Aerodynamic Performance and Flow Structure Investigation of Contra-rotating Wind Turbines by CFD and Experimental Methods

M Faisal, X Zhao*, M-H Kang and K You
School of Aeronautics, Northwestern Polytechnical University, Xi’an, Shaanxi, 710072, China

*Corresponding author: xuzhao@nwpu.edu.cn

Abstract. Wind tunnel experiment was carried out to measure mechanical power of the contra-rotating wind turbine (CRWT) and single rotor wind turbine (SRWT) under various working conditions. Particle Image Velocimetry (PIV) device was employed to measure the transient flow field on the meridional plane of the rotors. The azimuthal velocity vector, magnitude and streamline were obtained by the average PIV image of ten intervals in a rotation cycle. CFD model of steady flow was developed to predict aerodynamic performance and flow structure of the CRWTs using S-A turbulent model. The single channel model was employed with periodic boundary condition. CFD results were validated with wind tunnel test in literature and experiment in this paper. Results show that CFD method has a satisfied accuracy in power prediction, with a relative error less than 34%. The azimuthal average velocity distributions predicted by experimental and CFD methods are close and the maximum value are approximate to each other. Results demonstrate a high velocity zone at the outside of the front rotor tip and a low velocity zone near the middle of the rear rotor. CRWT has a complicated flow structure. Particularly at the rear rotor, flow becomes uneven and streamline curvatures are more ununiformed.

1. Introduction
Contra-rotating wind turbines (CRWTs) are investigated for high energy efficiency of power plant and high wind energy utilization rate of wind farm. According to theory, the maximum energy conversion efficiencies are 0.59 and 0.64 correspondingly for single and two turbines having identical radius[1]. Besides, CRWT has other attractive features such as near-zero reactive torque on the tower, near-zero swirl in the wake, relative high inter-rotor rotational speed and high extraction of energy at low wind speed[2]. It enhances turbine efficiency[2-5], stability, densification, annual energy production[6] and therefore improves wind farm efficiency, reduces cut-in wind speed[2] and generation cost. It is particularly suitable for the fixed cross-sectional area requirement or small turbine spacing situation[7].

The flow structure of the CRWT hasn’t been fully understood, leaving the flow interaction between the front and rear rotor unrevealed. Although researchers have carried out some investigation of flow field structure of the CRWT based on free wake vortex lattice method[8] or PIV measurement of the flow field[9]. The tip vortex in CRWT were found to expand wider than in SRWT. The results are limited on theory assumptions or based on the mini scale of the turbine. To obtain a real flow structure of the 1.1m diameter turbines, both the wind tunnel experiment[10] and CFD simulation were carried out.
2. Validation of CFD result with predecessors experimental data

Usui et al. [11] carried an experiment of CRWT in Japan in 2012. It is composed of a 3-bladed front rotor and a 5-bladed rear rotor. The radius of the front and rear rotors are 0.25m and 0.21m. The blades were configured by airfoil MEL002. The wind tunnel experiment were carried out to obtain power and velocity component in the flow field. The velocity component were measured by the five holes pitot tube at different section perpendicular to the axis away from the downstream of the flow (M1-M3). During the experiment, the wind speed, rotor speed and pitch angle were verified.

CFD software NUMECA was employed to validate the test data. Navier-Stokes equation was discretized with S-A turbulent model employed. The computational domain is a combined region \([0.1R_f,5R_f]\) \(\times\) \([0,2\pi/3]\) \(\times\) \([−3R_f,0.1R_f]\), where \(R_f\) is the front rotor radius, see Figure 1. Mesh division is \(87 \times 51 \times 107\) and \(87 \times 63 \times 99\) respectively. Total grid number is 1.54 million in 18 blocks. Similarly, 17 layers of grids are arranged in the boundary layer, thickness for the first is \(3 \times 10^{-5}\)m, expansion ratio is 1.4, resulting in \(Y^+\) on the blades less than 10. The interface between the front and rear domain is set conservation coupling by pitching row type to guarantee the conservation of mass, moment and energy.

