PIV Measurement and Simulation of Building Thermal Plume under No-wind Conditions

Tiantian Zhang1, Dongliang Han1, Yufei Tan1, and Jing Liu

1Key Laboratory of Cold Region Urban and Rural Human Settlement Environment Science and Technology Ministry of Industry and Information Technology, School of Architecture, Harbin Institute of Technology, China

Abstract. To investigate the buoyancy characteristics of building façades, this study deals with experimental and numerical investigations on building façade flow and temperature field in different façade temperature conditions with different height-aspect ratio of building facade. PIV measurements are conducted in a static wind platform for obtaining the mean velocity, the vertical velocity and the horizontal velocity around the building surface. To simulate the plume characteristics accurately, this study adopt three numerical model and compared the numerical results with the PIV experimental data. The result indicated that the simulation result of the RNG k-ε model best agreement with experimental data. Analysis of numerical results indicated that the wind speed kept increasing in the vertical direction, while the wind speed first increased and then decreased in the horizontal direction. The air temperature tended to be constant in the vertical direction, but air temperature first drooped sharply and tended to be constant in the horizontal direction. As the heat flow density increases, the building facade plume strength increases. When the wall heat flow increases from 100W/m² to 200 W/m², the maximum velocity of the wall increases by 1.6 m/s and the temperature increases by 15 ℃.

1 Introduction

At present, with the accelerating process of urbanization worldwide, global climate change is gradually intensified, the contradiction is becoming more and more prominent between the urban rapid development and urban environmental problems. Frequent calm weather, the accumulation of pollutants in urban areas and the increasingly serious heat island effect have brought severe challenges to urban air quality and thermal comfort. Therefore, studying the characteristics of natural convection (thermal plume) caused by temperature difference at urban building facades under no-wind environment is helpful to improve the thermal environment. In recent years, a lot of researches [1-6] have been done on how to improve the thermal-wind environment under no-wind conditions. And, previous scholars mainly studied outdoor thermal-wind environment and proposed optimization methods on the following five aspects: building height, urban ventilation corridor, urban green rate, urban form and urban road layout. Few studies have focused on building scale, especially to thermal plumes on building surfaces. In this paper, the thermal plume characteristics of building facades under different wall heat flow densities were studied by combining experimental and simulation methods, and PIV experiment and CFD simulation were carried out under on-wind environment. The aim of this paper is to verify the accuracy of three different turbulence models: RNG k-ε, Standard k-ε and Low Reynolds Number model using PIV experiments. Secondly, experimental and simulation results are used to study the thermal plume characteristics under different wall heat flow densities. In addition, the research results of this study are expected to provide theoretical basis for the emission of urban heat source and pollution source in a windless environment.

2 Experiment measurement

2.1 Experiment setup

The purpose of this paper is to study wind-thermal environment near the building facade under no-wind conditions. In order to create a calm experimental environment, a 1.2m×1.2m×1.2m calm wind experimental device is built, as presented in Fig.1. The experimental device is divided into two parts, on the upper part of the space (high: 80cm), The size of solid wood block 20cm×10cm×40cm was used as a single building model (Scale 1:100) to conduct heat plume experiment, and the electrothermal film was pasted on the building facade. At the lower part of the space (high: 40cm), tracer particles are sprayed into the space to eliminate the influence of particles on the flow field in the upper experimental space. Because the jet particles have a certain initial velocity, the flow field of natural convection on vertical surface will be significantly affected. During the test, the tracer particles were first sprayed into a lower storage space. After the lower space is filled with tracer gas and the electrothermal film was pasted on the building facade to simulate the heating of the building facade. At the lower part of the space (high: 40cm), tracer particles are sprayed into the space to eliminate the influence of particles on the flow field in the upper experimental space. The experimental device is divided into two parts, on the upper part of the space (high: 80cm), The size of solid wood block 20cm×10cm×40cm was used as a single building model (Scale 1:100) to conduct heat plume experiment, and the electrothermal film was pasted on the building facade to simulate the heating of the building facade. After the lower space is filled with tracer gas and the electrothermal film works stably, the vent opens. Tracer gas enters the upper space...
due to buoyancy and moves up along the surface of the building model. When the tracer gas flow reaches a stable state, the PIV system is activated.

