Development and validation of a CFD model used for vertical axis wind turbines simulations

V Bostan, M Guțu and I Rabei
Technical University of Moldova, Department of Basics of Machinery Projecting, 9/8 Studentilor str., block of study nr. 6, Chisinau, Republic of Moldova
E-mail: marin.gutu@pmai.utm.md

Abstract. This paper describes the design of a CFD model based on the finite element analysis, used for determining the performance of a vertical axis wind turbine. The validation of the proposed model is done by simulating a real rotor, chosen to be the 6 kW Quietrevolution wind turbine. The power curves of the real and the simulated rotors are considered as comparison criteria. The goal is to have close shapes of the two power curves.

1. Introduction
Simulations are an important tool in engineering that generally lead to cost reduction of product development and time. Finite element analysis is very important means in this sense and they serve as a base for the present work. Besides choosing the appropriate mathematical model behind physics of the simulated system, it is important to choose the right shape and size of the finite elements. It is also important that the elements are well adapted for the specific system to be analysed.

This work presents a CFD model created for determining the performances of a vertical axis wind turbine. The CFD model is made using ANSYS CFX software. The CFD model is used to determine the power curve of a real wind turbine namely Quietrevolution QR 6 produced by a British company. For this turbine the real power curve is known and is compared to the simulated one. This comparison is considered as validation variant of the CFD model.

2. Parameters of the analyzed wind turbine
The analyzed vertical axis wind turbine presented in the figure 1 has the constructive parameters presented in table 1. The parameters are taken from the technical specification document [1]. From the pictures of the QR6 turbine was determined the helical angle which is 120°. The chord length is not indicated in the specification so it was determined by measuring it from the pictures by taking the proportions between the known and measured dimension of the diameter and unknown and measured dimension of the chord. This was done by analyzing many images of the turbine so that the error is as small as possible. The chord length obtained is approximately 195 mm.

Another very important parameter of the wind rotor is the airfoil. This parameter is not specified either because most probable this is kept as a commercial secret by the company. As observed from the pictures, the airfoil used seems to be symmetric. We adopted NACA 0018 as it is among the most used symmetric airfoil for vertical axis wind turbines. The CAD model of the simulated rotor is presented in the figure 2. To be noticed that the simulated rotor does not have the axial shaft and the struts. This is done in order to simplify the simulation complexity and reduce simulation time.
The annual power output curve of the QR6 turbine is given in the specification [1] up to 7.5 m/s wind speed. Up to this wind speed the power curve is extrapolated. An analysis [2] found in the internet presents a comparison between more power curves and QR6 power curve is among them. The QR6 power curve presented here is for the values of the wind speed up to 20 m/s.

### Table 1. Rotor parameters

| Parameter      | Value  |
|----------------|--------|
| Height         | 5.5 m  |
| Diameter       | 3.1 m  |
| Swept area     | 16 m²  |
| Chord length   | 0.195 m|
| Airfoil        | NACA 0018 |
| Helical angle  | 120°   |

3. **Fluid domain modeling and meshing**

The correctness of this curve was proved by comparing it with the extrapolated power curve from the official technical specification for wind speeds up to 7.5 m/s. Therefore the power curve presented in the analysis [2] is considered as the power curve of the real QR6 turbine and it is accepted as reference (figure 8).

The CAD models of the rotor, rotor domain (rotating domain) and fluid domain (static domain) were done using SolidWorks and then imported in ANSYS. The domains’ dimensions are set using the recommendations of the source [3]. The dimensions of the fluid domain are: length - 50 m, width – 24 m, height - 14 m. The dimensions of the rotor domain are: height 6.5 m, diameter 3.9 m. The rotor is placed at a distance of 10 m from the inlet and in the middle with respect to the four sides. To facilitate the boundary conditions settings the following named selections are created: Inlet, Outlet and Openings. The three surfaces of the rotor domain are named as well, that is the lids and the cylindrical surface. Also the surfaces of the blades are named. An important aspect regarding the blades is that the leading and trailing edges must be defined by distinct lines. This is done when the CAD model of the rotor is made. These lines are very helpful for setting the dimensions of meshing around the blade. An interface is created between the rotor domain and fluid domain (Fluid-Fluid).

The meshing for rotor and fluid domain is done separately (figure 3). The meshing for the fluid domain is done as follows: the maximum *Face size* and *Tet size* for the fluid domain is 650 mm. Meshing method is *automatic*. Inflation is done at the interface with the rotor domain: *First layer height* - 180 mm, *Maximum layers* - 7 and *Growth rate* is 1.19. The *element size* of the interface surfaces between rotor and domain is 140 mm. The number of elements is 615594. For the rotor domain the maximum *Face size* and *Tet size* of the element is 180 mm. The element size on the blades’ surfaces is 5×10 mm. In order to reproduce the phenomenon of the boundary layer on the surface of the blade where strong fluctuations in fluid velocity occur, prismatic finite elements have been generated by expanding them from the surface of the blade outwards. This was done by the
Inflation Layer process that was imposed on the blade surface with the Total Thickness option with Number of Layers = 9, the Growth Rate = 1.18 (relative thickness of two adjacent layers), and Growth Rate Type = Geometric (geometric or exponential expansion rate) First layer height - 1 mm (figure 4). The total number of elements is 6356488.

