Design and development of CEWES software for complex environment wind engineering simulation

Fan Zhao, Shuang Wu, Chen Qi, Hanli Bai¹ and Xianzhuo Wang

Computational Aerodynamics Institute, China Aerodynamics Research and Development Center, Mianyang 621000, China

¹Email: starworks@carde.cn

Abstract. The design and development of the complex environment wind engineering simulation software CEWES was carried out, relying on the National Numerical Wind Tunnel Project (NNW). First, based on the characteristics of the physical problem that the software aims to solve, the requirements for the development of complex environment wind engineering simulation software are proposed, and three main modules of the software will be developed: structured grid flow field solver, unstructured grid flow field solvers modeling module of complex terrain and surface. Subsequently, the appropriate mathematical and physical model and numerical solution algorithm are selected for the flow field solver. The CEWES software uses the finite volume method for discretization with second-order accuracy, solves the RANS equations based on the SIMPLE algorithm, uses the k-ε turbulence model to solve the turbulence, and supports large-scale parallelism calculation. Third, the software design was carried out in accordance with the requirements of the CFD solution process and the modular program, focusing on the program architecture, data structure and subroutine interface design, and coding implementation based on the detailed design. Finally, the CEWES software was tested with typical examples. The test results of the calculation examples show that the software calculation results have good accuracy and large-scale parallel computing capabilities, and are suitable for wind engineering simulations in complex terrain environments.

1. Introduction

In recent decades, with advances in computer technology and computational fluid dynamics (CFD) algorithms, using numerical simulation methods to study wind engineering problems has become a major research method [1]. Computational wind engineering is widely used in wind resource assessment, aerodynamic performance analysis of wind turbines, and wind environment research in urban area [2].

At present, the numerical simulation research on wind engineering problems mainly adopts CFD simulation software for solving low-speed flow. The international mainstream commercial CFD software includes Ansys Fluent, Ansys CFX, Starccm+, PHOENICS, etc. The graphical interface of commercial software is easy to learn and has good computational robustness. It has occupied a major market share in universities and research institutes. Since the source code of commercial software is not open to users, users don't know anything about the program's solving methods, i.e., like a "black box" in which they can't see the situation inside. Including OpenFOAM, Code Saturne, OpenFVM, etc. [3-5], open source software is usually not easy to use, due to their low computational efficiency, and lack of engineering verification, thus rarely used in engineering problems. In recent years, cloud-based CAE software for wind engineering such as SimScale is developing rapidly, but the web browser...
based user interface is not so efficient [6]. At present, domestic CFD software has insufficient R&D strength and extremely low market share. Since there is no project and commercial promotion, the developed program is mainly for internal research and use, and there is still a long way to go to market promotion and engineering use. Recently, using machine learning of satellite images to predict wind field characteristics in large areas [7] also receives growing interests.

This paper adopts advanced software engineering development mode to develop a software system CEWES for numerical solution of complex environmental wind engineering problems. The software has the ability to efficiently process complex geographic information and generate CAD models, and provides important support for wind field simulation under real complex terrain conditions. The software provides structured and unstructured CFD solvers, and provides a powerful tool for complex terrain wind resource assessment, wind turbine aerodynamic performance analysis, urban block wind environment research, and low-speed complex flow simulation.

2. CEWES software general framework
Wind engineering software mainly uses numerical simulation to study the interaction between wind in the atmospheric boundary layer and human activities and man-made facilities. The issues related to energy and environment in the field of wind engineering include: 1) Wind energy evaluation and utilization related flow simulation, such as wind farm site selection evaluation, wind field analysis, wind turbine blade aerodynamic performance evaluation and optimization [8], etc.; 2) Environmental wind engineering issues such as environmental protection, ecology, and safety are mainly the diffusion and quality migration of pollutants in the wind environment. 3) The simulation requirements of energy power system. It includes the flow simulation problems of energy power generation facilities and power systems.

When the CEWES software was developed, in order to reconstruct the complex terrain surface, a module that can quickly generate a complex terrain surface digital model based on geographic information data (GIS) was developed. The generated surface model can be used for the surface grid generate.

For the solve of flow field, CEWES provides two solvers: structured and unstructured solvers. The structured solver is suitable for the flow with simple geometry, with high accuracy and calculation speed; for more complex geometry and ground surfaces, unstructured solver would be more convenient.

