Guidelines for the computational domain size on an urban-scale VAWT

Qiuyun Mo\textsuperscript{1,a}, Huisen Guan\textsuperscript{1,b,*}, Shengwen He\textsuperscript{1,c}, Yanyan Liu\textsuperscript{1,d}, Rongbin Guo\textsuperscript{1,e}

\textsuperscript{1}Department of Electrical and Mechanical Engineering, Guilin University of Electronic Technology, Guilin, 541004, China

\textsuperscript{a}email:6285745@qq.com,\textsuperscript{b}email:19012201033@mails.guet.edu.cn,\textsuperscript{c}email:61735027@qq.com,\textsuperscript{d}email:1539599580@qq.com,\textsuperscript{e}email:1648467460@qq.com

\*corresponding author:email:19012201033@mails.guet.edu.cn

Abstract: Focusing on the H-type vertical axis wind turbines (VAWTs) widely used in urban, the SST $\omega$ turbulence model was selected to establish a two-dimensional Computational Fluid Dynamics (CFD) model, the moment coefficient ($C_m$), and power coefficient ($C_p$) was taken as the evaluation indicators. The CFD model was used in different domain sizes. The simulation results under the sizes were used to investigate the effect of the computational domain size on the numerical simulation results of the aerodynamic performance of the VAWT. Furthermore, a distance from the turbine center to the domain inlet and outlet of 10D (D: diameter of the turbine) each, a domain width of 10D and a diameter of the rotating core of 1.5D are found to be safe choices to minimize the effects of blockage and uncertainty in the boundary conditions on the results. It provides a reference for the selection of the domain size when CFD simulation is implemented in a vertical axis turbine to get its prediction of the performance in the future accurately.

1. Introduction

In recent years, Darrieus (the most typical type is $\phi$ and H) vertical axis wind turbines (VAWTs) are widely used in special wind energy collection scenarios such as offshore, cities and remote villages because of their simple structure, high power coefficient, low costs of installation maintenance and their omni-directional capability for environments with frequent changes in wind direction\cite{1}, and the growing research enthusiasm of scholars has been received at home and abroad. However, in the last two to three decades, compared with horizontal axis wind turbines (HAWTs), there have been fewer studies, resulting in lower performance of VAWTs than HAWTs. In order to use the CFD model to predict performance more accurately in the research of VAWTs, high-resolution grids, appropriate computational domain size, and appropriate boundary condition settings have been becoming more and more essential to solve the flow details in time and space. And whether the flow field can be accurately solved or not is directly determined by it. Relevant scholars have conducted research on the size requirements of the computational domain in several typical environmental flows. For example, Franke J et al. \cite{2-4} formulated the optimal computational domain size selection criteria for flows in urban environments to minimize unnecessary influence of the domain boundary; For the flow of wind turbines, the study on the selection criteria of the computational domain size (for HWTs \cite{5-7} and VAWTs \cite{8-10}) is valuable, but the scope of application is more limited, and has not carried out a more systematic study on the size of the computational field \cite{5-10}, and it is impossible to obtain a reliable
minimum requirements for the size of the computational domain. In summary, formulating the best criteria for computational domain size has become the goal of this research.

2. Computational Methodology

2.1 VAWT geometrical and operational characteristics
A 3-bladed H-type VAWT equipped with the low speed symmetric NACA0015 airfoil, and the chord (C) is 0.2m; The diameter of wind wheel is 1m; the installation angle (β) of blades is 0°; and Swept area (AS) is 1m², taking into account the working area of the VAWT in the city should be as small as possible, the high ratio of height to diameter structure was selected. The ratio of height to diameter and the ratio of shaft radius to radius are h/2R=1, r/R=1/25 respectively, where R represents the installation radius of the blade is 0.5m, r represents the radius of the shaft; The free stream velocity (U∞) is 10m/s, the tip speed ratio (λ) as a constant is 2.2, and the turbine rotational velocity (ω) is 44 rad/s. These values can then be used as guidelines to ensure the accuracy of CFD results in case the turbine is operating at a moderate λ and the flow on the blades is not strongly separated. Schematic of the VAWT (top view) in Fig. 1.

