

NUMERICAL AND EXPERIMENTAL INVESTIGATION OF H-DARRIEUS VERTICAL AXIS WIND TURBINE

Normunds Jekabsons¹, Sabine Upnere¹, Janis Kleperis^{2,3}

¹Ventspils University College, Latvia; ²SIA "Multipla Energy", Latvia;

³Institute of Solid State Physics, Latvia

normundsj@venta.lv, upnere@gmail.com, kleperis@latnet.lv

Abstract. The present work focuses on a numerical and experimental study of the aerodynamic performance of the straight-bladed Darrieus type turbine, one of the most common vertical axis wind turbine (VAWT). The aim of the research is to evaluate the characteristic parameters for practically applicable turbine designs using Computational Fluid Dynamics (CFD) tools. Several constructions of wind turbine rotors assembled from three- or six-blade symmetric NACA0015 airfoils were investigated numerically. The two- and three-dimensional turbulent flow computations around the VAWT were performed using the Unsteady Reynolds-averaged Navier-Stokes approach. Finite Volume approximation of PDE's and solution were performed with the extended version of the open sourced CFD toolkit OpenFOAM. Rotation of the wind turbine rotor has been realized by Generalized Grid Interface (GGI) which uses weighted interpolation to evaluate and transmit the flow values across a pair of conformal or non-conformal coupled boundaries between the stationary and rotating domains. Flow was modelled as incompressible. Comparison of the overall turbine performance parameters has been studied through experimental measurements and numerical simulations. The characteristic torque trends as functions of wind free-stream velocity and the rotor rotational frequency have been obtained.

Keywords: vertical axis wind turbine, NACA0015 airfoil, torque, CFD.

Introduction

The straight-bladed Darrieus type turbine is one of the most common vertical axis wind turbines (VAWT) used to generate electricity from wind energy. VAWTs could be promising for conditions corresponding to low wind speed and urban areas [1] to improve the wind energy outcome, therefore, several configurations of small-scale wind turbines have been investigated to find them more effective for practical application. Aerodynamic behaviour of Darrieus type turbines is rather complex and therefore subject to interest for numerical and experimental studies in fluid dynamics.

Traditionally, production of VAWT power can be related to work of frontal wind impulse transfer (drag) or one produced by aerodynamic lift forces from rotating blades. The Darrieus devices are vertical-axis lift-type rotors, which are characterized by their high speed and high efficiency [2]. According to Betz's equation, a lift turbine has a theoretical efficiency of 59.3 %, while the drag type devices may yield 19-40 %. In reality, the efficiency of even modern HAWTs never exceeds 45-50 % [3]. Operation in-dependency of the wind direction is the main VAWT advantage [4]. Contrary, poor self-starting ability and mechanical safety of large devices is the main disadvantage.

In general, in Computational Fluid Dynamics (CFD) mainly three approaches have been used to deal with moving geometries: the grid adaptation, re-meshing and the grid overlay or the sliding interface techniques [5]. One of the earliest methods applied for the rotating components were single reference frame and multi reference frame (MRF). MRF is useful for several applications such as centrifugal compressors and pumps [6; 7], but our previous calculations shown that MRF is not a good choice for VAWTs modelling. In this paper Generalized Grid Interface (GGI) [8] is used to interpolate the flow values between two: rotating and stationary sub-domains.

The research contains two parts: numerical simulations and experimental measurements. There are performed two- and three-dimensional numerical calculations using the Unsteady Reynolds Averaged Navier-Stokes (URANS) method and GGI approach. Simulations have been done by open source CFD toolkit OpenFOAM, the so called extended version 1.6. Although CFD is a widely used approach to model aerodynamic characteristics of streamline bodies, the whole modelling of VAWT is still challenging, like to find the numerically stable and realistic Finite Volume (FV) implementation, appropriate set of numerical schemes as well as the models for turbulence. The characteristic torque trends as functions of the wind free-stream velocity and rotor rotational frequency have been CFD modelled for different numbers of blades and turbine solidities.

The actual measured performance is needed to validate and confirm the simulation results. It is performed independently; the obtained results are checked against the numerical findings.

