TKI 2016 XIV Konferencja Naukowo-Techniczna
TECHNIKI KOMPUTEROWE W INŻYNIERII

Design and performance numerical verification of a small Diffuser Augmented Wind Turbine

Krzysztof Sobczak, Konrad Kacprzak, Rafał Wieczorek, Michał Lipian, Maciej Karczewski, Krzysztof Jóźwik
Institute of Turbomachinery, Lodz University of Turbomachinery
email: krzysztof.sobczak@p.lodz.pl, konrad.kacprzak@ p.lodz.pl, rafiki_lodz@poczta.onet.pl, michal.lipian@p.lodz.pl,
maciej.karczewski@p.lodz.pl, krzysztof.jozwik@p.lodz.pl

ABSTRACT: The investigations were aimed to develop a numerical procedure for designing a small Diffuser Augmented Wind Turbine (DAWT). Owing to a strong interaction between the flow acceleration caused by the diffuser and the turbine rotor, standard procedures used for bare Horizontal Axis Wind Turbines (HAWT) could not be applied. The proposed iterative procedure was based on two-dimensional (2D) numerical flow simulations through and around the diffuser, where the rotor influence was taken into account by inserting a source term, a body force, into the momentum equation. A parametric study of different blade geometries was performed to select the optimal blade and diffuser geometries. Selected, optimal designs of the turbine were numerically verified with three-dimensional (3D) simulations. Significant differences with respect to the 2D predictions were observed, which resulted from complex 3D flow structures in the diffuser.

KEYWORDS: wind turbine, flanged diffuser, design procedure, CFD, separations

1. Introduction

The power of wind turbines is proportional to the cubic power of wind velocity. Therefore, over the time different wind acceleration devices, e.g. in the form of diffusers surrounding a HAWT, were developed to increase the rotor induction velocity. The most important aspects of diffuser applications are presented, e.g., in [1] or [2]. Their different arrangements were investigated mostly experimentally. The continuous development in Computational Fluid Dynamics (CFD) methods and computer performance allowed researchers to investigate the problem numerically with increasing precision. The turbine equipped with a diffuser with a flange (brim) was proposed in [3] and further investigated experimentally [2] and numerically [4]. Flow separations downstream of the brim decrease the pressure in this region and increase the flow rate through the turbine rotor (Fig. 1).

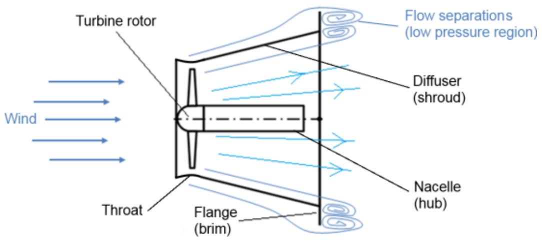

Fig. 1. Scheme of DAWT with brim

An interaction between the accelerated flow in the diffuser and the wind turbine rotor makes the design process more difficult than for a typical bare HAWT. A literature review of DAWTs did not reveal one universal approach to their design. Therefore, it was decided to develop own numerical procedures of turbine design which were able to take into account the rotor influence on the flow acceleration in the diffuser.

2. Numerical procedure

Two-dimensional 2D numerical simulations of the flow with the Reynolds Averaged Navier Stokes (RANS) method were incorporated in ANSYS CFX. The computational domain used in the procedure is presented in Fig. 2. The rotor load was taken into account by adding the body force term according to the procedure presented in [3] by means of the porous domain. The interaction of the flow through the diffuser and the rotor load required the iterative procedure presented in Fig. 3 in blue. At the beginning, input data (flow conditions, diffuser geometry, number of rotor blades) were defined. Additionally, distributions of the profile chord along the blade height were defined. It was also assumed that at all radii the SG family airfoils (SG6040 and SG6041) were positioned at their optimal angle of attack, defined by the optimum lift-to-drag ratio.

