Numerical studies of flows around a wind turbine equipped with a flanged diffuser shroud by using an actuator-disc model

Masaru Hasegawa[a], Yuji Ohya[b], Hiroaki Kume[c]

[a]*Dept. of Aeronautics and Astronautics, Kyushu University, 6-1Kasuga-koen,Kasuga,Japan*
[b]*Research Institute for Applied Mechanics, Kyushu University, 6-1 Kasuga-koen,Kasuga,Japan*
[c]*Kawasaki Heavy Industry, Kawasakimati 1 Kakamigahara, Japan*

ABSTRACT: Unsteady 3-D direct numerical simulations based on FDM are carried out for flow fields around a wind turbine equipped with a flanged diffuser. Generally, it is difficult to simulate numerically the flow around rotational bodies like rotors of wind turbines, because of unsteadiness due to a moving body and complex geometry. Therefore, we have devised an actuator-disc model for a wind turbine for simulating the resistance and rotational forces on fluid. Introduced those volume forces derived from the actuator-disc model into the external terms in N-S equations, the unsteady flow around a wind turbine can be simulated. The results of numerical simulations are compared with the wind tunnel tests and show a good agreement for the velocity and pressure fields.

KEYWORDS: Wind Energy, Wind Turbine, Flanged Diffuser, Actuator Disc, Wind Tunnel Test, CFD

1 INTRODUCTION

The power in wind is well known to be proportional to cubic power of the wind velocity approaching the wind turbine. This means that even a small amount of acceleration gives a large increase in the energy output. Therefore, many research groups have tried to find a way to accelerate the approaching wind effectively.

Recently, Ohya et al.[1)] have developed an effective wind-acceleration system. Although it adopts a diffuser-shaped structure surrounding a wind turbine like the others previously proposed, the feature that distinguishes it from the others is a large flange attached at the exit of diffuser shroud. Figure 1 illustrates an overview of the present wind-acceleration system. A flange generates a large separation behind it, where a very low-pressure region appears to draw more wind compared to a diffuser with no flange. Owing to this effect, the flow coming into the diffuser can be effectively concentrated and accelerated. In this system, the maximum velocity is obtained near the inlet of diffuser and thus a wind turbine is located there as shown in Figure 1.

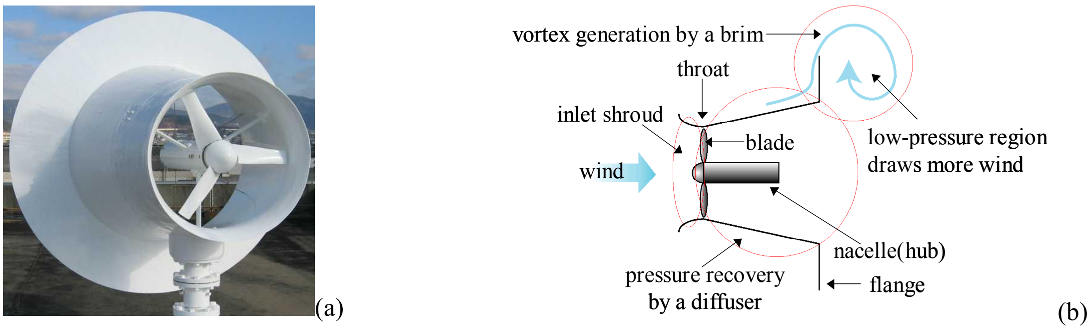

Figure 1. Wind turbine equipped with a flanged diffuser shroud:(a) 500W prototype;(b) flow mechanism around a flanged diffuser. Five-fold increase in output power as compared to conventional wind turbines is achieved.

2 NUMERICAL SIMULATIONON AROUND A WIND TURBINE EQUIPPED WITH A FLANGED DIFFUSER SHROUD

2.1 *Computational method*

A three-dimensional direct numerical simulation (DNS) based on the finite-difference method (FDM) was carried out for the flows around a wind turbine equipped with a flanged diffuser shroud. The governing equations consist of the continuity and Navier-Stokes equations. Figure 2 illustrates a computational grid. A H-O type computational grid is adopted with a grid number of $160 \times 85 \times 61$. For the boundary conditions, a uniform flow at the inlet, Sommerfeld radiation condition (SRC) at the outlet, and a slip condition at the periphery are adopted. *D* denotes the throat diameter of a flanged diffuser. The detail of simulation code is described in Table 1.