For small-scale wind turbines in real dimensions, three test methods were found in the literature [9]. One is wind tunnel testing, which has the advantages of being able to precisely control the wind velocities and measure the wind turbine power output. However, in Latvia available wind tunnels are small in size and suitable only for testing of small models of VAWTs. There are very large wind tunnels abroad, for example, three European Strategic Wind tunnels (ONERA, DNW, ETW), 17 with diameter 2.5 m or above in the UK, and elsewhere, but VAWT transportation, renting the test time in foreign laboratories requires considerable budget resources. The second is field testing in natural conditions, but it can take many months to collect enough data at all wind speeds and all blade configurations. The third method that can be used for small wind turbines is vehicle-based testing [9; 10]. In these tests, the wind turbine is installed on a tower that is fixed to a truck or trailer. The performance of the turbine can be measured across a full range of wind speeds by driving the vehicle at different speeds on a controlled course. We chose this method because of its high throughput allowing rapid evaluation of VAWT performance with relatively low costs.

Experimental setup

The experimental installation is shown in Fig. 1. VAWT used in the experiment consists of 2+2+2 symmetric NACA0015 airfoils.

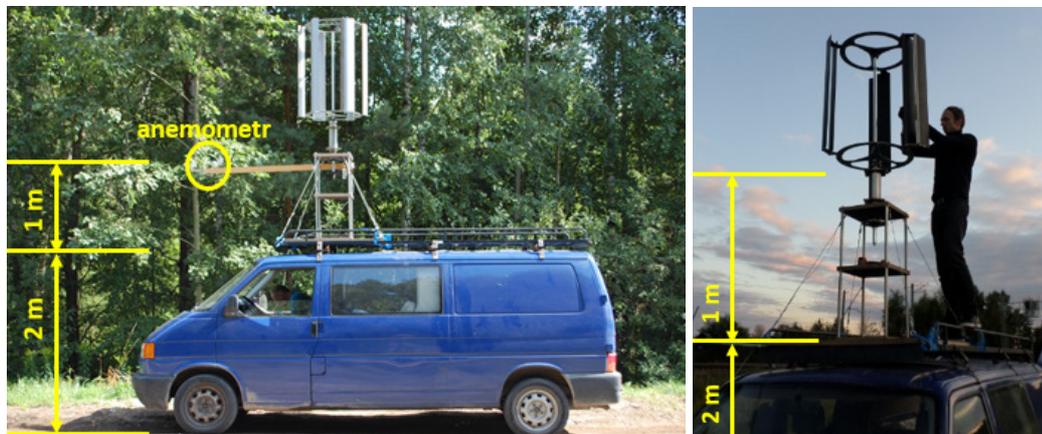


Fig. 1. Experimental installation

For the experiments, a road with low traffic load was selected, which also is wide and with smooth surface, located in flat relief and several kilometres long. The tests are performed outdoors in absence of rain and snow and without noticeable wind and wind gusts ($<2.0 \text{ m}\cdot\text{s}^{-1}$). In the experiments the car Volkswagen Carevelle, anemometers Papouch TX20ETH and FENNEL FTA-1 and the digital tachometer A13M2236B are used (Fig. 1). The height of the minivan is 2 m, and the platform 1 m high was constructed to keep as possible VAWT out of the turbulence region caused by the moving car.

Numerical implementation

In order to estimate the impact of the VAWT configuration to its performance, series of CFD runs was performed. Based on characteristic wind velocities the flow is assumed to be incompressible isothermal and turbulent. The Finite Volume method based OpenFOAM (OF) standard solver for dynamic mesh changes was used. To get closure of the problem the turbulence model has been applied. In the present study two standard OF RANS type turbulence models have been used: Spalart-Allmaras one-equation model and RNG k-epsilon two-equation model [11]. To deal with the pressure-velocity coupling segregated the algorithm PISO [12] is applied.

The operation of VAWT is characterised by the tip-speed-ratio (TSR). TSR is the ratio of the highest velocity of the rotating blade compared to the wind speed [3]:

$$TSR = \frac{R\omega}{u}, \quad (1)$$

where R – radius of the rotor, m;

u – wind free-stream velocity, $\text{m}\cdot\text{s}^{-1}$;
 ω – angular velocity, $\text{rad}\cdot\text{s}^{-1}$.