![Fig. 1 Schematic of the VAWT (top view)](image)

2.2 Numerical Method

2.2.1 Fundamental governing equations
When the Mach number is less than 0.3, the flow can be treated as an incompressible flow. The flow velocity in the actual working condition of the object of this study is low, so it can be regarded as incompressible. The flow governing equations can be expressed as Eqn.1-2.

\[
\frac{\partial u_i}{\partial x_i} = 0
\]

\[
\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{1}{Re} \frac{\partial}{\partial x_j} \left( \frac{\partial u_i}{\partial x_j} \right)
\]

where \( u_i \) is the velocity component of the fluid along \( x_i \) direction, \( i=1,2 \), \( u_1 = u \) and \( u_2 = v \) represent the velocity components in horizontal and vertical directions respectively in case; \( t \) is time; \( p \) is pressure; Reynolds number \( Re = \rho U_{\infty} C / \mu \), \( \rho_{\infty} \) is the density of the free stream, \( U_{\infty} \) is the free stream velocity, \( C \) is the chord length of airfoil, and \( \mu \) is the dynamic viscosity coefficient.

2.2.2 SST \( k - \omega \) Turbulence Model
The transportation equations of turbulence kinetic energy \( k \) and specific dissipation rate \( \omega \) in SST \( k - \omega \) turbulence model\cite{11} are respectively defined by Eqn. 3-4.

\[
\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_i} (\rho k u_i) = \frac{\partial}{\partial x_j} \left( \Gamma_k \frac{\partial k}{\partial x_j} \right) + G_k - Y_k
\]
\[ \frac{\partial}{\partial t} (\rho \omega) + \frac{\partial}{\partial x_j} (\rho \omega u_j) = \frac{\partial}{\partial x_j} \left( \Gamma_\omega \frac{\partial \omega}{\partial x_j} \right) + G_\omega - Y_\omega + D_\omega \]  \hfill (4)

Where \( G_k \) is the velocity gradient of turbulence kinetic energy; \( G_\omega \) is the velocity gradient of specific dissipation rate; \( \Gamma_k \) and \( \Gamma_\omega \) are the effective diffusion coefficients of \( k \) and \( \omega \) respectively; \( Y_k \) and \( Y_\omega \) are the turbulence dissipation terms of \( k \) and \( \omega \) respectively; \( D_\omega \) is the cross diffusion term; \( \rho_c \) is the density of the free stream.

\( \Gamma_k \) and \( \Gamma_\omega \) can be obtained by Eqn. 5-6 respectively.

\[ \Gamma_k = \mu + \frac{\mu_t}{\sigma_k} \] \hfill (5)

\[ \Gamma_\omega = \mu + \frac{\mu_t}{\sigma_\omega} \] \hfill (6)

Where \( \sigma_k \) and \( \sigma_\omega \) are turbulent Prandtl numbers for \( k \) and \( \omega \) respectively.

### 2.3 CFD Modeling

The computational domain and boundary conditions are set according to the physical parameters of each part of the VAWT given above and the general setting rules in the numerical simulation calculation using Fluent. Computation domain is divided into static domain and rotation domain, the rotational radius of the turbine is \( R \). The distance from the turbine center to the domain inlet (\( D_i \), hereinafter referred to as the inlet size), the distance from the turbine center to the domain outlet (\( D_o \), hereinafter referred to as the outlet size), the width of the domain (\( W \), hereinafter referred to as the width size) and the diameter of the rotating core (\( D_c \), hereinafter referred to as the rotation domain size). Boundary conditions: inlet is defined as velocity-inlet boundary, outlet is defined as pressure-outlet boundary, the interface between rotating domain and stationary domain is defined as sliding grid interface boundary for data communication, and the blades and the two boundary AC and BD are defined as non-slip wall boundary, where both AC and BD are defined as symmetry boundary.

The specific computational model is shown in Fig. 2:

As shown in Fig. 3, the mesh is gradually refined along the region close to the blade by dividing different zones with different densities. This division method can capture the details of the flow field around the turbine, making the simulation more in line with the actual working conditions\(^{[12-13]}\), and avoiding the excessive number of meshes affecting the simulation efficiency.
The SST $k-\omega$ turbulence model is selected as the turbulence model, which has high calculation accuracy in the numerical computational of two-dimensional unsteady flow field \cite{21}. At the same time, the Pressure-based solution method is used in the calculation process. The pressure-velocity coupling was selected for the Coupled algorithm. The discrete format of time and space adopts the Second-order upwind scheme and the bounded 2nd order. The numerical calculation is performed based on the above settings. The power coefficient($C_p$), the moment coefficient($C_m$) and the tip speed ratio($\lambda$) are described by Eqn. 7-9. The 2D CFD simulations were performed using the settings above.