Fig.1. The schematic diagram of experimental apparatus.

2.2 PIV setup and post-processing

A high-resolution PIV system was adopted in this experiment, including hardware and software, as shown in Fig 2. Hardware includes CCD camera with a 50-mm f/1.8D lens from the AF Nikko, particle generator, synchronizer, laser generator (532-nm wavelength and 135-mJ beam intensity), tracer particle conveyor and computer. The software part mainly includes high-speed camera control system, synchronizer control system and particle image analysis and calculation system.

Fig. 2. 2D-PIV measurement system.

3 Numerical simulation

3.1 Physical model development

The basic equation governing the flow law of fluid is derived from three conservation laws of physics, namely the conservation of energy, momentum and mass. Such basic equation is called the basic governing equation of fluid. There are three fundamental conservation laws that need to be met in computational fluid analysis.

\[
\frac{\partial}{\partial t} \rho + \nabla \cdot (\rho \mathbf{u}) = 0 
\]

\[
\frac{\partial (\rho \mathbf{u})}{\partial t} + \nabla \cdot (\rho \mathbf{u} \mathbf{u}) = -\nabla p + \nabla \cdot \tau + \rho \mathbf{F} 
\]

\[
\frac{\partial (\rho E)}{\partial t} + \nabla \cdot (\rho E \mathbf{u}) = \nabla \cdot (\tau \mathbf{u}) + \rho \mathbf{F} \cdot \mathbf{u} + q 
\]

Where, \( \mathbf{u} \) is the velocity component in the i direction, \( \rho \) is fluid density (kg/m³), \( T \) is temperature (°C), \( c_p \) is specific heat capacity, \( S_T \) is Viscous dissipation term (J).

3.2 Turbulence model

At present, the two-equation model based on Reynolds average has been widely used in outdoor air distribution simulation. Among the two equation models, the model is the most famous, so that it has almost become the synonym of eddy viscosity model. Among them, the K-ε model includes standard K-ε model, RNG K-ε model, low Re number K-ε model and their related derived models, which together constitute the main application model of two-equation model.

3.3 Model simplification and calculation domain determination

This paper studies the outdoor thermal-wind environment of the building, so the interior of the building is directly treated as a solid area. In addition, considering that the building object itself is relatively regular, the target building is directly simplified as a regular solid cuboid. First of all, in order to verify the feasibility of simulation, a model with the same size as the experimental wood block was constructed for simulation calculation. The model was a cuboid with the length, width and height of 20cm, 10cm and 40cm respectively. Then, in order to study the characteristics of the wall thermal plume of the actual building size, an enlarged model was established according to the ratio of 1:100 to the experimental model. The physical model was a cuboid with length, width and height of 20m, 10m and 40m respectively.

Through the study of related similar calculation cases, considering the characteristics of the research object of this building model, the calculation domain size was finally selected as: except the bottom boundary overlapping with the bottom of the building, the other boundaries including four side boundaries and the top boundary should be 5 times the height of the building from the surface of the building model. In addition, the ratio of the area covered by the model to the area of the whole computational domain should not exceed 3%.

3.4 Meshing and independent verification

After analysis, this model adopts node setting method to divide structured grid, as shown in Fig 4. In order to coordinate with the use of RNG low Re number K-ε model, O-block is generated outside the model to facilitate the appearance of reasonable boundary layer encryption region near the wall surface. The total number of grids is 112,904 and the total number of nodes is 120,324.
In order to verify the grid independence, for the simulation model with a height of 40 cm, the wind speed at the measuring point 1 cm away from the hot surface of the building model and 30 cm away from the ground was taken as the verification reference, and the wall temperature was set at 45°C. The number of grids was set as 50,000, 100,000, 200,000 and 400,000 respectively for numerical simulation. The wind speed calculation results of selected measurement points varied with the number of grids, as shown in Fig 4.

3.5 Boundary condition setting and solution setting

According to the theoretical research, the simulation chooses constant heat flow boundary conditions, the heating surface chooses three different sizes of heat flow density, 50W/m², 100W/m², and 200W/m². Other building wall heat flux is 0. The boundary of the region was calculated, the free outflow boundary condition was selected at the top, and the symmetric boundary condition was selected in the other directions. The simulated outdoor initial temperature was set to 20°C. The momentum equation, turbulence equation and dissipation rate are all discrete second-order schemes. The SIMPLE scheme is selected for velocity and pressure coupling, and the residual convergence value is 1e-05.

