones. Due to the turbulence model used, the first element along a wall should have a value of \( y^+ \leq 1 \) to obtain consistent results, [10, 11]. The \( y^+ \) is the dimensionless distance from the wall and defined as:

\[
y^+ = \frac{u_t y}{v},
\]

where \( u_t \) is the friction velocity, \( y \) is the distance to the wall and \( v \) is the kinematic viscosity of the fluid.

### 4.1.1. Mesh 1

![Mesh 1](image)

Figure 4 – Mesh 1

![Blade zone of mesh 1](image)

Figure 5 – Blade zone of mesh 1
This mesh has a blade discretization of 71 points for each side, the trailing edge zone with 21 points, fig. 5, the boundary layer zone with 25 points, fig. 6, the domain boundary zone 100 points, and the blades zone with 300 points on the outer edge and 200 points in the interior edge, Fig. 4.

4.1.2. Mesh 2

This mesh was created because of the necessity to build a tree-dimensional model. The blade discretization is 53 points for each side, the trailing edge zone with 18 points, the boundary layer zone with 15 points, fig.8, the domain boundary zone with 36 points, and the blades zone with 75 points on the outer and interior edge, Fig. 7.
4.1.3. Mesh 3D

The 3D mesh was obtained by extruding the mesh 2. The discretization consists in 30 points for the volume above and 30 points for the volume below the top of the blade, figs. 9 and 10, resulting in enough discretization for the edge effects to be accounted for.

![Figure 9 – Mesh 3D](image)

![Figure 10 – Mesh 3D, detailed near the top of the blades](image)

4.2. 2D results

The comparison study was made for wind speeds, $u_{\infty}$, values between 6 and 18 m/s or TSR between 7 and 2.3 respectively, considering a constant rotational speed of 400 rpm. This option was made by keeping the Reynolds number with only minor changes because the rotational speed dominates the approach speed to the blade.

![Figure 11 – $C_p$ vs. TSR – Mesh 1 and mesh 2 with both airfoils](image)
There is a difference in results between the two meshes although the trends are identical. We see that there is a decrease in $c_p$ values for the mesh 2 and they have much lower values for higher TSRs. Both airfoils show the same maximum $c_p$ at the same TSR optimum 3.5 but the optimized airfoil on the mesh 2 having a value of 3.2. For higher values of TSR the $c_p$ values are slightly lower for the optimized airfoil and immediately after the optimum TSR higher values are obtained, see fig. 11.

![Power vs. Wind Speed](image1.png)

Figure 12 – $P_{turbine}$ vs. $U_\infty$ – Mesh 1 and mesh 2 with both airfoils.

The trend of the $P_{turbine}$ vs. $U_\infty$ curve is the same for both airfoils up to near the maximum power wind speed of the NACA 0015 airfoil and different later. The optimized airfoil has 30% more maximum power than the NACA 0015 airfoil and it is reached at an higher $U_\infty$ (lower TSR), fig. 12. Both meshes exhibit very similar results for both airfoils, with only a small gap between them and mesh 2 seems to be slightly to the right, fig. 12. The difference between them until near maximum $P_{turbine}$ for the NACA 0015 airfoil averages 50W. The maximum power of the optimized airfoil varies between meshes but not to the NACA 0015 airfoil.

![Torque vs. Angle](image2.png)

Figure 13 – Torque comparison between airfoils throughout a full turbine rotation at optimized airfoil project TSR, 3.
In fig. 13 it can be seen that the airfoil curvature of the optimized airfoil results in a variation for zero lift near the angular position 90°. This change represents a larger lift between 90° and 180°. This result, coupled with increased thickness and a higher incidence angle to the flow between 90° and 180°, represents a significant increase in torque produced by blade. Between 0° and 90° the curvature has the opposite effect, but due to the increase in thickness the power loss is not as significant as the increase experienced between 90° and 180°. Between 180° and 360° torque is almost the same.

4.3. 3D results

The comparison study was made with $U_\infty$ values of 12, 14 and 16 m/s (TSR values of 3.5, 3 and 2.6 respectively) in each case, and with $U_\infty$ 10 and 18 m/s (TSR values of 4.2 and 2.3 respectively) for the NACA 0015 airfoil, with constant rotational speed of 400 rpm for the same reasons as before. Because of the higher computational times and adjustments needed for this model, it was only possible to obtain results for 5 points for the NACA 0015 airfoil and 3 points for the optimized airfoil.

![Figure 14 – $C_p$ vs. TSR – Mesh 2 and mesh 3D with both airfoils](image1)

![Figure 15 – $P_{turbine}$ vs. $U_\infty$ – Mesh 2 and mesh 3D with both airfoils](image2)
It can be observed, in Fig. 14, that the NACA 0015 airfoil maintains the optimum point of TSR for 2D and 3D flow conditions. Due to the small amount of computed points, for 3D flow, we can only conclude for the optimized airfoil that power decrease follows the NACA 0015 trends. This difference in both airfoils is on average 50%, fig. 15.

4.4. Computation time

In Table 1 we find the meshes number of elements and the computation time required for each mesh.

<table>
<thead>
<tr>
<th>Zone</th>
<th>Mesh 1</th>
<th>Mesh 2</th>
<th>Mesh 3D</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>NACA</td>
<td>OPT</td>
<td>NACA</td>
</tr>
<tr>
<td>Farfield</td>
<td>12.200</td>
<td>12.200</td>
<td>15.693</td>
</tr>
<tr>
<td>Exterior of blades</td>
<td>17.949</td>
<td>18.555</td>
<td>6.053</td>
</tr>
<tr>
<td>Core</td>
<td>6.170</td>
<td>6.177</td>
<td>1.057</td>
</tr>
<tr>
<td>Blade layer</td>
<td>3.360</td>
<td>3.360</td>
<td>1.456</td>
</tr>
<tr>
<td>Trailing edge</td>
<td>480</td>
<td>480</td>
<td>238</td>
</tr>
<tr>
<td>Blade tip interior</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>TOTAL</td>
<td>47.839</td>
<td>48.452</td>
<td>27.885</td>
</tr>
<tr>
<td>Computation time (4 rotations)</td>
<td>~4 hours</td>
<td>~2 hours</td>
<td>~120 hours (5 days)</td>
</tr>
</tbody>
</table>

The calculation time is proportional to the meshes number of elements, the same happened in terms of used memory for each calculation. The difference between the meshes 1 and 2 is noticeable. The mesh 1 has 60 times less elements than the mesh 3D and mesh 2 has 30 times less than the mesh 3D.

5. Conclusions

The numerical results presented for optimized airfoil has shown a power output increase in comparison with the standard NACA 0015. The major difference between the optimized airfoil and the NACA 0015 is the increase of the maximum power output as function of the wind velocity. The increase in power output is about 30%. This also clearly seen in the torque output, as function of the angular position. In 3D simulation there was great difficulty in setting the relaxation coefficients. As expected, the obtained results demonstrate an average of 50% decrease of the power output for the 3D calculations, when comparing to a 2D calculation. Another important conclusion, the three-dimensional simulation is almost unfeasible using the serial version (one processor) of the FLUENT solver. The memory limitations also forced a lower quality discretization of the domain.

We also conclude that further research is need to design better airfoil sections that meet the requirements of the VAWTs. This kind of turbines offer great advantages over HAWT, especially in areas with highly variable winds such as urban environments. Regarding the current development, this work has shown that remarkable differences for the power coefficient can be obtained by using a 2D optimization of the airfoil section. Thus making VAWTs an alternative solution for small installations and of great ecological value and financial.
Bibliography