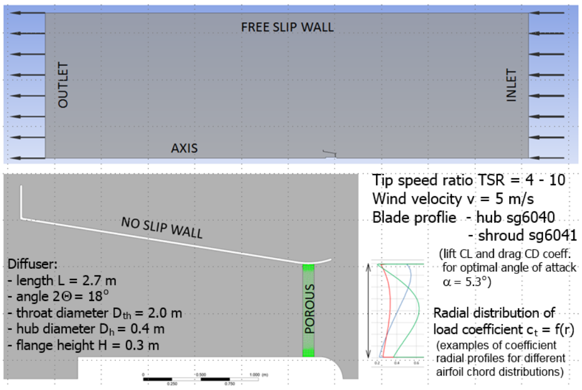

Fig. 2. Computational domain arrangement

The iterative procedure started from an arbitrary initial velocity distribution for which the radial distribution of the rotor load was estimated. Then, the CFD simulation was run and provided a new velocity field which took into account the rotor load. In the next loop, the rotor load was adjusted according to a new velocity distribution in the diffuser and the turbine power was estimated. The procedure was repeated until the convergence of the turbine power was achieved (relative difference between the power coefficient estimated in two subsequent iterations was less than selected threshold value). Usually, up to 10 iterative loops were sufficient to reach the convergence and the procedure lasted less than half an hour on a PC. Additionally, a distribution of the blade pitch angle along the blade height could be derived and blade could be designed.

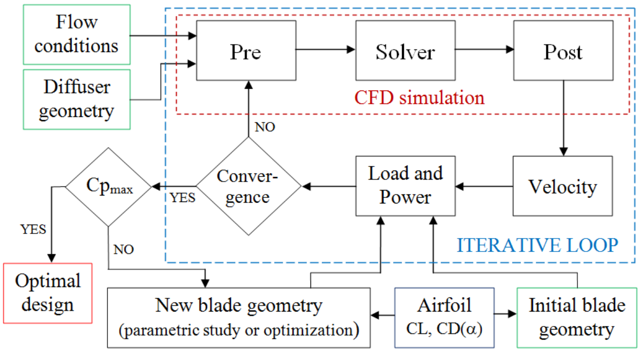

Fig. 3. Numerical procedure scheme

The procedure could be repeated for other blade geometries, i.e. for different radial distributions of airfoil chords with a parametric study approach or automatic optimization. In the studies presented in this paper, the parametric study was applied on the assumption of a linear distribution of the airfoil chords along the blade height. The tests indicated that the optimal ratio of the hub-to-shroud chords was close to 3 (A/C). By repeating the procedure for different chord distributions, the rotor performance characteristics could be determined and the optimal blade configuration indicated. In Fig. 4 one can see characteristics for different Tip Speed Ratios (TSR) of the rotor. The procedure was repeated also for different arrangements of the diffuser.

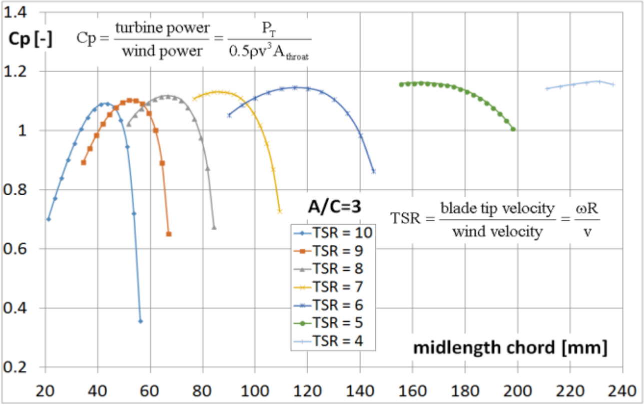

Fig. 4. Power coefficient prediction for different blade chords and tip speed ratios

In the cases of TSR = 6 and 8, optimal blade configurations were selected and 3D numerical simulations were carried out. The circumferentially averaged velocity distributions in the diffuser from these simulations are compared to 2D predictions in Fig. 5. Predicted values of the power coefficient are shown in Table 1. As one can see, the 2D method overestimates the turbine performance. Discrepancies in the turbine performance prediction result from a complex 3D flow structure in the diffuser induced by the blade presence. Much higher discrepancies were present in the case of TSR = 6, where longer chords intensified the flow field more disturbed as presented in Fig. 6. In further steps, the blade manufacturing method has to be selected and the structural analysis performed.

