Microclimate Regulation in Glass Greenhouses

Xinping Lv

{15854585772@163.com}

No. 1 Daxue Road, Shandong Normal University, Changqing District, Jinan City, Shandong Province
China

Abstract. The yield of greenhouse crops is affected by a variety of climatic factors, including temperature, humidity, and wind speed. Among them, suitable temperature and wind speed are crucial for plant growth. In this paper, a mathematical model is established to optimize the greenhouse fan, so that it can obtain the appropriate wind speed and temperature, and improve its uniformity, which solves the important problems in the current design of glass greenhouse.

Keywords: k-ε turbulence model; Energy conservation equation; Finite volume method; CFD simulation optimization; Optimization model

1 Introduction

The yield of greenhouse crops is affected by various climate factors, including temperature, humidity, and wind speed [1]. Among them, suitable temperature and wind speed are crucial for plant growth [2].This study aims to optimize greenhouse microclimate control systems through mathematical modeling and numerical analysis to enhance crop yield and environmental sustainability. The preliminary work is based on the theories of fluid dynamics and thermodynamics, relying on k-ε turbulence model and energy conservation equation, the wind speed and temperature field models of the glass greenhouse are established. Then, the finite volume method commonly used in fluid dynamics CFD is used for numerical calculation and simulation of the model, and the position and wind speed of the greenhouse fan are optimized. To obtain the appropriate wind speed and temperature in the glass greenhouse, and improve its uniformity, in order to accurately predict and simulate the air flow dynamics and temperature distribution in the greenhouse.

2 Main body

2.1 Modeling of fluid mechanics and heat transfer

a.Hydrodynamic model

Continuity equation:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho v) = 0 \quad \text{.................................(1)}$$

MSIEID 2023, December 08-10, Guangzhou, People's Republic of China

DOI 10.4108/eai.8-12-2023.2344777

Momentum equation:

$$\frac{\partial(\rho v)}{\partial t}+\nabla\cdot(\rho vv)=-\nabla P+\nabla\cdot\tau+\rho g \quad \cdots\cdots\cdots\cdots\cdots\cdots(2)$$

Energy equation:

$$\frac{\partial(\rho E)}{\partial t}+\nabla\cdot(\rho Ev)=-\nabla\cdot q+\nabla\cdot(\tau\cdot v)+\rho v\cdot g \quad \cdots\cdots\cdots\cdots\cdots\cdots(3)$$

Where ρ represents the density of the fluid, t represents time, v represents the velocity vector, ∇ represents the gradient operator, P represents the pressure, represents the stress tensor, g represents the gravitational acceleration, E represents the total energy per unit mass, and q represents the heat flux.

b.Heat transfer model

Heat transfer models are used to describe how and at what rate heat is transferred through an object or fluid. It can be described by different equations and models according to the specific properties and conditions of the problem.

(1) Heat conduction model: describes the process of heat conduction through a solid or liquid. According to Fourier's law of heat conduction, the heat conduction equation can be used to describe:

$$\frac{\partial T}{\partial t}=\alpha\nabla^2 T \quad \cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots(4)$$

Where T is temperature, T is time, α is the thermal diffusion coefficient, and∇2 is the Laplacian operator.

(2) Heat convection model: describes the process of heat transfer through fluid convection. For natural convection, dimensionless parameters such as Grashofnumber and Prandt1number can be used to estimate the heat transfer rate. For forced convection, parameters such as Reynoldsnumber and Prandtl number can be used to describe thermal convection heat transfer.

(3) Radiative heat transfer model: describes the process of heat transfer through radiation, such as heat radiation, radiation absorption and radiation transfer. Radiative heat transfer can be described using the Stefan-Boltzmann law and Kirchhoff's law.

(4) Coupling of heat transfer models: In some cases, heat transfer can involve the process of heat convection and heat conduction at the same time. In this case, it is necessary to consider the heat conduction equation and the fluid mechanics equation at the same time, and to solve the coupling.

Heat transfer models also usually need to be solved by combining boundary conditions and initial conditions. The specific heat transfer model formulas and equations will vary from problem to problem, and in practical applications there may be more complex models and equations to take other factors into account, such as phase transitions, turbulence, etc. However, heat conduction and radiation heat transfer are ignored in this paper, so the problem is solved by establishing a heat convection model.

2.2 Setting of boundary conditions and initial conditions

a.Boundary condition

The equations of fluid mechanics, in particular the Navier-Stokes equations, describe the motion of fluids. To understand these equations, one must specify the boundary conditions, which describe the behavior of the fluid at the boundary. The following are common types of boundary conditions in fluid mechanics:

1. No-slip condition:

The velocity of the fluid at the boundary of the solid is equal to the velocity of the solid surface. For a stationary solid surface, this means that the velocity of the fluid at the boundary is zero.