![Figure 1. The general framework of CEWES software.](image)

The module composition of CEWES software is shown in Figure 1. The core modules of this software are two flow field solvers and a GIS-based surface reconstruction module, which is used to generate the earth’s surface model or other models such as external wind turbines. Then, grid will be generated in these models to solve the flow field by structured or unstructured solvers, analyze the characteristics of the wind field, the aerodynamic characteristics of the wind turbine and the diffusion characteristics of pollutants.
3. CFD solver architecture and data structure

3.1. Software architecture design
The flow field solver of this software can be structured or unstructured. The program structure and simulation process of these two solvers are the same. The main difference is the data structure. This article takes the structured solver as example to introduce the program architecture and data structure of the software. The main part of the software includes a preprocessing program, a parallel solving kernel and a GUI module, as shown in Figure 2. The physical model module and post-processing module can be expanded outside the main body of the solver. The physical model module includes commonly used turbulence models, multi-reference system models, multi-phase flow models, etc., providing key calculation modules for complex turbulence field calculations, wind turbine rotating flow simulations, and pollutant diffusion and other multi-phase flow simulations.

![Figure 2. CFD solver architecture of CEWES.](image)

3.2. Data structure design
When CFD software performs large-scale calculations, it needs to use a large amount of memory space for data storage, including calculation parameters, 2D data, and 3D data. The data structure of the CFD calculation program plays a key role in the efficiency of software calculation and the convenience of program development and maintenance.

In the CEWES software design stage, all parameters and variables used in CFD numerical calculations are reasonably classified, and corresponding arrays and classes are established. Mainly include the following categories:

3.2.1. Calculation parameter class. The calculation parameter class includes solving control parameters, boundary condition parameters, etc. Solving control parameters are steady or unsteady, iterative steps, sub-relaxation factors, mixed format mixing factors, algebraic equations solving control parameters, etc. The flow parameters of working conditions are the incoming flow velocity, angle of attack, pressure reference point, turbulent incoming flow parameters, etc. These parameters are usually integer, real number or logical variable, with a small amount of data, which can be read directly through a parameter file and packaged in a class.

3.2.2. 2D type data. This type of data is mainly for data storage related to boundary surfaces, such as boundary conditions, docking relationships, wall distances, etc.
3.2.3. 3D type data. Including geometry type, flow field variable type and coefficient matrix type, etc., the data structure of these data is uniform, but due to different data usage scenarios, different classes are generally established according to the above classifications. The geometry category includes grid corner coordinates, grid center coordinates, cell volume, cell surface area, geometric interpolation factor, etc. Flow field variables include velocity, pressure, density, turbulent kinetic energy, dissipation rate, viscosity coefficient, etc. The coefficient matrix class is mainly the coefficients of algebraic equations constructed by the discrete NS equations, and some temporary coefficient matrices constructed for the iterative solution of the algebraic equations. The structure solver is usually a diagonal band-shaped sparse matrix, while the present software structure solver forms a seven-diagonal matrix.

This article takes structured grid solver as an example. The storage of grid or flow field variables often adopts two data structures: high-dimensional array or one-dimensional array. The data structure of the high-dimensional array is based on the three directions of I, J, K and the grid block number to construct a 4-dimensional array, the data type is U (I, J, K, NB). The data structure of the one-dimensional array is to store the flow field variables in a one-dimensional array, and convert the index number of the one-dimensional array corresponding to different I, J, and K values under each grid block through the index number conversion function. The data type is U (IJK), where IJK = function IJKID (I, J, K, NB).

Using high-dimensional arrays to store variables is relatively easy to write code, which is suitable for the study of simple problems, but the memory usage and data access speed of high-dimensional arrays is not high. The one-dimensional array has high storage utilization and fast access speed, which is suitable for engineering calculations. This software is mainly oriented to practical engineering applications of CFD, and will use a one-dimensional array to store data such as flow field variables.

Assuming that the current calculation grid unit is point P, the adjacent units in the six directions up, down, east, west, south, and north are represented by T, B, E, W, S, and N, as shown in Figure 3. The calculation method of the index numbers of different units in the grid block is shown on the right side of Figure 3.

![Diagram of grid storage location and compass notation](image)

**Figure 3.** The mark and index number of the grid cell and its neighboring cells.

4. Mathematical and physical model of fluid mechanics simulation

4.1. Governing equations

In fluid mechanics, low-speed flow is usually treated as incompressible flow, and density changes (such as natural convection caused by temperature) are only considered in a few special cases. The general formula [9] of Reynolds average Navier-Stokes equation can be expressed as formula (1):

$$\frac{\partial \rho \psi}{\partial t} + \text{div}(\rho \nabla \psi - \Gamma_\psi \cdot \text{grad} \psi) = q_\psi$$  \hspace{1cm} (1)

For continuous equations:

$$\psi = 1, \quad \Gamma_\psi = 0, \quad q_\psi = 0$$  \hspace{1cm} (2)
For the momentum equation in the $x$ direction:

$$\psi = u, \quad \Gamma_\psi = \mu_{eff}, \quad q_\psi = -\frac{\partial P}{\partial x} + \operatorname{div}(\mu_{eff} \frac{\partial V}{\partial x})$$

(3)

The momentum equations in the $y$ and $z$ directions have similar expressions.