The power coefficient (c_p) is the ratio of the power generated by the wind turbine compared to the available energy in the wind [3]:

$$c_p = \frac{T\omega}{\frac{1}{2}\rho u^3 2RH} \quad (2)$$

where T – average torque, $\text{N}\cdot\text{m}$;
 H – height of the rotor, m ;
 ρ – air density, $\text{kg}\cdot\text{m}^{-3}$.

The computational domain has been decomposed in the stationary part and in the central moving sub-domain, which contains blades. The moving part revolves at the same angular velocity as the rotor. Of the computational domain the velocity inlet is defined, with a constant velocity profile, pressure outlet and two sides are symmetry boundaries. The blades are described as rotating walls with no-slip condition.

Feasibility study has been done to analyse turbulence models, mesh independence, numerical schemes and several other convergence criteria. Flow around non-rotating, single NACA0015 airfoil for different angles of attack was modelled in 2-dimensions using steady-RANS model. The calculated aerodynamic coefficients have been compared with the data from [13]. Analysis has been performed at $8.57 \text{ m}\cdot\text{s}^{-1}$ free-stream velocity corresponding to $Re=10\,000$. The results show good compliance (max $\sim 4.5\%$ error) with the experimental values at small angles of attack. If the angle is larger than 10 degrees, error increases significantly. On average, turbine blades mostly have small angles of attack at high TSR values ($\text{TSR}>2$), therefore, the applied technique is useful. Whereas at lower TSR, the results may not fit so well with the experiments.

Configurations of the analyzed wind turbines

The investigated wind turbine configuration cases of 2D are summarized in Fig. 2. There are turbines consisting of three- or six-blade symmetric NACA0015 airfoils and different rotor solidity (defined as $\sigma=Nc/D$, where N is the number of blades, c is the chord length and D is a rotor diameter).

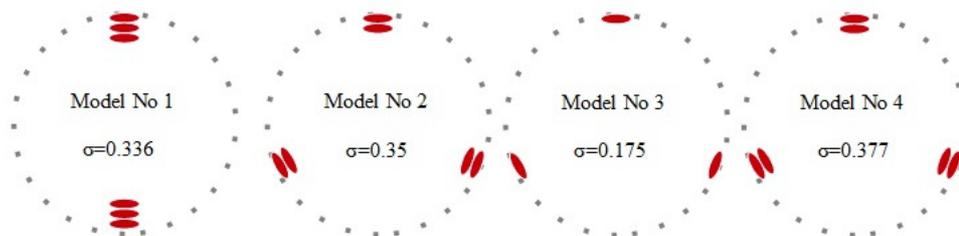


Fig. 2. **Wind turbine configurations:** No. 1 (3+3 blades); No. 2 (2+2+2 blades); No. 3 (1+1+1 blade) and No. 4 (2+2+2 blades)

Model No 1 is composed from 3+3 identical blades with 180 degrees between both sets of blades, while No 3 consists of three blades with 120 degrees for any two blades. Models No 2 and No 4 have 2+2+2 blades spaced 120 degrees away from each other. The computational domain and boundary conditions of the six-blade three-dimensional case with $\sigma=0.377$ (model No 4) is shown in Fig. 3. Existence of the plane of symmetry in the height direction is utilized for twofold mesh volume savings.

Mesh optimization has been realized to further minimize the computational cost of the 3D model. The computational domain has been divided in several sub-domains with different mesh design by means of maximum cell size (see Fig. 4).