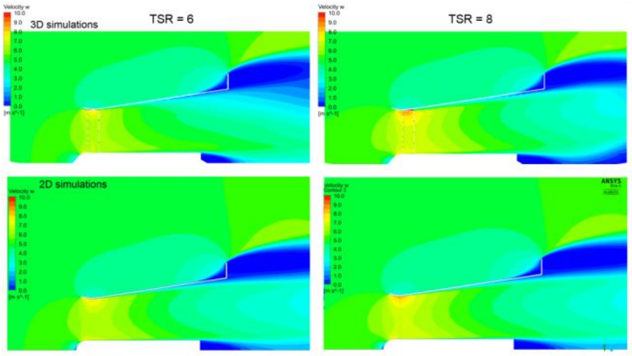

Fig. 5. Comparison of 2D and 3D predictions

Table 1. Simulations results for 2D and 3D flow predictions

	TSR = 6		TSR = 8	
	2D	3D	2D	3D
Cp [-]	1.13	0.748	1.12	0.950

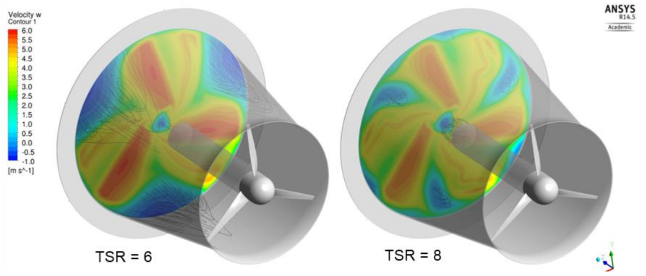

Fig. 6. Comparison of the 3D flow field prediction at the diffuser outlet for TSR = 6 and 8

3. Summary

A custom-made numerical procedure was used to develop the Diffuser Augmented Wind Turbine. It is based on a simplified 2D CFD flow prediction. Selected optimum designs were next verified with 3D accurate CFD flow simulations. The obtained results showed that:

1) 2D procedure allowed different geometries of the rotor blade and diffuser to be tested in a short period of time and the optimal design to be selected.
2) Upon verification with the 3D CFD simulations, the 2D procedure overestimated the turbine performance due to inability to account for complex 3D flow structures induced by the blades.

The research leading to these results has received funding from the Polish-Norwegian Research Programme operated by the National Centre for Research and Development under the Norwegian Financial Mechanism 2009-2014 in the frame of Project Contract No. Pol-Nor/200957/47/2013

References

[1] Gilbert B., Foreman K., *Experiments with a diffuser-augmented model wind turbine*, J. Energy Resources Technology, Vol. 105, pp. 46-53, 1983.

[2] Ohya Y., Karasudani T., Sakurai A., Abe K., Inoue M., *Development of a shrouded wind turbine with a flanged diffuser*, J. Wind Eng. Ind. Aerodyn., Vol. 96, pp. 524–539, 2008.

[3] Abe K., Nishida M., Sakurai A., Ohya Y., Kihara H., Wada E., Sato K., *Experimental and numerical investigations of flow fields behind a small wind turbine with a flanged diffuser*, J. Wind Eng. Ind. Aerodyn., Vol. 93, pp. 951–970, 2005.

[4] Takahashi S., Hata Y., Ohya Y., Karasudani T., Uchida T., *Behavior of the Blade Tip Vortices of a Wind Turbine Equipped with a Brimmed-Diffuser Shroud*, Energies, Vol. 5, pp. 5229-5242, 2012.