The boundary condition of the model are set as: far field for the circumferential and bottom surfaces of the model, periodic boundary for two pairs of shaft sections, Navier-Stokes wall for the rotating blades and Euler wall for the hub.

A typical working condition was selected for benchmark. The free stream velocity is 12m/s, the front rotor tip installation angle is 5.7° and tip speed ratio is 3.4. The rear rotor tip installation angle is 15° and tip speed ratio is 2.5. The axial distance between the front and the rear wind rotors is 0.04m. Turbine power efficiency was calculated and given in table 1. It can be found that CFD result is greater than that of the experiment[11]. The energy efficiency difference for the front, rear and CRWT are 1.14%, 0.42% and 1.56% respectively. The errors are quite small which demonstrated that CFD prediction is accurate.

To validate the flow details across the CRWT, average tangential velocity \((V_t)\) and axial velocity \((V_m)\) at different cross sections perpendicular to the axis were compared, see Figure 2. Section M1 is in the upstream of the front turbine, M2 in the middle of the front and rear turbine, and M3 in the downstream of the rear turbine. It is clearly that the CFD result is close to that of the experiment, except small difference near the hub. This is due to the reason that in CFD, hub is treated as inviscid Euler wall. But in experiment, it was viscous and rotating wall. However, the velocity comparison shows that CFD simulation is accurate in flow field prediction.

### Table 1. Energy efficiency comparison for the CRWT

|          | C1p/% | C2p/% | C3p/% |
|----------|-------|-------|-------|
| EXP[11]  | 18    | 13    | 31    |
| CFD      | 19.14 | 13.42 | 32.56 |

**Figure 1.** CFD model and surface mesh on blades

**Figure 2.** Average tangential velocity and average axial velocity at different cross sections
3. CFD result comparison with authors’ experimental data

3.1. Wind tunnel experiment

The CRWT experiment was carried out in the low speed acoustic wind tunnel in Northwest Polytechnic University. It is a recirculation tunnel with an open test section in plenum chamber. The free stream velocity at the test section is 5~45 m/s with accuracy 0.5%, while the flow turbulence no more than 0.2%. The layout of the power measurement is shown in figure 3. Both turbines are composed of 3 blades and drive their own permanent magnetic generator via a torque meter. They have identical radius of 0.55m. The blades were made by wood and configured by NACA4412 airfoil. The front blades have a smaller chord length than the rear blades. They are different twist angles. The geometry of the blades were designed by aerodynamic coupling design and more details can be found in literature[10]. With the connection of a chain wheel, the rear rotor drove a parallel shaft. The output of each generators were linked with rectifier bridges. Therefore, the alternating currents were rectified to direct currents. As variable loads, two slide rheostats were adjusted to change the direct currents, resulting in turbine rotation speed change. The rotation of the front and rear rotor were at range 200~900rpm at wind velocity range 6~14m/s. For SRWT measurement, the rear rotor was removed and the single rotor was fixed at the front rotor location.

![Figure 3: Test model of CRWT](image1)

![Figure 4: PIV measurement of flow field](image2)

Figure 4 shows the flow field measurement layout. The meridional plane was considered to be the key zone for flow field investigation. Therefore, transient flow measurement was carried on. The blade tip zone was focused at range 293.5 mm × 293.5mm, see more details in section 3.3. The three-dimensional PIV system was made by Dantec company in Denmark, which is mainly composed of laser, CCD camera, synchronous controller and dynamic studio, a professional image processing software. Among them, the laser model is class 4 laser product of Dantec company. The pulse duration is 4ns. The laser wavelengths are 1064nm and 532nm. The CCD camera pixel is 2048 × 2048. The lens is Nikon 85mm f/1.4 lens. The maximum sampling frequency is 10Hz. In the experiment, 400W smoke generator was used as smoke generation. The azimuth locking technology adopts self-developed synchronous controller, which mainly includes Omron E6B2-CWZ6C encoder, signal acquisition and trigger circuit board and LCD. The trigger frequency and phase can be set through the touch of screen display window.