For the energy equation:

$$\psi = T, \quad \Gamma_\psi = \frac{k}{C_p}, \quad q_\psi = \frac{S_h}{C_p}$$

(4)

For the turbulent kinetic energy equation:

$$\psi = k, \quad \Gamma_\psi = \mu + \frac{\mu_t}{\sigma_k}, \quad q_\psi = G_k - \rho \varepsilon$$

(5)

The turbulent dissipation rate equation:

$$\psi = \varepsilon, \quad \Gamma_\psi = \frac{\mu_{eff}}{\sigma_\varepsilon}, \quad q_\psi = \frac{\varepsilon}{k} (C_1 G_k - C_2 \rho \varepsilon)$$

(6)

Ideal gas state equation (when density changes need to be considered):

$$p = \rho RT$$

(7)

4.2. Numerical discrete method

For the control equations of low-speed incompressible flow, general solving algorithms include artificial compressibility method, projection method, pressure Poisson equation method and pressure correction algorithm. Among them, the pressure correction algorithm has been widely used in commercial software and open source software since it was proposed in 1972 [10]. It has become the main algorithm for solving incompressible flow fields. This software will also use SIMPLE algorithms during development.

The pressure correction method for solving incompressible viscous flow was SIMPLE (semi-implicit method for pressure-linked equations) algorithm which was widely used to solve incompressible flow.

The basic procedure of the pressure correction algorithm is as follows:

1. Give the initial value $P^*$ of the pressure field, start iteration;
2. Substitute $P^*$ into the momentum equation to solve $u$, $v$, $w$, and obtain the velocity values $u^*$, $v^*$, $w^*$;
3. $u^*$, $v^*$, $w^*$ are obtained by solving a given initial pressure, and cannot accurately satisfy the continuity equation. Construct a pressure correction equation with the continuity equation and solve it to get the pressure correction value $P'$. The corrected pressure $P$ is:

$$P = P^* + P'$$

(8)

Substitute the pressure correction value $P'$ into the momentum equation, and solve the velocity correction value $u'$, $v'$, $w'$, then the corrected velocity value is:

$$u = u^* + u', \quad v = v^* + v', \quad w = w^* + w'$$

(9)

Take the pressure correction value $P'$ obtained by equation (8) as the new $P^*$, and repeat steps (2) and (3) until the convergence condition of the continuity equation is satisfied.

5. Calculation example verification

5.1. Surface reconstruction module

When studying the characteristics of the micro-scale wind field near the surface, the surface contour has a great influence on the characteristics of the wind field. Constructing a fine three-dimensional digital model of the surface contour can better simulate and predict the characteristics of the near-
surface wind field. Generally, the surface information is obtained by the geographic information system, which is divided into two types: satellite scan data and surveying and mapping data, but they cannot be directly used for grid generation in CFD simulation. CEWES software provides a set of rapid surface modeling modules based on GIS information, which can quickly convert surface GIS information into digital models or surface grids required for CFD simulation. Figure 4 shows the effect of surface DEM data display in a local area of Weifang City, and Figure 5 shows the reconstructed surface after surface triangulation based on DEM data. The triangulated surface can be directly exported as a surface in STL format, for grid generation in the mesh software.

Figure 4. Surface DEM data in a local area of Weifang City.  
Figure 5. Triangle mesh reconstruction in a local area of Weifang City.

5.2. 0012 airfoil calculation example

The wing is a key aerodynamic part of an aircraft. The lift of a fixed-wing aircraft is mainly generated by the wing. The drag generated by the wing also contributes a certain proportion to the total aircraft resistance. The wing shape and wing design are thus important for aircraft design. When the airfoil is at a large angle of attack, due to the back pressure gradient of the trailing edge, as the angle of attack increases, the boundary layer in the leeward zone of the wing separates and vortex shedding occurs, resulting in a decrease in lift and a stall. Therefore, the calculation of the aerodynamic force of the airfoil and the flow around the wing is very important, and relevant scholars have made numerous efforts on this topic [11,12].

The calculation of NACA0012 airfoil is mainly to verify the capability of the software to calculate low-speed aerodynamics. The mesh of the airfoil is shown in Figure 6, where the viscous support layer close to the wall is processed by the wall function method.

Calculation conditions: the incoming flow velocity is 44.2 m/s, and the Reynolds number is 2.5×10^6.