2. Slip condition:

The fluid is allowed to have a non-zero velocity at the boundary, which may be applicable in microfluidics or high speed gas flows.

3.Dirichlet condition for pressure:

Specify a fixed pressure value on the boundary. This is common when simulating open channel flows.

4.Neumann condition for pressure:

The pressure gradient on the boundary is specified, not the pressure itself. This is used for the outflow boundary, where the pressure change may be unknown, but it can be assumed that there is no positive pressure gradient.

5.Dirichlet condition for temperature:

The temperature on the boundary is set to a fixed value that applies to the wall or heating/cooling surface of the heat exchanger.

6.Neumann condition for temperature:

Specify the heat flux at the boundary, which is useful for specifying the heating or cooling rate of the wall.

7.Convection condition:

The heat exchange at the boundary is determined by convection, which involves the temperature of the ambient fluid and the convective heat transfer coefficient.

8. Radiation boundary conditions:

When it comes to thermal radiation, the temperature or heat flux at the boundary is affected by the radiation effect.

9. Periodic condition:

Applied at the inlet and outlet of the fluid, assuming that the flow pattern at these two boundaries is the same, which is useful when simulating periodic flows or geometric structures.[3]

10.Free surface boundary conditions:

For a free surface in contact with a liquid and a gas, the boundary conditions should be able to describe the shape and dynamics of the surface.

11. Symmetric boundary conditions:

When the physical problem has symmetry, symmetric boundary conditions can be applied on the plane of symmetry, assuming that the direction velocity is zero and the shear stress along the plane is zero.

According to the requirement of this problem, we set the boundary conditions as no slip boundary conditions and fixed pressure boundary conditions.

b.Setting of initial conditions

In a heat transfer model, the initial condition is the initial temperature distribution at various points in the object or fluid at the beginning of the simulation process. The setting of initial conditions has an important influence on the accuracy and stability of simulation results.

1.Uniform initial temperature: Assume that the entire object or fluid has the same temperature at the initial moment, for example:

$$T(x,y,z,t=0)=T0 \qquad (5)$$

2.Non-uniform initial temperature: the initial temperature at different points given according to the actual situation, such as:

$$T(x,y,z,t=0)=f(x,y,z) \qquad (6)$$

In this case, we're going to set the initial condition to be a uniform initial temperature.

2.3 CFD simulation

a.Model construction and grid division

In this study, the relevant parameters of the geometric model were determined according to the size of the test plastic greenhouse. The interior space of the greenhouse is taken as the calculation domain, and the 3D solid model is established by ICEM software. The northwest corner of the greenhouse is O(0,0,0) as the model origin, the due south direction is X axis forward, the due east direction is Z axis forward, and the vertical upward direction is Y axis forward. The calculation domain is divided into unstructured grids, and in order to improve the mesh quality, local encryption processing is carried out on the pipeline outlet and the area near the wall.[4] The number of nodes in the divided grid model is 20,000, and the total number of grid units is 130,000. The grid quality is controlled according to the EquiAngleSkewness standard, which meets the calculation requirements, and subsequent simulation can be performed.

b.CFD model equation

The basic governing equations of CFD numerical simulation include continuity equation, momentum equation and energy equation.

In this study, the standard k-ε turbulence model with good convergence and high calculation accuracy is used to solve the airflow flow process in the greenhouse.The turbulence in the near-wall area of greenhouses is not sufficiently developed, so the standard wall function is introduced to deal with the flow in the near-wall area.

c.Boundary condition

The heat exchange inside the greenhouse is dominated by convection, and the boundary between the greenhouse envelope and the opposite side is set as the wall, in which the greenhouse glass structure is set as the convection boundary condition. The fan boundary is set to the speed inlet condition 2m/s and the temperature is 40℃.

d.Data calculation

In the test process, the external temperature is stable and the fan running state is stable, so the environment can be considered to be in steady state. The solution process is selected based on the pressure solver setup and ignores the air gravity. The governing equations are discretized based on finite volume. In the solution, the convergence standard of energy residual is set to 10-6, radiation residual is set to 10-6, radiation residual is set to 10-5, and the convergence standard of other variables is set to 10-3.