3.2. Power comparison

Several working conditions were selected for power comparison, see table 2. As for the single turbine, when the free stream velocity is 14m/s, the blade root installation angle is 45.54° and the rotating speed is 730rpm. For free stream velocity 10m/s, the blade root installation angle is 40° and the rotating speed is 519rpm. For the CRWT, when the free stream are 14m/s and 10m/s respectively, the blade root installation angles are both 35° for front blade and 42° for rear blades. However, the rotating speeds are 848rpm for the front rotor and 469rpm for the rear rotor in free stream 14m/s case. The rotating speeds
are 200rpm for the front rotor and 180 for the rear rotor in free stream 10m/s case. The distance between front rotor and rear rotor is 0.22m.

In the experiment, the thickness of the blade trailing edge were manufactured in 3mm for strength consideration. While the CFD model applied the full size of airfoil, leaving the trailing edge smaller than 1mm. Considering the mean chord length ratio of the test model to CFD model is 0.932. Take the assumption that the power is proportional to cord length of the blade. The original mechanical power obtained from the experiment was modified, derived by ratio 0.932. The modified value were shown in table 2 as the experimental power.

The numerical models of single and CRWT were created separately. The mechanical power obtained by experiment were used as a reference. In table 2, the power ratio of CFD method to experimental data were listed in cracket.

| Free stream(m/s) | SRWT(W) | CRWT | Total |
|------------------|---------|------|-------|
|                  | EXP     | CFD (ratio) | EXP | CFD (ratio) | EXP | CFD(ratio) |
| 14               | 456     | 611(1.34)   | 511.8 | 570(1.11)   | 157.73 | 205.57(1.30) | 669.53 | 727.73(1.09) |
| 10               | 187.78  | 202.68(1.08) | 22.53 | 23.64(1.05) | 39.57 | 36.21(0.92)  | 62.10  | 59.85(0.96)  |

As can be seen from table 2, power predictd by CFD simulation is 0.92~1.34 of the experimental data. Generally, simulation powers are a little bit greater than the experiment. This is because in the experiment, the friction of gear, bearing, torque meter and other mechanical structures consume a certain amount of power, while they are ignored in CFD model. Meanwhile, the test rig has a certain obstruction effect on the free stream, which is also ignored in CFD model. Never the less, CFD model has a satisfying accuracy in power prediction.

3.3 Flow field comparison
The PIV measurement were taken near blade tip. Figure 5 shows the shooting zone with size 293.5 mm × 293.5mm near blade tip. The captured region is 0.25m~0.61m on radial direction, which is from middle of the blade to the outside of the tip. Once the blade started to rotate, CCD camera caught the picture at 0.1s interval. And the 10 intervals results were averaged. For example, at free stream velocity 14m/s, the averaged velocity vector was shown on the left of figure 6.

As can be seen from table 2, power predictd by CFD simulation is 0.92~1.34 of the experimental data. Generally, simulation powers are a little bit greater than the experiment. This is because in the experiment, the friction of gear, bearing, torque meter and other mechanical structures consume a certain amount of power, while they are ignored in CFD model. Meanwhile, the test rig has a certain obstruction effect on the free stream, which is also ignored in CFD model. Never the less, CFD model has a satisfying accuracy in power prediction.

3.3 Flow field comparison
The PIV measurement were taken near blade tip. Figure 5 shows the shooting zone with size 293.5 mm × 293.5mm near blade tip. The captured region is 0.25m~0.61m on radial direction, which is from middle of the blade to the outside of the tip. Once the blade started to rotate, CCD camera caught the picture at 0.1s interval. And the 10 intervals results were averaged. For example, at free stream velocity 14m/s, the averaged velocity vector was shown on the left of figure 6.