Figure 7(a) shows the comparison of the calculated and experimental lift coefficient curves, and Figure 7(b) shows the comparison between the calculated and experimental results of NACA0012 airfoil surface pressure distribution at an angle of attack of 11.9° [13]. It can be seen that the calculation results of the linear section lift curve are in good agreement with the experiment, and the stall section still agrees with the experiment before the lift coefficient does not decrease.

Figure 6. The grid of NACA0012 airfoil.
5.3. Wind turbine example

This paper tested a horizontal axis wind turbine example, the incoming wind speed was 8m/s, and the wind turbine rotation speed was 16.362 rpm. The calculation uses the multi-reference frame method (MRF) to simulate the rotation of the wind turbine, the calculation grid scale is about 8 million, and the MPI parallel calculation is used.

From the numerical simulation results, it can be seen that the flow field around the wind turbine blade and the wake flow field behind the blade are reasonable, which verifies that the CEWES software can be applied to the simulation analysis of the aerodynamic characteristics of the wind turbine on a larger scale.

Figure 7. NACA0012 airfoil lift coefficient and surface pressure coefficient.

Figure 8. The streamlines of wind turbine.

Figure 9. U velocity spatial distribution at 20m behind the wind turbine.

Figure 8 shows the spatial flow line around the wind turbine. It can be seen from the figure that before the incoming flow arrives at the wind turbine, its flow direction does not change significantly. After encountering the wind turbine blade, the streamline is obviously bent due to the blade's obstruction. It is the action of flow around the blade that produces lift on the blade and pulls the blade to rotate. Figure 9 shows the x-direction (axial) velocity distribution at 20m behind the blade. After the
incoming flow passes through the wind turbine, its velocity distribution is greatly different from the uniform distribution at the inlet. The velocity in the middle area directly behind the rotation area of the blade is slightly smaller than that in the surrounding area.

6. Conclusions
1) This paper describes a framework of wind engineering simulation software CEWES, which has clear processes and good data structure, and takes into account the efficiency of program operation and the convenience of software development and maintenance;
2) A surface reconstruction module was developed in the CEWES software, which can easily reconstruct the CAD model required for CFD simulation based on geographic information, and can finely simulate the wind field characteristics under complex terrain conditions. These functions are not provided in the current commercial softwares;
3) The numerical simulation results of the 0012 airfoil calculation example show that the CEWES software developed in this paper can better predict the airfoil aerodynamic characteristics, especially the stall characteristics at high angles of attack, and verify the reliability of the software for solving the flow field. This is because we improved the standard k-ε model by modifying the turbulent kinetic energy limit;
4) The simulation example of the wind turbine flow field shows that the CEWES software has multi-block grid and parallel computing functions, can carry out large-scale parallel computing of complex geometry, and has good engineering prediction accuracy.

References
[1] Blocken B 2014 50 years of Computational Wind Engineering: Past, present and future J. Wind Eng. Ind. Aerodyn. 129
[2] He D X 2006 Wind Engineering and Industrial Aerodynamics[M] National Defense Industry Press (in Chinese)
[3] Hargreaves D. M. Wright N. G. 2007 On the use of the k–ε model in commercial CFD software to model the neutral atmospheric boundary layer J. Wind Eng. Ind. Aerodyn. 5 95
[4] https://www.code-saturne.org/cms/ Aug 20, 2020
[5] https://www.openfoam.com/ Aug 20, 2020
[6] https://www.simscale.com/wind-engineering/ Aug 20, 2020
[7] Nezhad M M, Heydari A, Groppi D 2020 Wind source potential assessment using Sentinel 1 satellite and a new forecasting model based on machine learning: A case study Sardinia islands Renewable Energy 155
[8] Shourangiz-Haghighi A, Haghnegahdar M A, Wang L 2020 State of the Art in the Optimisation of Wind Turbine Performance Using CFD Archives of Computational Methods in Engineering 27
[9] TAO Wenquan 2001 Numerical Heat transfer(Second Edition) [M]. Xi’an Jiaotong University Press (in Chinese)
[10] Patankar S V 1972 A calculation procedure for heat, mass and momentum transfer in three-dimensional parabolic flows International Journal of Heat and Mass Transfer 10 15
[11] Tulapurkara E G. 1997 Turbulence models for the computation of flow past airplanes Prog Aerospace Sci 33
[12] Rogers S E.et al. 1992 Efficient simulation of incompressible viscous flow over single and multi-element airfoil AIAA-92-0405
[13] Chorin A J. 1997 A numerical Method for Solving Incompressible Viscous Flow Problem J. Comp. Phys. 2