Numerical investigation of a reduced scale Lenz wind turbine model for aerodynamic tunnel applications

I O Bucur¹, I Malael¹ and D Preda²

¹Romanian Research and Development Institute for Gas Turbines - COMOTI, Iuliu Maniu Ave, No. 220D, 061126, Bucharest, Romania
²S. C. ROLIX IMPEX SERIES S. R. L., Basarabiei Ave, No. 256, Bucharest, Romania

Corresponding author’s e-mail address: ioana.bucur@comoti.ro

Abstract. Renewable energy sources represent efficient and reliable energy solutions for the modern world, as they are eco-friendly alternatives to fossil fuels or nuclear power plants. The technologies available nowadays allow researchers to perform in-depth computational fluid dynamics analysis for systems that can generate green energy. The wind energy industry developed considerably as classic wind turbine models (horizontal axis wind turbines and vertical axis wind turbines) are constantly optimized and new configurations are studied in order to assess better performances. This paper presents the numerical investigation campaign of a reduced scale Lenz wind turbine model. The Lenz model has three blades that are attached directly on a vertical shaft. For the numerical simulations of the model, the ANSYS Fluent software is employed. For the evaluation of its self-starting behaviour the six degree of freedom method was employed and the configuration was studied for different moments of inertia. Furthermore, the chosen range of inlet velocities allowed the investigation of the influence of high Reynolds numbers on the proposed Lenz model and the vorticity magnitude contours were computed for different azimuth angles. Future work includes the validation of the numerical results with experimental data obtained during a wind tunnel testing campaign.

1. Introduction

Renewable sources of energy can be developed up to a point where they can significantly reduce the dependency on nuclear energy, fossil fuels or other environmentally unfriendly alternatives. Wind turbines are systems that convert the kinetic energy of the wind into electrical or mechanical energy. These systems have specific configurations, depending on their axis of rotation, in accordance with the area they are installed in and wind regimes, being divided into two large categories: horizontal axis wind turbines (HAWTs) and vertical axis wind turbines (VAWTs), which are analyzed comparatively in [1]. HAWTs are unsuited for low wind speed regions, reaching their potential in isolated areas with high wind velocities, in these conditions being able to generate higher amounts of energy compared to VAWTs [2]. However, VAWTs can harvest energy from any direction, being omnidirectional structures and can also be installed in urban areas, due to their smaller configuration and to the fact they generate lower levels of noise when compared to HAWTs [3]. Most common configurations of VAWTs include Savonius, Darrieus and H-Darrieus wind turbines and derived from these, new geometries have been developed, such as Crossflex wind turbine, combined Savonius-Darrieus rotor, Zephyr turbine or Lenz [4]. The advantages of a Lenz type VAWT include low cost fabrication, as its
design is not very complicated, reliability, improved starting behaviour and good performances for low tip speed ratios [5]. A mathematical model for calculation activities that present as an outcome the geometric parameters for a Lenz turbine is described in paper [6]. For this configuration, the maximum value of the power coefficient, thus, the best efficiency is encountered at values of the tip speed ratio below 1 [7].