Fluent was used to compute and solve the problem. Standard wall function was adopted, second-order upwind discrete scheme was adopted, and the coupling of pressure momentum and velocity momentum was discretized by SIMPLE algorithm. The residual convergence criterion is less than 10-6 for the energy equation and less than 10-3 for the continuity equation and k-ε equation. The calculation time step is 1s, and the calculation method is 250 steady state steps + 2000 unsteady state steps. One of the unsteady state steps is iterated for 5 times, and the numerical calculation result is passed.[5]

2.4 Problem result

1.The model and grid division of question 1 are realized in Ansys Fluent software.analyzed as shown in Figure1 This study determines the relevant parameters of the geometric model according to the size of the indoor greenhouse. Taking the northwest corner of the greenhouse as the model origin O(0,0,0), due south direction is X axis forward, due east direction is Z axis forward, and vertical upward direction is Y axis forward.

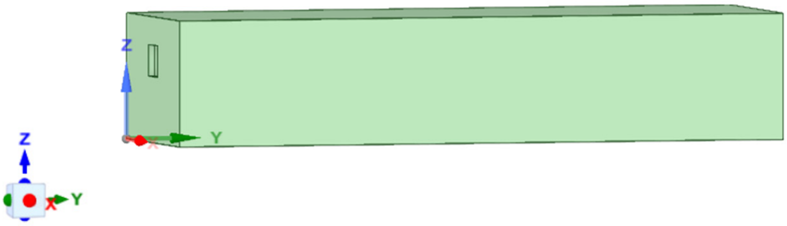

Fig. 1. Model establishment

2. The divided grid model is shown in Figure2, where the number of nodes is 20,000 and the total number of grid units is 130,000. The grid quality is controlled according to

EquiAngleSkewness standard, which meets the calculation requirements, and subsequent simulation can be carried out.