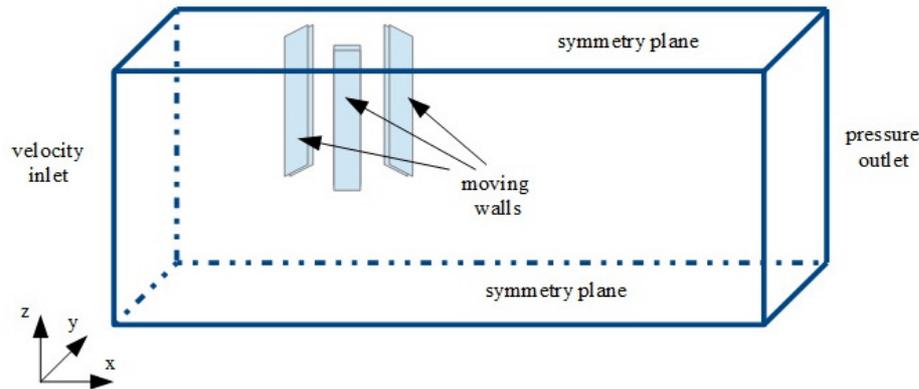


Fig. 3. Boundary conditions of three-dimensional case with 2+2+2 blades

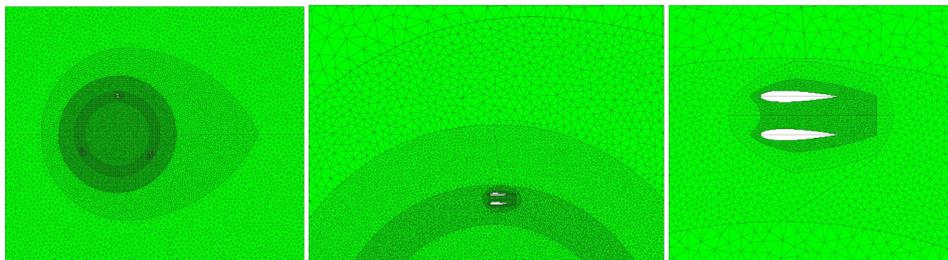


Fig. 4. Sub-domains of the mesh

Characteristic cell size near the airfoil surface has been determined by considering the turbulence model near-wall treatment of the boundary layer. For that, a non-dimensional wall distance y^+ for wall-bounded flow in the following formula is used:

$$y^+ \equiv \frac{u_\tau y}{\nu}, \quad (3)$$

where u_τ – friction velocity, $\text{m}\cdot\text{s}^{-1}$;
 y – distance to the nearest wall, m ;
 ν – local kinematic viscosity, $\text{m}^2\cdot\text{s}^{-1}$.

Hence, for high-Re boundary layer sub-models used the non-dimensional quantity y^+ is kept in the interval 30-200 for most of the blade surface most of the time.

Results and discussion

URANS can be used to capture the fluctuating part of the flow. The modelling has been performed at $6.0 \text{ m}\cdot\text{s}^{-1}$, $7.0 \text{ m}\cdot\text{s}^{-1}$, $8.57 \text{ m}\cdot\text{s}^{-1}$ and $10.0 \text{ m}\cdot\text{s}^{-1}$ free-stream velocity u . Air density is assumed to be $1.205 \text{ kg}\cdot\text{m}^{-3}$ and turbulence intensity is 0.5 %. The angular velocity of the turbine is varied from 60 revolutions per minute (rpm) to 360 rpm.

It seems to be a rule of thumb that in order to achieve a quasi-periodic solution it is necessary to model at least 5 revolutions from the initial state (see Fig. 5a). This leads to rather long computation time. Unfortunately, at the moment of writing several of the calculated cases have not reached this limit, especially the ones for higher wind speeds ($8.57 \text{ m}\cdot\text{s}^{-1}$ and $10.0 \text{ m}\cdot\text{s}^{-1}$), where the turbine may pass just 3 to 4 revolutions. It is expected that average torque of these transient model cases has higher value as in pseudo-stationary regime. Comparison of the 2D modelling results for the different turbine configurations has been shown in Fig. 5b. It can be seen that some of the points are poorly converging.

As it can be seen from Fig. 5b, at least one model case exceeds the Betz limit. It has been explained by the effect of the size of the computational domain – from the derivation of the Betz's equation it follows that the formula is valid for the case with an infinite space surrounding the turbine, whereas in our models domains have finite width, just of a few rotor diameters. Noticeable, for 2D

calculus shown in the Figure, the domain has only two dimensions as well. Although the numerical results are approximated, they are useful for comparison of the performance of different VAWT configurations.