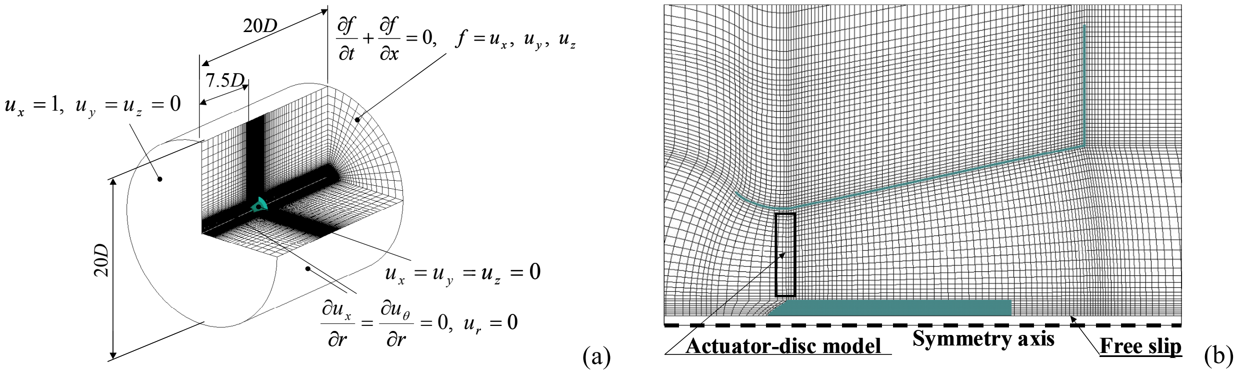

Figure 2. Computational grid:(a) Whole domain and boundary conditions;(b) Computational grid around a flanged diffuser shroud

Table 1. Simulation code and computational conditions

Coordinate system	3D BFC
Variable arrangement	collocated
Computational grid	H-O type
Scheme	Finite difference method (DNS)
Coupling algorithm	F-S method
Time advancement method	Euler explicit method
Discretization of advective terms	3^{rd}-order upwind scheme
Iteration formula for Poisson equation	SOR method
Reynolds number	10,000 (based on a flanged diffuser throat diameter *D*)

2.2 *Actuator-disc model*

Generally, it is difficult to simulate numerically the flow around rotational bodies like rotors of wind turbines, because of unsteadiness due to the moving body and complex geometry. Instead, Abe et al.[2,3] adopted a disc-loading method and Sorensen et al. [4] adopted an actuator-disc model as a numerical model for wind turbines.

To model the resistance and rotational forces acting on the fluid passing a wind turbine, we have devised an actuator-disc model. A new type of actuator-disc model is based on a blade-element theory. If a set of characteristic parameters, such as blade aerodynamic coefficients, pitch angles and the blade shape are given, the external forces acting on rotating blades are specified. The blade aerodynamic coefficients of a wind turbine obtained in wind tunnel tests were used as shown in Figure 3. The volume forces are shown in Eqs. (1), (2). Eqs.(1) and (2) are substituted into the external force terms in the Navier-Stokes equations.

$$f_x = -(C_L \cos\beta + C_D \sin\beta)\frac{Zc}{4\pi r \Delta L}\rho V^2 \tag{1}$$

$$f_\theta = -(C_L \sin\beta - C_D \cos\beta)\frac{Zc}{4\pi r \Delta L}\rho V^2 \tag{2}$$

Here Z is the number of blades, c is the chord length, r is the position in the radius direction, ρ is the density, and $\beta=\tan^{-1}(u_x/(r\omega-u_\theta))$. In Eqs. (1) and (2), C_L and C_D are, respectively, the lift and drag coefficients for the relative angle of attack, $\alpha=\beta-\gamma$, where γ is the pitch angle. Figure 4 illustrates an overview of lift and drag acting on a blade element of a wind turbine.