$$C_p = \frac{T_\omega}{0.5\rho A U_\infty^3}$$  \hspace{1cm} (7)

**2.4 The setting of computational domain size**

The SST $k-\omega$ turbulence model is selected as the turbulence model, which has high calculation accuracy in the numerical computational of two-dimensional unsteady flow field \cite{21}. At the same time, the Pressure-based solution method is used in the calculation process. The pressure-velocity coupling was selected for the Coupled algorithm. The discrete format of time and space adopts the Second-order upwind scheme and the bounded 2nd order. The numerical calculation is performed based on the above settings. The power coefficient($C_p$), the moment coefficient($C_m$) and the tip speed ratio($\lambda$) are described by Eqn. 7-9. The 2D CFD simulations were performed using the settings above.

$$C_p = \frac{T_\omega}{0.5\rho A U_\infty^3}$$  \hspace{1cm} (7)
By Eqn. 7-9, \( C_p \) can also be described as Eqn. 10.

\[
C_p = \lambda \rho C_m
\]  

(10)

Where \( T \) is the rotor torque, \( \omega \) is the turbine rotational velocity, 44 rad/s, \( U_{\infty} \) is the free stream velocity, 10 m/s, \( A_s \) is the swept area, 1 m\(^2\), \( R \) is the rotor radius, 0.5 m, \( \lambda \) is the tip speed ratio, 2.2, \( \rho \) is the air quality density, 1.225 kg/m\(^3\).

The Control Variable Method is used to study the sensitivity of the computational domain size. The test matrix is shown in Table 1, and all sizes are represented by the turbine diameter (D).

| Table 1 Test matrix for the sensitivity of the domain size |
|-----------------|-----|---|-----|-----------------|
| Parameter       | Di  | Do | W   | Domain size (L x W) |
| Inlet size(Di)  | 2.5 | 10 | 10  | 12.5 x 10        |
|                 | 7.5 | 10 | 10  | 17.5 x 10        |
|                 | 10  | 10 | 10  | 20 x 10          |
|                 | 12.5| 10 | 10  | 22.5 x 10        |
|                 | 15  | 10 | 10  | 25 x 10          |
| Outlet size(Do)| 5   | 10 | 10  | 10 x 10          |
|                 | 10  | 10 | 10  | 15 x 10          |
|                 | 15  | 10 | 10  | 20 x 10          |
|                 | 20  | 10 | 10  | 25 x 10          |
|                 | 25  | 10 | 10  | 30 x 10          |
| Width size(W)   | 5   | 10 | 10  | 5 x 15           |
|                 | 10  | 10 | 10  | 15 x 10          |
|                 | 15  | 10 | 10  | 15 x 15          |
|                 | 20  | 10 | 10  | 15 x 20          |
|                 | 25  | 10 | 10  | 15 x 25          |
| Rotation field size(Dc) | 5   | 10 | 10  | 1.5 x 15         |
|                 | 1.75| 10 | 10  | 15 x 10          |
|                 | 2   | 10 | 10  | 15 x 10          |
|                 | 2.25| 10 | 10  | 15 x 10          |

3. Results & Discussion (Sensitivity Analysis: effect of domain size)

The Realizable \( k - \varepsilon \) turbulence model with enhanced wall function is used to initialize the flow field by solving the steady-state URANS. The data sampling starts after the full development of the flow field. By monitoring the change of the lift coefficient and drag coefficient, when the flow field can be fully developed can be determined. As shown in Fig. 4 and Fig. 5, the time step is set to 0.001 s. After 0.8 s, the lift coefficient (\( C_L \)) and the drag coefficient (\( C_D \)) change periodically. It can be known that the numerical calculation results should be extracted after 0.8 s of flow field development.
3.1 Inlet size \((D_i)\)

The inlet size is set to 2.5, 5, 7.5, 10, 12.5, 15 times the diameter of the turbine respectively, and the numerical calculation is carried out at the azimuth angle of 0 to 360°.