Computational fluid dynamics (CFD) evolved considerably with technological improvement of computers and their performances. CFD allows researchers to analyze various phenomena using only a software, thus many industries benefit from it, as the developed prototypes and systems can be firstly studied on a computer and optimized accordingly, then manufactured and tested. This approach is widely spread, for applications regarding the automobile industry, aerospace industry, renewable energies etc [8]. According to the resources available (computer memory, capacity and performances) CFD numerical investigations can be 2D or 3D. In paper [9], the authors compare experimental data from a three-bladed H-Darrieus wind turbine with 2D and 3D numerical results. For a wind speed above 5.81 m/s and corresponding Reynolds numbers, the 3D simulation underestimates the real peak measured in the wind tunnel, but overestimates the lowest real power coefficient value. For wind turbines, 2D numerical simulations were proven to suffice and to approximate with an acceptable error the performances of such systems. In order to properly determine the transient flow field around a wind turbine using 2D CFD numerical simulation, one should carefully select the model, the methods, the schemes and when required the Courant number. Furthermore, the calculation grid must be finer around the blades and the transition from the rotor fluid to the stator fluid should be made smoothly through an interface. Keeping these aspects in mind, in paper [10] the authors compared experimental data from a VAWT with 2D numerical simulations and successfully approximated the peak in power coefficient using two different models: \( k-\omega \) SST and \( \gamma-Re \). In paper [11] numerical results are also compared with experimental data. The models used for the numerical investigation are \( k-\omega \) SST and \( k-\varepsilon \) RNG and the authors concluded that the \( k-\omega \) SST better approximated the power curve. These are just a few examples from the vast state of the art regarding 2D numerical simulations of VAWTs and there are many other examples, such as the studies conducted by Castelli et al. [12] and Ferreira et al. [13]. One of the main advantages of 2D numerical simulations is the fact that they require lower computational performances than the 3D ones, while giving reliable information regarding the flow field. On the other hand, because they do not take into consideration any of the 3D effects that may occur in the real functioning of the wind turbines, if the mesh is coarse and the set-up is poor it might generate errors. By using the six degree of freedom method (6DOF) for a 2D numerical investigation, the effects of the mass and moment of inertia are taken into account, thus improving the results [14].

In this paper a Lenz vertical axis wind turbine was designed employing the SolidWorks CAD software. The model was then numerically investigated using CFD methods by employing the ANSYS Fluent software in order to study its starting behaviour and efficiency. The methodology for the proposed investigation consists of applying the six degree of freedom method, that uses the blades’ moment of inertia for their angular motion computing. This method allows the study of the self-starting behaviour of a wind turbine for different wind speeds and different moments of inertia, respectively different values for the turbine’s mass [15]. Furthermore, numerical results generated by this method were proven to be reasonable when compared to experimental results [16]. The starting behaviour for the proposed geometry in this paper was studied for different values of the turbine’s moment of inertia, respectively mass, at wind speeds of 12 m/s and 14 m/s.

2. Geometry and methodology

In order to generate the 3D model of the Lenz wind turbine the SolidWorks software was used. The geometry of the blade together with its dimensions are given in figure 1 and are expressed in millimeters. The studied model has three blades, attached directly to the main shaft, with a blade length of 0.5 m and a rotor radius of 0.2 m, as illustrated in figure 2.
For the numerical investigation a 2D mesh was generated using ANSYS Meshing, with the quadrilateral method. The domain was divided into a rotor domain, that included the blades and the shaft, and a stator rectangular domain. The total number of elements was 103 866. On the left-edge of the rectangular domain the velocity inlet was defined, whereas on the right-edge the pressure outlet was set. The mesh is illustrated in figure 3.

Figure 1. Blade dimensions for the Lenz turbine.

Figure 2. Lenz turbine geometry: a) top view, b) isometric view.

Figure 3. Mesh for the numerical investigation: 103 866 elements and 105 407 nodes.
In the following figures the mesh around the blades and the mesh for the rotor domain is presented, and the rotor-stator interface is visible. The value for $y^+$ was 1.

![Figure 4. Mesh close-up: rotor domain.](image)

![Figure 5. Mesh close-up: blade.](image)

For the study of the turbine’s mass influence on the starting behaviour, the baseline and 3 other cases with different mass were considered at two different wind speeds, as presented in table 1. The mass values for the other cases represent 50%, 75%, respectively 125% from the baseline model’s mass.

| Table 1. Studied cases. |
|-------------------------|
| **Mass [kg]** | **Moment of inertia [kgm$^2$]** |
| Baseline       | 3          | 0.12          |
| Case 1 - 50%   | 1.5        | 0.06          |
| Case 2 - 75%   | 2.25       | 0.09          |
| Case 3 - 125%  | 3.75       | 0.15          |

In order to determine the angular velocity of the model and the period of time in which the turbine reached its equilibrium state for a given mass, thus a constant value for the angular velocity in time, the Dynamic Mesh feature in Fluent was employed, using the 6DOF method, without any initial turbulence applied during the numerical analysis.