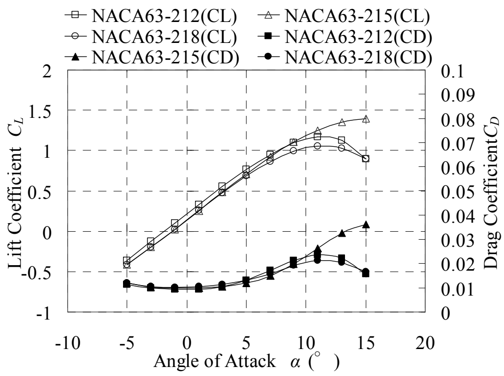

Figure 3. Lift and drag coefficients of a blade

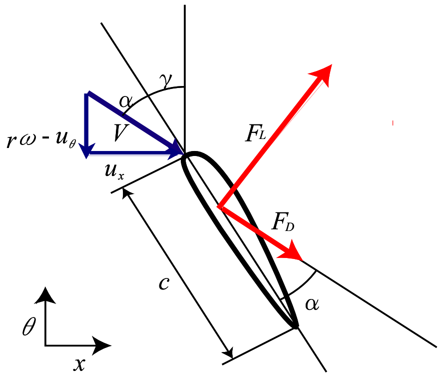

Figure 4. Overview of lift and drag acting on a blade element of a wind turbine

2.3 *Computational results*

In Figures 5-6, time-averaged wind velocity and static pressure estimated by a numerical simulation are compared with experimental results. A good agreement is seen between them.

To investigate the effect of actuator-disc model on the flow passing a diffuser shroud, the instantaneous flow fields and the three-dimensional time-averaged streamlines without and with a wind turbine are compared in Figures 7-8.

As seen in the Figure 7(a) without a wind turbine, a massive separation is generated all over the near-wall region inside the diffuser shroud. On the other hand, in the Figure 7(b) with a wind turbine, the flow goes along the inside wall with no separation. A volume force, Eq. (1), means a resistance force on fluid, playing a rule of the control of a flow separation.

As seen in the Figure 8(a) without a wind turbine, the streamlines spread in an axisymmetric plain. On the other hand, in the Figure 8(b) with a wind turbine, the streamlines spread three-dimensionally after passing an actuator-disc model. The other volume force, Eq. (2), means a rotational force on fluid, leading to the generation of a fully three-dimensional flow.

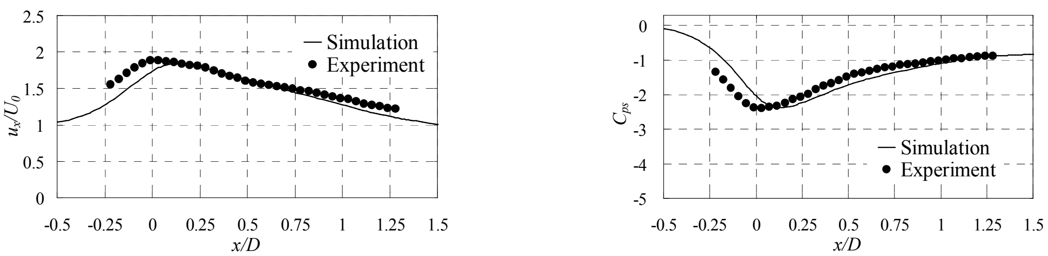

Figure 5. Distribution of wind velocity and static pressure without a wind turbine along the streamwise line at r=0.3D