![Figure 5](image1.png) ![Figure 6](image2.png)

**Figure 5.** Exposure rate adjustment via image acquisition software for static blades  
**Figure 6.** Azimuthal averaged velocity vector distribution for V=14m/s

The free stream is in right direction. The front rotor is on the left while rear rotor is on the right. Take the free stream velocity 14m/s for an example, figures 6~8 show the comparison of the velocity vector, velocity magnitude and streamline between two methods.
Figure 7. Azimuthal averaged velocity magnitude distribution in EXP and CFD for V=14m/s

Figure 8. Azimuthal averaged streamline distribution in EXP and CFD for V=14m/s

It is obviously that both methods reveal similar velocity distribution trend. In figure 6, there is a high velocity range near the front rotor tip. The vector is along radial outward and downstream direction, which can be demonstrated in figure 8. It represents the streamline curvature and demonstrates the expansion of flow tube. Although figure 7 reveals that the maximum velocity magnitude (16m/s) in experiment is greater than that of CFD result (13m/s), but the distribution pattern is quite similar. Both demonstrate the low velocity domain near the middle of the rear rotor. Figure 7 shows that the CFD result has a more smooth velocity distribution than the experiment. Figure 8 shows that in experiment, the streamline is more uniformed near blade tip than middle of the blade. The curve change in experiment is more complicated than that in CFD result. This is because flow was treated as steady flow in CFD model with periodic boundary, which brings the even distributed azimuthal averaged performance.

4. Conclusions

The power and flow field prediction of the CRWT were predicted by CFD and experimental methods. Results show that the steady flow CFD models have a satisfied accuracy in power prediction. Referring to experimental power, for Usui’s test [11], the relative error of CFD is less than 1.56%; for authors’ experiment, the relative error less than 34%. CFD method predicts a quite close average tangential and axial velocity distribution on the cross sections to the measured data by Usui et al. For the authors’ experiment, both methods predict the similar velocity distribution near blade tip. The maximum velocity magnitude of PIV result is a bit bigger than CFD result. It is regarded as test rig obstruction effect. It shows that the CRWT has a complicated velocity distribution. A relative high velocity zone existed at the external of the front blade tip and a low velocity zone in the middle of the rear blade. Particularly, the velocity streamline curvature at the rear rotor is more uneven than the front rotor.
References

[1] Newman BG. Actuator-disc theory for vertical-axis wind turbines. *Journal of Wind Engineering and Industrial Aerodynamics* 1983; 15: 347-355.

[2] Riadh WH, Yoicu G, Yeu Y et al. Performance of a contra-rotating small wind energy converter, *ISRN Mechanical Engineering* 2011; Article ID 828739.

[3] Appa K. Energy innovation small grant (EISG) program (counter rotating wind turbine system) final report, California, US: EISG; 2002.

[4] S Lee, H. Kim, S Lee. Analysis of aerodynamic characteristics on a counter-rotating wind turbine. *Current Applied Physics* 2010; (10): 339-342.

[5] Jung S, No T, Ryu K. Aerodynamic performance prediction of a 30kW counter-rotating wind turbine system. *Renewable Energy* 2005; 30(5): 631-644.

[6] Shen WZ, Zakkam VAK, Sørensen JN et al. Analysis of counter-rotating wind turbines. *Journal of Physics: Conference Series* 2007; IOP Publishing, 75(1): 012003.

[7] Yuan W, Tian W, Ozbay A et al. An experimental study on the effects of relative rotation direction on the wake interferences among tandem wind turbines. *Science China Physics Mechanics & Astronomy* 2014; 57(5):935-949.

[8] Lee S, Son E, Lee S. Velocity interference in the rear rotor of a counter-rotating wind turbine. *Renewable Energy*, 54:235-240, 2013.

[9] Ozbay A, Tian W, Hu H. Experimental investigation on the wake characteristics and aeromechanics of dual-rotor wind turbines, *Journal of Engineering for Gas Turbines and Power*, 138, 2016

[10] Zhao X, Zhou P, Liang X. The aerodynamic coupling design and wind tunnel test of contra-rotating wind turbines, *Renewable Energy*, 146,1-8, 2020.

[11] Usui Y, Kubo K, Kanemoto T. Intelligent wind power unit with tandem wind rotors and armatures (optimization of front blade profile). *Journal of Energy and Power Engineering*, 6, 1791-1799, 2012.