For the simulations the used model was $k$-$\omega$ SST with 2 equations, that combines two previously defined turbulence models, namely Wilcox’s $k$-$\omega$ and $k$-$\varepsilon$. The employed turbulence model uses the original $k$-$\omega$ model for the inner region of the boundary layer, whereas for the analysis of the outer region switches to the standard $k$-$\varepsilon$ model, as explained in [17]. The calculation method was SIMPLE, using 2nd discretization order schemes. The time step was set at a value of 0.0005 s [18] and the total number of iterations varied with the inlet velocity, as for a greater wind speed value the turbine accelerated more rapidly, thus less iterations were necessary. However, the maximum number of iterations per time step was set for both velocities at 50 iterations per time step. The cases presented in table 1 were firstly set at an initial inlet velocity of 12 m/s, then at an inlet velocity of 14 m/s. The blades were set as rigid bodies under the Dynamic Mesh option, the rotor fluid was set as a passive rigid body and the stator fluid was set as stationary.
The purpose of this investigation was to see the mass influence on the starting behaviour of the studied model, as well as to acknowledge it’s efficiency for high Reynolds numbers. In the following section the results are summarized.

3. Results

3.1. Velocity inlet V=12 m/s
Firstly, the four cases presented in Table 1 were investigated for a velocity inlet of 12 m/s. The evaluation of the starting behaviour was possible by applying the 6DOF method, that allowed the ascertainment of the evolution of the turbine’s angular speed in time. Angular velocity variation in time was plotted for all four cases in figure 6.

![Figure 6. Angular velocity variation in time - V=12 m/s.](image)

As can be observed in figure 6, the necessary time for the wind turbine to achieve its equilibrium state and reach a constant angular velocity becomes larger with the growth in mass and moment of inertia. The starting time dependency with mass can be characterized as directly proportional. For each case the turbine reached an equilibrium state at a constant angular velocity value of approximately 52 rad/s and a tip speed ratio value of 0.86. The estimated starting time for each case is given in figure 7.

The starting time given above represents the amount of time necessary for the Lenz wind turbine to stabilize itself at a constant value for the angular velocity. It can be observed in figure 4 that the mass, thus the moment of inertia, is directly proportional to the starting time, as stated above. The first and second case accelerate with 51.35%, respectively 24.32% faster than the baseline, whereas the third case needs a time with 20.94% larger than the baseline in order to reach a constant angular velocity. The power coefficient value at a wind speed of 12 m/s and an angular velocity of 52 rad/s was approximately 0.038. The first case reached a constant angular velocity after approximately 7.2 s, the second case reached it after 11.2 s, the third case after 17.9 s, whereas the baseline reached a constant value for the angular velocity after 14.8 s.

3.2. Velocity inlet V=14 m/s
The same investigation was done for an inlet velocity of 14 m/s. The results are plotted in figure 8, that illustrates the angular velocity variation in time for the four cases presented in Table 1. The estimated starting time for each case is represented in figure 9.
Figure 8. Angular velocity variation in time - $V=14\ m/s$.

Figure 9. Starting time for each case - $V=14\ m/s$.

For this inlet velocity the equilibrium angular velocity was 63 rad/s, at a tip speed ratio value of 0.9. The power coefficient at a wind velocity of 14 m/s for the studied geometry was 0.003. The dramatic decrease in power coefficient was predictable, as studies from literature on the matter provide experimental results regarding small scale Lenz models that conclude that the efficiency of this turbine decreases abruptly with the increase of the tip speed ratio to approximately 1 or above. The peak in efficiency for such a configuration occurs often at tip speed ratios below 1 [7]. In order to improve the overall efficiency flow control methods can be applied [19].