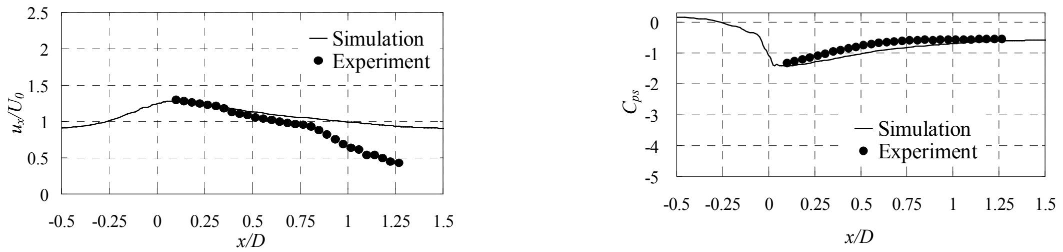

Figure 6. Distribution of wind velocity and static pressure with a wind turbine along the streamwise line at r=0.3D

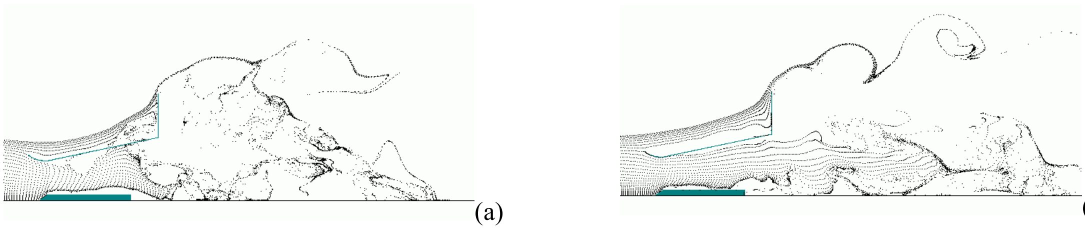
(a) (b)

Figure 7. Instantaneous flow fields:(a) without a wind turbine;(b) with a wind turbine

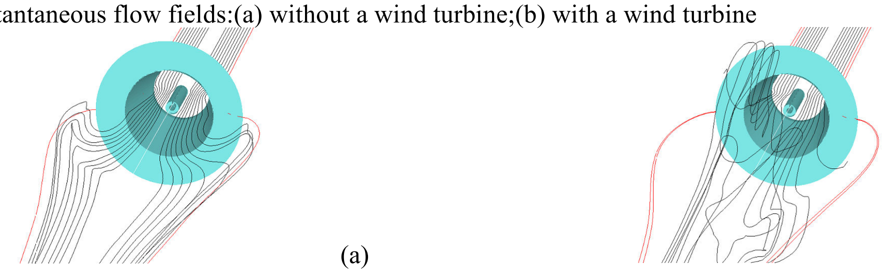
(a) (b)

Figure 8. Three-dimensional time-averaged streamlines:(a) without a wind turbine;(b) with a wind turbine

3 CONCLUSIONS

Numerical studies based on 3-D DNS were carried out for flow fields of a wind turbine equipped with a flanged diffuser shroud. For a wind turbine, we have devised an actuator-disc model which represents resistance and rotational forces on the approaching wind. The results obtained are as follows:

- The time-averaged wind velocity and static pressure inside a diffuser shroud show good agreement between numerical simulations and wind tunnel tests.
- A volume force representing a resistance force on fluid can control a flow separation inside a diffuser shroud. The other volume force representing a rotational force on fluid can generate a fully three-dimensional flow.

4 REFERENCES

1. Y. Ohya, T.Karasudani, A.Sakurai, M.Inoue, Development of a high-performance wind turbine equipped with a brimmed diffuser shroud, Trans. Japan Soc. Aeronaut. Space Sci. 49 (2006), in press.
2. K. Abe, Y. Ohya, An investigation of flow fields around flanged diffusers using CFD, J. Wind Eng. Indust. Aerodyn.92 (2004) pp.315-330.
3. K. Abe, M. Nishida, A. Sakurai, Y. Ohya, H. Kihara, E. Wada, K. Sato, Experimental and numerical investigations of flow fields behind a small wind turbine with a flanged diffuser, J. Wind Eng. Indust. Aerodyn.93 (2005) pp.951-970.
4. J.N. Sorensen, W.Z. Shen, Numerical Modeling of Wind Turbine Wakes, Trans. ASME, J. Fluids Eng.124 (2002) pp.393-399