The simulation results are shown in Fig. 6. The power coefficient obtained in the computational domain of \(D_i = 2.5D\) is more than 21% larger than that obtained in the computational domain of \(D_i = 10D\). For the computational domains of \(D_i = 10D, 12.5D,\) and \(15D\), the power coefficient \((C_p)\) decrease monotonically with the increase of \(D_i\), and the difference in the obtained results is smaller and smaller. The main reason for the above results is that the small computational domain limits the full development of the flow field, resulting in large deviation. When the inlet size of the computational domain is greater than 10D, the flow field has been fully developed. With the increase of the inlet size, the difference of the power coefficient is very small, only 0.01%. For the influence of moment coefficient \((C_m)\), it can be seen from Fig. 7 that in the azimuth range of 30° to 130°, the calculated results at \(D_i = 2.5D\) are quite different from those at other inlet sizes, and the obtained moment coefficient is higher than other cases. When \(D_i\) is greater than 5D, the obtained moment coefficient has negligible difference with the increase of \(D_i\). In summary, \(D_i = 10D\) can be regarded as the smallest safety dimension to avoid overestimate the performance of the VAWT.

3.2 Outlet size \((D_o)\)

The outlet size is set to 5, 10, 15, 20, 25 times the diameter of the turbine respectively. It can be concluded from Fig. 8 that the power coefficient obtained in the computational domain of \(D_o = 5D\) is 2.1% smaller than that obtained in the computational domain of \(D_o = 10D\). The power coefficient \((C_p)\) increases monotonically with the increase of \(D_o\) and the difference between the results is smaller and
smaller. When \( \text{Do} = 25\text{D} \), the difference is reduced to 0.2 \%. It can be known that the flow field is large enough to make the flow field fully developed. For the moment coefficient ( \( \text{C}_m \) ), it can be seen from Fig. 9 that when \( \text{Di} = 5\text{D} \) and \( \text{Do} \) larger than 5\text{D}, the moment coefficient obtained with the increase of outlet size will not be affected. In summary, \( \text{Do} = 10\text{D} \) can be regarded as the smallest safety dimension to avoid underestimating the performance of the VAWT.

3.3 Width size (W)

The width size is set to 5, 10, 15, 20, 25 times the diameter of the turbine respectively. As shown in Fig. 10, when \( \text{W} = 5\text{D} \), the power coefficient is greater than when \( \text{W} = 10\text{D} \), reaching 16.39 \%, the main reason is that too small width size limits the propagation of flow disturbance, which makes the numerical simulation results have large deviation. When \( \text{W} \) is greater than 10\text{D}, the power coefficient (\( \text{C}_p \)) decreases monotonically with the increase of \( \text{W} \) and the difference in the results is small, only about 0.6 \%, indicating that the flow field has been basically fully developed. For the influence of moment coefficient (\( \text{C}_m \)), it can be seen from Fig. 11 that in the azimuth range of 50° to 145°, the moment coefficient obtained when \( \text{W} = 5\text{D} \) is quite different from that under other inlet sizes, and it is higher. When \( \text{W} \) is greater than 10\text{D}, the moment coefficient obtained has negligible difference with the increase of \( \text{W} \).

3.4 Rotation domain size (Dc)

The rotation domain size is set to 1.5, 1.75, 2, 2.25 times the diameter of the turbine respectively. As shown in Fig. 12, when \( \text{Dc} = 1.5\text{D} \), the difference in the obtained power coefficient (\( \text{C}_p \)) is very small, which is less than 0.03 \%; For the influence of moment coefficient (\( \text{C}_m \)), as shown in Fig. 13, it shows the same law, when \( \text{Dc} = 1.5\text{D} \), the value of \( \text{Dc} \) continues to increase, and the difference in the calculation results of moment coefficient is small, and exists between 50° and 135° azimuth angles of the turbine, which is in good agreement with the above: the
difference in moment coefficient is common in the upwind area of the turbine.

![Graphs showing power and torque coefficients](image)

**Fig.12 Power coefficient**

**Fig.13 Torque coefficient**

4. Conclusions
The effects of blockage caused by the not large enough computational domain size will limit the full development of the flow field. When the computational domain size is large enough, the results are basically stable under different computational domain sizes, showing that the numerical simulation results are not correlated with the computational domain size. However, too large computational domain size will bring about high computational costs.

The current study based on ANSYS Fluent software, the moment coefficient ($C_m$) and power coefficient ($C_p$) are used as the evaluation indexes of the calculation results, and the two-dimensional CFD model of H-type VAWT under a moderate tip speed ratio operation is established. The sensitivity of the calculation domain size is studied and analyzed by using the Control Variable Method. The test matrix of the computational domain size is shown in Table 1.