4 Result and analysis

4.1 Model validation

In Fig. 5, it can be seen that in the vertical direction of wind speed distribution, the wind speed measuring points 1-4 cm away from the surface are basically consistent with the simulated wind speed curve, with the maximum deviation of less than 10%, while in the position close to the wall, the experimental data of some measuring points have a large deviation from the simulated value. Consideration is caused by two reasons. On the one hand, as mentioned above, the heating time of the electric film is short, and the air near the wall is not heated completely. On the other hand, the temperature of the air near the wall is higher, and the heat transfer between the air and the surrounding air is more intense, which makes the reading fluctuate significantly in the actual test process and is not easy to be stable, resulting in errors in artificial estimation. It can be seen from Fig 5 that the numerical simulation results are slightly different from the prototype, but the difference between measured and simulated results is not large, basically less than 10%, which meets the requirements of accuracy.

4.2 Influence of heat flux on building elevation temperature

As can be seen from Fig 6, with the increase of heat flow on the wall, the peak temperature of the thermal plume increases and the high temperature area increases. When the wall heat flow is 200W/m², the highest temperature is 52°C. When the wall heat flow is 100W/m², the highest temperature is 43°C. When the wall heat flow is 50W/m², the highest temperature is 37°C. It can also be found from the figure that the temperature variation trend of the wall heat flux is consistent in the three cases. This is because...
the radiation of the wall temperature to the surrounding air meets the law of radiation attenuation, and the simulated conditions are ideal. Therefore, the temperature variation trend is consistent when the wall heat flux is changed. Within the range of 0-5m from the wall, the temperature decreases from the highest to about 20°C, and then basically remains unchanged.

4.3 Influence of heat flux on building wind speed field

Fig.7. Curve of wind speed temperature change at different heat flux.

Fig. 7 shows the horizontal velocity distribution of heat plume under different wall heat flow conditions (wall heat flow density 50W/m², 100W/m², and 200W/m²). In general, with the increase of the distance from the wall, the horizontal velocity at different heights increases first and then decreases. Under the combined action of buoyancy and surface drag, the horizontal velocity reaches the maximum in the range of 0.5-1.0m from the wall surface. In the range of 1.0-5m, the wind speed decreases continuously, because the heating effect of the wall surface on the surrounding air is affected by the distance. Moreover, with the increase of wall heat flow density, the wind speed variation trend of the three wall heat flow conditions is consistent, but only the wind speed size changes. This indicates that the wall heat flow density does not affect the thermal plume characteristics.

5 Conclusion

Through PIV test and simulation study on the thermal plume characteristics of building facades under different wall heat flow densities, the following conclusions are drawn:

For the temperature field, with the increase of the wall heat flow density, the temperature change trend is basically the same, but the maximum temperature is different. When the heat flow density is 50W/m², 100W/m² and 200W/m², the maximum temperature is 37°C, 43°C, 52°C respectively. In general, in the vertical direction, as the distance from the ground increases, the temperature first increases and then stays the same, and finally increases, in the horizontal direction, the temperature first drops sharply and then stays the same.

For the wind speed field, with the increase of heat flux, the wind speed near the wall will also increase, but the change trend of wind speed is basically unchanged. With the increase of the heat flux of the building wall, the maximum flow velocity appears at the top of the area near the wall, and the peak velocity of the heat plume at 50W/m², 100W/m² and 200W/m² are 1.5 m/s, 2.1m/s and 2.6m/s, respectively.

In this study, only the heat plume along the vertical facade direction under different heat flow densities was studied. Due to the limitation of experimental tests, influencing factors such as model height, building shape and surface roughness were not studied. In this study, the characteristics of the thermal plume of a single building model are proposed, and the formation, evolution and interaction of the thermal plume of the building complex will be the future research direction.

Acknowledgments

This work was supported by the National Natural Science Foundation of China (Grant No. 51808162) and the Fundamental Research Funds for the Central Universities (Grant No. HIT.NSRIF.2020038).

References