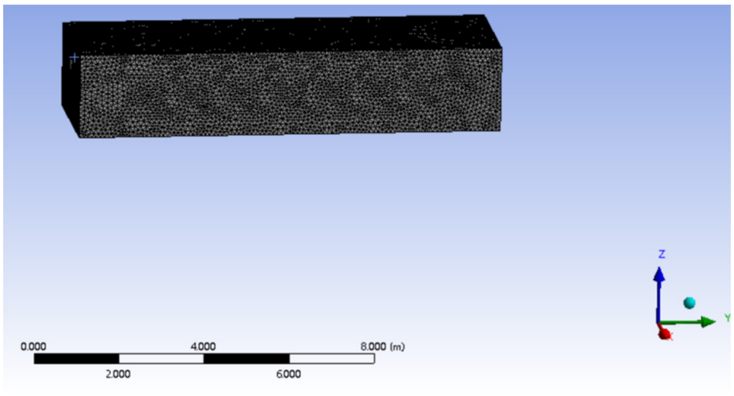

Fig. 2. Grid processing

3. The skewness index of the model is analyzed as shown in Figure3 and Figure4, with values below 0.8 and above 0.2.

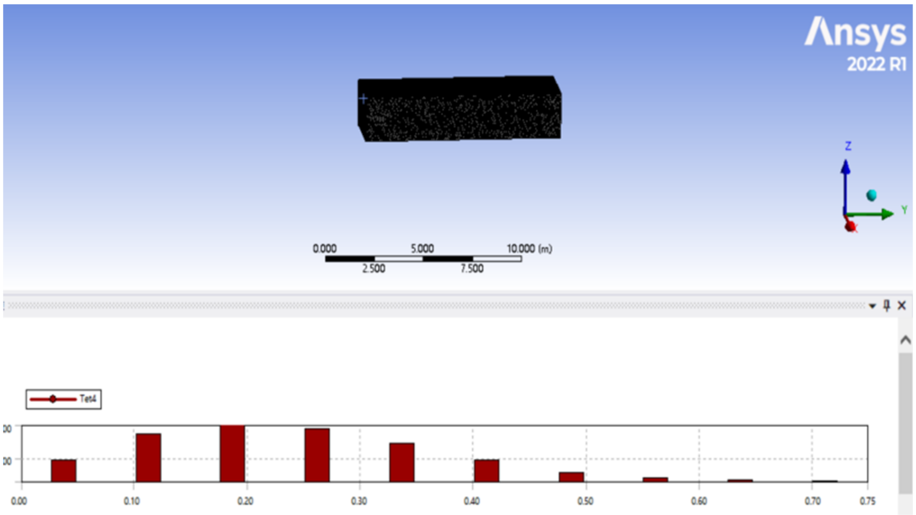

Fig. 3. Grid quality index 1

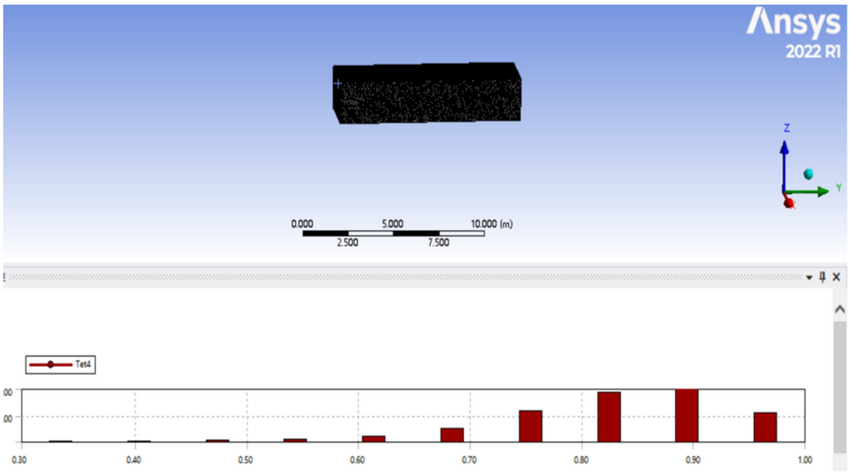

Fig. 4. Grid quality index 2

4. Visualization of the model was carried out to provide intuitive information, analyzed as shown in Figure5 and the distribution of wind speed and temperature was visualized at a section at a height of 0.5m. The test results showed that: in the closed greenhouse without crops, the average temperature at the crop canopy (0.5m) was about 24.8°C, and the average wind speed was about 0.48m/s.

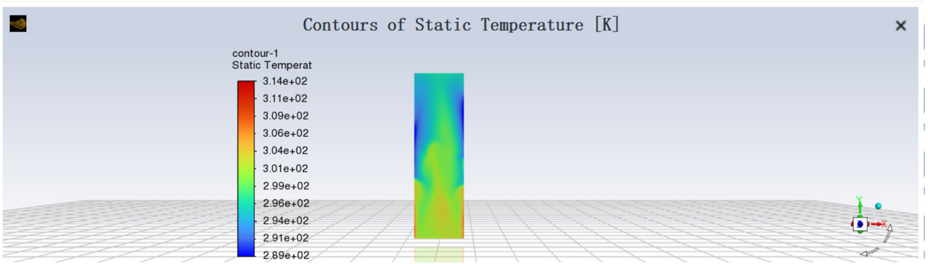

Fig. 5. Distribution of temperature calculation results

3 Conclusions

1. By placing the fan in the greenhouse mirror and making the wind direction opposite, the distribution uniformity of the air flow in the greenhouse can be effectively improved, so an even number of fans can be placed as far as possible.

2. The position of the fan should take into account the hot spot and dead Angle position in the greenhouse, and the installation of the fan in these places to achieve effective ventilation and circulation of air can effectively improve the uniformity of the wind.[6]

3. Excessive wind speed will cause a slight increase in temperature near the fan and in some areas inside the greenhouse. This may be needed to sustain crop growth

The stable temperature environment causes certain effects.

4. The closer the fan is to the median of the greenhouse height (1 meter), the more it helps to create more uniform temperature conditions in the crop area.

5. If a scientific ventilation scheme is used, the average amount of air required for vegetable greenhouses per hour is between 1 and 2.5 cubic meters. According to the required air volume, the corresponding size of the fan can be selected to achieve the best balance of ventilation effect and energy saving. A smaller fan can be used for a smaller greenhouse area, and a larger fan can be used for a larger area.

References

[1] Singh M C, Singh J P, Pandey S K, et al. Factors affecting the performance of greenhouse cucumber cultivation-a review[J]. International Journal of Current Microbiology and Applied Sciences, 2017, 6(10): 2304-2323.

[2] Liu Y, Li D, Wan S, et al. A long short-term memory-based model for greenhouse climate prediction[J]. International Journal of Intelligent Systems, 2022, 37(1): 135-151.

[3] Katsoulas N,Sapounas A ,De Zwart F,et al.Reducing ventilation requirements in semi-closed greenhouses increases water use efficiency[J].Agricultural Water Management, 2015, 156:90-99.DOI:10.1016/j.agwat.2015.04.003.

[4] Zhang Chen; Fang Hui; Cheng Ruifeng; Yang Qichang; Wei Xiaoran; Wu Chenyong. Aerodynamic study of lettuce based on wind tunnel system [J]. Journal of China Agricultural University,2019,24(12):96-103.

[5]]Wan Min; Yang Wei; Liu Z Q. Microenvironment simulation and crop transpiration analysis in solar greenhouses with different ventilation modes [J/OL]. Transactions of the Chinese Society for Agricultural Machinery,1-22[2023-11-26]

[6] Hong Y J. CFD analysis and experimental study on temperature field of large shoulder high glass greenhouse under different ventilation modes [D]. Jiangsu University,2018.