The following conclusions were obtained for the investigated turbine:

1. When the inlet size, outlet size, width size and rotating domain size are not less than 10D, 10D, 10D and 1.5D, respectively, the selection of computational domain size is independent of the simulation results of $C_p$.

2. When the inlet size, outlet size, width size and rotating domain size are not less than 5D, 5D, 10D and 1.5D, respectively, the selection of computational domain size is independent of the simulation results of $C_m$.

3. When studying the effect of different domain sizes the numerical calculation results of $C_m$, we can obtain that the difference mainly exists in the upwind area of the turbine, so the flow field structure characteristics of the upwind of the turbine will be the focus of the following study.

Based on the settings of the geometric and operational characteristics of the studied turbine, the above conclusions are limited to the CFD simulation of the VAWT without dynamic stall at moderate tip speed ratio. For the CFD simulation of the VAWT with strong flow separation, it may require a larger computational domain size to minimize the effects on the simulation results. In the future research, the minimum requirement of the computational domain size should be extended to the CFD model of the VAWT with strong flow separation.

**Acknowledgments**
This investigation is one of the phased results of the National Natural Science Foundation of 《Study on Energy Efficiency Evaluation Mechanism of Wind Power System Based on Multi-scale and Multi-factor Synergy》 (51465010) and Guangxi Natural Science Foundation of 《Research on Key Technology of Vertical Axis Wind Power Generation System Based on Energy Collection and Multidisciplinary Optimization》 (2018GXNSFAA050077).
References

[1] Tian Haijiao, Wang Tielong, Wang Ying. An overview of the development of vertical axis wind turbines [J]. Applied Energy Technology, 2006, 11 (107): 22 - 27.

[2] Franke J, Hellsten A, Schluzen H, and Carissimo B, "Best practice guideline for the CFD simulation of flows in the urban environment," COST, Hamburg, Germany, 2007.

[3] Blocken B, "Computational Fluid Dynamics for urban physics: Importance, scales, possibilities, limitations and ten tips and tricks towards accurate and reliable simulations," Building and Environment, vol. 91, pp. 219-245, 2015.

[4] Tominaga Y, Mochida A, Yoshie R, Kataoka H, Nozu T, Yoshikawa M, and Shirasawa T, "AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings," Journal of Wind Engineering and Industrial Aerodynamics, vol. 96 (10-11), pp. 1749-1761, 2008.

[5] Sørensen NN, Michelsen JA, and Schreck S, "Navier-Stokes predictions of the NREL phase VI rotor in the NASA Ames 80 ft×120 ft wind tunnel," Wind Energy, vol. 5 (2-3), pp. 151-169, 2002.

[6] Sarlak H, Nishino T, Martínez-Tossas LA, Meneveau C, and Sørensen JN, "Assessment of blockage effects on the wake characteristics and power of wind turbines," Renewable Energy, vol. 93, pp. 340-352, 2016.

[7] Chen TY and Liou LR, "Blockage corrections in wind tunnel tests of small horizontal-axis wind turbines," Experimental Thermal and Fluid Science, vol. 35 (3), pp. 565-569, 2011.

[8] Balduzzi F, Bianchini A, Maleci R, Ferrara G, and Ferrari L, "Critical issues in the CFD simulation of Darrieus wind turbines," Renewable Energy, vol. 85, pp. 419-435, 2016.

[9] Ross I and Altman A, "Wind tunnel blockage corrections: review and application to Savonius vertical-axis wind turbines," Journal of Wind Engineering and Industrial Aerodynamics, vol. 99 (5), pp. 523-538, 2011.

[10] Biswas A, Gupta R, and Sharma KK, "Experimental investigation of overlap and blockage effects on three-bucket Savonius rotors," Wind Engineering, vol. 31 (5), pp. 363-368, 2007.

[11] Menter FR. Two-equation eddy-viscosity turbulence models for engineering applications[J]. Aiaa Journal, 1994, 32.

[12] Du Ping an. The basic principle of finite element mesh division [J]. Mechanical Design and Manufacturing, 2000,1: 34 - 36.

[13] Shi Zhongmin, Liu Mingshi, Guo Xiaoling. The effect of the computational domain on the numerical simulation of the flow around a cylinder[J]. China's water transport (in the next half and a half),2013 (7):40.

[14] Wang Y , Shen S, Li G, et al. Investigation on aerodynamic performance of vertical axis wind turbine with different series airfoil shapes[J]. Renewable Energy, 2018, 126(OCT.):801-818.