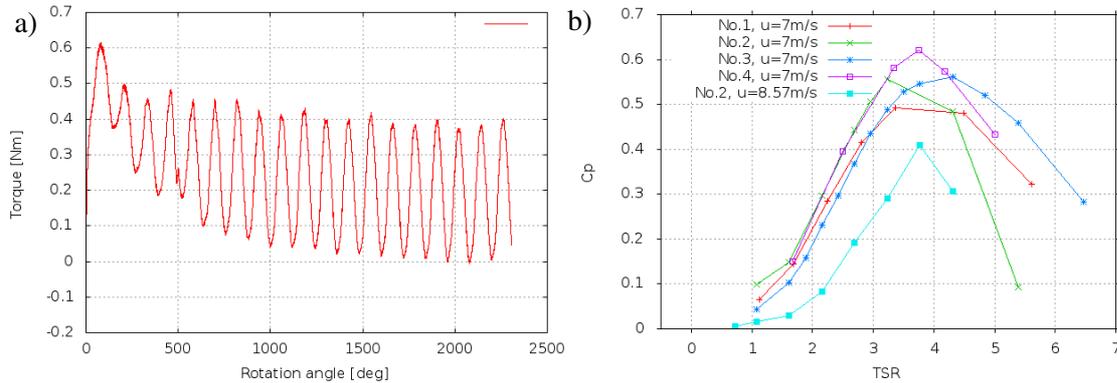


Fig. 5. Results: a – time history example of torque; b – comparison of different model configurations

In the experiment the wind turbine did not start by itself. Instead it has to be cut-in with 120 rpm applied from the motor. Then it was left by itself, the angular velocity for a while increases and finally reaches a stationary value for the given wind speed, shown as “+” in Fig 6b. We would like to speculate that this unconstrained angular rotation speed is dictated by internal (bearings of the shaft) and external frictional losses. One can use these facts in order to speculate and relate the numerical results with the measured free rotation, shown in Fig. 6b. First of all, since the turbine rotates with a minimal load, it can be a feasible assumption that maximum power is generated at lower rotational velocities. It holds for all simulations. The sample points are shown in Fig. 6b. Secondly, one can estimate the free rotation speed by assuming the moment of friction to be proportional to the angular velocity. For that estimate an iterative formula has been applied $T_n = T_0 \cdot \omega_n / \omega$. Thus, the free-rotation angular velocity is then read from the simulated torque curve, Fig. 6a, by iterative application of the above formula. Fig. 6b shows in such a manner obtained free rotation points; we will call them maximum points. These appear to be more close to the measured load curve as ones from power maxima.

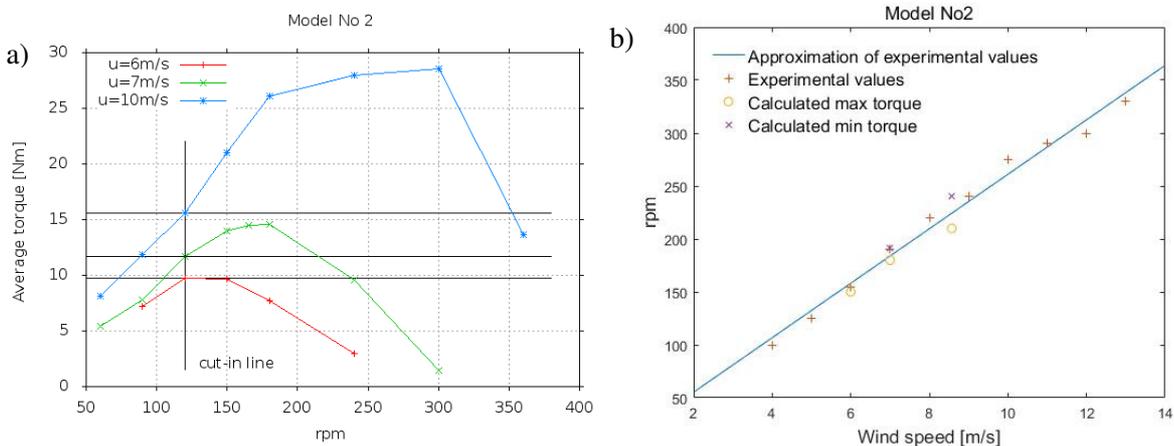


Fig. 6. Prediction of minimum and maximum points: a – numerical calculation results at several wind speed; b – comparison with the experiment

The pseudo-analysis above must be taken with a caution. One must remember the following opposite trends:

- two-dimensional calculations give higher torque values;
- to predict minimum torque points it is assumed that the cut-in speed is 120 rpm, it is possible that the necessary speed is lower, thus lower the initial torque;
- the analysis is based on the quasi-stationary model, no inertia of the shaft, rotor blades and other rotating weights on the shaft are taken into account.