For this inlet velocity, the first and second studied cases, with lower masses and moments of inertia, managed to reach the equilibrium state with 49.18% faster in the first case compared to the baseline and with 26.22% more rapidly in the second case in comparison with the baseline. Regarding the third case, with a heavier mass and a greater moment of inertia, the necessary time for the model to reach a constant angular velocity, hence its equilibrium state, is with 22.95% larger than in the baseline case. As in the previously discussed case, according to the calculated percentages, the mass influence on the starting time is directly proportional. For this case, as shown in figure 9, the starting time for each case is as follows: 3.1 s for case 1; 4.5 s for case 2; 7.5 s for case 3 and 6.1 s for the baseline.

As expected, at a higher velocity inlet, the starting time of the turbine decreased considerably. However, acknowledging the decrease in power coefficient, which for this case had a value of only 0.003, it is preferred a lower wind speed for which the efficiency is greater, even though the starting time is slightly higher.

3.3. Reynolds number discussion

For the current investigation the wind velocities chosen were 12 m/s and 14 m/s, as these cases correspond to the transition from a $0.912 \times 10^5$ to a $1.064 \times 10^5$ Reynolds chord number. When the Reynolds number exceeded a value of $1 \times 10^5$, the turbine’s efficiency reduced significantly. The effect the Reynolds number has on the aerodynamic characteristics and overall performances of a wind turbine is experimentally assessed in paper [19], and the authors conclude that for highly turbulent Reynolds numbers Lenz type vertical axis wind turbines tend to have lower values for the efficiency.

In order to evaluate the turbulence in high Reynolds number flows, the vorticity magnitude contours are illustrated in the following figures (Figs. 10-13) for different azimuth angles, for wind speeds of 12 m/s and 14 m/s.

For an easier comparison between the vorticity contours at different wind speeds, figures from 10 to 13 each present the Lenz model at the same position of the blade (determined by the azimuth angle), but at different values for the inlet velocity.
In figure 10 the vorticity magnitude contours are presented for an azimuth angle of 0° at different wind speeds and implicitly different Reynolds chord numbers. Each picture has a legend represented on its left with the same interval of values in order to facilitate the comparison process. It can be observed that for a greater speed, thus a greater Reynolds number, the vorticity accentuates and has higher values. This effect, as noticed when comparing figure 10. b) with figure 10 a), creates bigger wakes, that influence the next blade and can have a negative impact on its performances.

![Figure 10. Vorticity magnitude: θ = 0°.](image)

The following images, from figure 11, illustrate the vorticity magnitude contours for a 30° azimuth angle. In this position, as opposed to the previously discussed one, the difference in intensity wakes between the two studied velocities is not as obvious. Still, it is definitely visible that for the greater Reynolds chord number and velocity inlet value the generated vortices are stronger, having a bigger impact on the overall flow, thus the overall performances.

![Figure 11. Vorticity magnitude: θ = 30°.](image)

Figure 12 illustrates the vorticity magnitude contours at an azimuth angle of 60°. Again, the differences between the two studied velocities seem slight and not as obvious as they were for an azimuth angle of 0°.
a) $V=12$ m/s  
b) $V=14$ m/s

Figure 12. Vorticity magnitude: $\theta = 60^\circ$. 

a) $V=12$ m/s  
b) $V=14$ m/s

Figure 13. Vorticity magnitude: $\theta = 90^\circ$.

For a higher Reynolds number, the flow detachment from the blade is delayed. For example in figure 13, for a lower Reynolds number, thus lower inlet velocity, it can be observed that for the blade located at an azimuth angle of $90^\circ$, the flow detaches from the blade as soon as the circular zone of the blade is over. For a higher velocity, respectively Reynolds number, the tendency of the flow is to stay attached to the blade even on its linear zone. This phenomena was reflected on the power coefficient, that was higher in the case with a velocity inlet of 12 m/s. The computed power coefficient for a velocity inlet of 12 m/s was 0.038, corresponding to a power of 8W, whereas for a velocity inlet of 14 m/s it was 0.003, corresponding to a power of 1W.