When checking the quality of the obtained meshing, skewness and orthogonality are the most important. The recommendation is to have maximum skewness lower than 0.95 and minimum orthogonality larger than 0.15.

Figure 3. Rotor and Stator fluid domain mesh.

Figure 4. Inflation around the blade surface.

4. Boundary conditions setup
As option for analysis type the Transient Blade Row model is chosen. The boundary conditions imposed are the following: entry into the computing field is made by the boundary determined by the rectangular base of the upstream Stator fluid domain [4]. At this border were imposed Inlet boundary conditions with the specification of the uniform and constant velocity distribution in the fixed reference system (V_x, 0, 0), where V_x is wind speed. Outlet from the computing domain is made by the downstream rectangular base by specifying the Outlet boundary conditions with the average static pressure 0 Pa. Surfaces of the Stator fluid domain are subjected to Walls boundary conditions with the free-slip specification that simulates a zero-adhesion virtual wall. Blades surfaces are subject to Walls boundary conditions with no slip specification which does not allow mass or energy transfer, and the speed on these surfaces is considered equal to 0 in relation to the speed of the adjacent cells. The surfaces at the intersection of the two Stator and Rotor subdomains are interface surfaces that model the connection of the two subdomains through the GGI method. Rotor rotation simulations specify Domain motion - Rotating and indicate the angular velocity of relative rotation \( \omega \). At this stage extra attention is required to certain details such as the direction of rotation of the rotor and wind direction, which can be changed with the (-) sign. In order to draw wind turbine rotor power curve several operating modes have been simulated, (table 2). Figure 5 shows calculation domain.
Table 2. Rotor operating modes

| Wind speed $V$, m/s | 4.6 | 6   | 8   | 10  | 12.5 | 14  |
|--------------------|-----|-----|-----|-----|------|-----|
| Number of rotation $n$, min$^{-1}$ | 97  | 126 | 168 | 211 | 263  | 295 |
| Tip speed ratio $\lambda$      |     |     |     |     |      | 3.8 |

Rotor operating modes were accepted using QR6 wind turbine specification for which rotor speed range is 100-260 min$^{-1}$ and cut in wind speed is 4.5 m/s. Taking into account rotor diameter and using well known calculation relation the approximately rotor tip speed ratio was estimated.

Using the syntax of the Command Expression Language (CEL), the following variables of interest were defined: the torque developed at the rotor axis in the fixed coordinate system ($T_z = \text{torque}_z()@\text{Blades}$) and the power ($\text{Power} = T_z \cdot \pi \cdot n / 30$), where $n$ is the rotational speed of the rotor that is set here as well.

Turbulence intensity for the domains is set medium - 5%. Turbulence model is set to be $k$-$\varepsilon$ for which $y^+$ parameter should be >11.

5. Solution and CFD results

The convergence of results has been attested by monitoring the variables of interest. Figure 6 (a) shows the power generated at the turbine shaft for the case when the rotor is rotating for wind speed of 10 m/s. The power curve was sketched using the mean value of resulted power for corresponding wind speed. In order to check flow field around the rotor, the velocity contour section for 10 m/s wind speed is depicted in the figure 6 (b). It is obvious that the blade passing through the downward zone will reduce the torque of the rotor because the air flow velocity decreases.

After comparative analysis of the both power curves (figure 7) the mean difference is around 12%. The power curve of the simulated rotor has values lower than the real wind turbine.

This deviation is caused by the lack of information about constructive parameters such as airfoil type used for the real wind turbine blades and the exact length of the blade chord. The real rotor, besides being helically curved is radially curved as it can be seen from the turbine’s pictures. The difference between the curvatures of the two rotors is another cause of error. The struts of the QR6 wind turbine seem to have an aerodynamic shape that might have positive contribution to the power output.

Figure 5. Rotor and fluid domain.
Of course, some aspects of the simulation settings might have caused errors too. For example, blade mesh dimensions and boundary layer is acceptable but not the best possible as this is very time and hardware consuming. With all deviations and the unknowns the results obtained can be considered as good. In conclusion, we can say that such a simplified CFD model can be successfully used for vertical axis wind turbine rotors performances evaluation.

![Graph](image1.png)

**Figure 6.** CFD analysis results: *a* – rotor power for 10 m/s wind speed, *b* - velocity contour.

![Graph](image2.png)

**Figure 7.** Wind turbines power curves.
6. References

[1] QR6 Vertical axis wind turbine brochure link: https://www.quietrevolution.com/products/

[2] Link: https://www.slideserve.com/romney/the-cal-epower-10-kw-vertical-axis-wind-turbine slide number 6/22

[3] Mohamed M H, Ali A M, Hafiz A 2015 A CFD analysis for H-rotor Darrieus turbine as a low speed wind energy converter Engineering Science and Technology, an International Journal 18 (1) pp 1-13.

[4] Bostan V 2014 Modele matematice în ingererie, probleme de contact, modelări și simulări numerice în aero-hidrodinamică ISBN 978-9975-80-831-6.