In the three-dimensional case the calculated torque values are approximately 25 % lower as in the 2D model, therefore, the presented results should be considered to the upper assessment. However, at first glance, these results for us seem to be promising. Also, as it was expected, the maximum power values are lying under the experimentally measured free-rotation curve.

Conclusions

1. Feasible study of single, non-rotating airfoil showed good agreement with the experiment at small angle of attack, but error increases when the angle exceeds 10 degrees.
2. In some cases the calculation results give that the wind turbine efficiency exceeds the Betz limit, which does not contradict with the theory due to limited size of the computation domain.
3. As it was expected, the maximum values are lying under the experimental curve and the minimum values are above the curve.
4. The two-dimensional numerical simulation method used in this paper gives larger wind turbine efficiency as the three-dimensional case. It is explained by diverse sizes of computational domains and the flow three-dimensional nature, which causes important impact on the blade ends. However, 2D simulations are useful for comparison of different wind turbine configurations.
5. In some cases the quasi-stationary solution is not achieved due to limited computational resources.

References

1. Burlando M., et al. Numerical and experimental methods to investigate the behaviour of vertical-axis wind turbines with stator. *J. of Wind Engineering and Industrial Aerodynamics*, vol. 144, 2015, pp. 125-133.
2. Sutherland H.J., et al. A retrospective of VAWT technology. New Mexico: Sandia National Laboratories, 2012, 64 p.
3. Daroczy L. et al. Comparative analysis of turbulence models for the aerodynamic simulation of H-Darrieus rotors. *Energy*, vol. 90, 2015, pp. 680-690.
4. Balduzzi F., et al. Feasibility analysis of a Darrieus vertical-axis wind turbine installation in the rooftop of a building. *Applied Energy*, vol. 97, 2012, pp. 921-929.
5. Ganine V. et al. Enhancing performance and scalability of data transfer across sliding grid interfaces for time-accurate unsteady simulations of multistage turbomachinery flows. *Computers & Fluids*, vol. 115, 2015, pp. 140-153.
6. Maksimovic P. Next steps in CFD. *Pump Design & Simulation*, November 2005, pp. 44-47.
7. O. Petit et al. The ERCOFTAC centrifugal pump OpenFOAM case-study. *Proceedings of 3rd IAHR International Meeting of the Workgroup on Cavitations and Dynamic Problems in Hydraulic Machinery and Systems*, October 14-16, 2009, Brno, Czech Republic.
8. Beaudoin M., Jasak H. Development of a Generalized Grid Interface for Turbomachinery simulations with OpenFOAM. *Proceedings of International conference "Open Source CFD International Conference"*, December 4-5, 2008, Berlin, Germany.
9. Rachman A. et al. An Experiment on Horizontal and Vertical Wind Turbines with Incorporation of Rounded Shroud Device Using Wind Simulation in a Vehicle. *International Journal of Innovation and Applied Studies*, vol. 3, 2013, pp. 365-374.
10. Matsumiya H. et al. Field Operation and Track Tests of 1-kW Small Wind Turbine under High Wind Conditions. *Journal of Solar Energy Engineering*, vol. 32, 2010, pp. 11002-11010.
11. Davidson L. An introduction to turbulence models. Goteborg: Chalmers University of Technology, 2015. 48 p.
12. Ferziger J.H., Peric M. *Computational methods for fluid dynamics*. Third edition. Berlin: Springer, 2002. 423 p.
13. Sheldahl R.E., Klimas P.C. Aerodynamic characteristics of 7 symmetrical airfoil sections through 180-degree angle of attack for use in aerodynamic analysis of VAWT. New Mexico: Sandia National Laboratories, 1981. 124 p.