4. Conclusions
In this paper the influence of a Lenz turbine’s moment of inertia on its starting behaviour was analyzed by means of CFD simulations, using the 6DOF method in order to evaluate the time in which for different masses the turbine achieves a constant angular velocity, thus an equilibrium state.

The presented Lenz model was investigated for four different masses and moments of inertia, at two different values for the velocity inlet, 12 m/s and 14 m/s, respectively. The results concluded that the starting time is directly proportional with the mass, respectively the moment of inertia.
Furthermore, the wind speeds for the investigations were established at two different values that represent the transition from an order of $10^5$ to an order of $10^6$ for the Reynolds chord number. This permitted the assessment of the effect of highly turbulent Reynolds numbers on the discussed geometry. The vorticity magnitude for different azimuth angles and at different values for the inlet velocity was represented in order to provide an in-depth look at the flow around the blades. This study was conducted in order to predict the behaviour of the analyzed reduced scale Lenz model at high Reynolds numbers, as well as to investigate the moment of inertia’s effect on its starting behaviour. The results will be validated in the future in the aerodynamic wind tunnel.

5. References

[1] Eriksson S, Bernhoff H and Leijon M 2008 *Renewable and Sustainable Energy Reviews* vol 12 p 1419
[2] Kanyako F and Janajreh I 2015 *Sustainable Cities and Society* vol 15 p 153
[3] Johari M K, Jailil M A A and Shariff M F M (2018) *International Journal of Engineering & Technology* vol 7 p 74
[4] Bhatta M M A, Hayat N, Farooq A U, Ali Z, Jamil S R and Hussain Z (2012) *Renewable and Sustainable Energy Reviews* vol 16 p 1926
[5] Nishiooka A H and Almeida O (2018) *International Journal of Advanced Engineering Research and Science* vol 5
[6] Gohil H P and Patel S T (2014) *International Journal for Scientific Research & Development* vol 2
[7] Sivamani S, Premkumar M T, Sohail M, Mohan T and Hariram V (2017) *Data in brief* vol 15 p 1035
[8] Tu J, Yeoh G H and Liu C (2018) *Computational Fluid Dynamics: A Practical Approach* (Butterworth-Heinemann)
[9] Howell R, Qin N, Edwards J and Durrani N (2010) *Renewable Energy* vol 35 (2) p 412
[10] Bianchini A, Balduzzi F, Bachant P, Ferrara G and Ferrari L (2017) *Energy Conversion and Management* vol 136 p 318
[11] Saryazdi S M E and Boroushaki M (2018) *International Journal of Renewable Energy Development* vol 7 p 23
[12] Castelli M R, Englaro A and Benini E (2011) *Energy* vol 36 (8) p 4919
[13] Ferreira C J S, Bijl H, Busssel G and Kuik G (2007) *Journal of Physics: Conference Series* vol 75
[14] Golarche A A, Moghiman M and MalAbad S M J (2016) *Modares Mechanical Engineering* vol 15 p 143
[15] Asr M T, Nezhad E Z, Mustapha F and Wiriadiidjaja S (2016) *Energy* vol 112 p 528
[16] Wang X, Luo X, Zhuang B, Yu W and Xu H (2011) *Proc. Of the ASME-JSME-KSME 2011 Joint Fluids Engineering Conference* vol 1 p 673
[17] Menter F R (1994) *AIAA Journal* vol 38 p 1598
[18] Maalai I, Popa V and Preda D (2018) *IOP Conf. Series: Materials Science and Engineering* vol 400
[19] Deori B, Barman S, Das S, Hussain M, Basumutary S M and Sharma K K (2015) *Journal of Material Science and Mechanical Engineering* vol 2 p 62

Acknowledgments

This work was carried out within POC - Competitiveness Operational Program, supported by the EU and Romanian Minister of Research and Innovation funds, project number POC 9/01.09.2016, MySmis 105890, ID P_40_309.