Analysis of wind environmental characteristics around a square building

Q H Chen¹, H T Hu¹, C Z Qian¹ and C P Chen¹ ² *
¹West Coast Wind Engineering Research Center, Xiamen University of Technology, Xiamen, Fujian, 361024, China
²Xiamen Ocean Vocational College, Xiamen, Fujian, 361100, China
*Corresponding Author: C P Chen, +865927769301, cpchen@126.com

Abstract. The simulation of the wind environment around a square building is conducted in this paper. Two obvious flow features and their law are presented. When the airflow comes to the building, it separates into four branches, namely up-and-down flows and lateral flows. Firstly, the up-and-down flows are demarcated at 3/4 of the height with varied incoming flow speed. The upward flow reaches maximum speed at the roof and runs over the building, forming vortex in the leeside. The downward flow also achieves the maximum value near the bottom. Secondly, the lateral flows separate symmetrically about the axis and the lateral speed increases from zero along the width direction till the stream reaches the edges of the building and then decreases gradually. The study of the wind flow around a single building plays a basic role in the wind environment research of multi-buildings.

1 Introduction
Along with the population aggregation, the contradiction between the rapid expansion of urban and limited land resources intensified, the urban construction becomes denser, greatly increasing the flow complexity near the ground and making adverse effect to the wind environment outside the buildings. These adverse impacts become worse especially in the weather with strong winds or typhoon.

Therefore, the wind environmental characteristics of buildings already become essential component of the wind engineering, and prediction of the wind environment around buildings has been carried out at the practical design stage[1-9]. To be mentioned, the working group for CFD prediction of the wind environment around one building was organized by the Architectural Institute of Japan(AIJ)[10]. Based on the studies above, the complexity features of the flow around buildings are clarified gradually. But the research focusing on the wind environment of a single building still needs to be further deepened, which is the basic step for wind environment research of architectural complex and has not been specified clearly yet, namely the detail laws of the airflow, such as the vortex separation, reflow and reattach status. Therefore, the flow characteristics surrounding a square building is studied further in this paper with Computational Fluid Dynamics (CFD) approach.

2 Validation of numerical method
Attempts to provide statements about the accuracy of CFD method adopted, the first step of this paper is to validate the accuracy of simulation approach and the experimental results by Meng and Hibi[11] are used to validate the results of the CFD simulation.
2.1 Numerical setup

The shape of experimental model is with 2:1:1(height: width: depth) and the height 160mm (see figure 1a). The height is represented as $h$ and the width/depth is represented as $b$. In the experimental case, the incoming flow velocity $u$ varies with height and $u_r$ (reference velocity at $z=h$) is 4.4m/s with Reynolds number $2.4 \times 10^4$. The origin is set in the center of the building and three components of velocity are described as $u_x$, $u_y$ and $u_z$, as shown in figure 1b.

The software Fluent is adopted for the simulation and the setting details are as follows:

- **Size of the computational domain:** $21b(x) \times 13.75b(y) \times 11.25b(z)$
- **Boundary conditions:**
  - Inflow boundary condition
  - Outflow boundary condition: Based on the values tested in the wind tunnel[11]
  - Surface of the building model
  - Surface of the wind tunnel floor: Wall with roughness length $z_0 (z_0=1.8 \times 10^{-4}m)$
  - Surface of wind tunnel side walls and ceiling: Symmetry
- **Turbulence model:** Standard $k-\varepsilon$

The hexahedral structured grid is employed for the discrete of the whole calculation domain and in order to improve calculation accuracy, the mesh refinement is considered near the building. Aiming to determine the appropriate meshing plan, three grid schemes are conducted (see table 1).

| Grid scheme      | Minimum grid size | Grid number |
|------------------|-------------------|-------------|
| Coarse mesh      | 0.005             | 1.04x10^6   |
| Medium mesh      | 0.002             | 2.50x10^6   |
| Fine mesh        | 0.001             | 5.25x10^6   |

2.2 Comparison of simulating and experimental results

The distributions of average wind velocity component $u_x$ on the vertical cross-section at the center of the building ($y=0$) is shown in figure 2. The values are normalized and only the velocity profiles of $u_x$ at the positions of $x/b=0.075$ and $x/b=0$ are presented in the figure due to space limitations. Generally, the calculated values agree well with the experimental values except the position near the roof. The explanation could be that when approaching the roof surface, the flow slows down quickly and the calculation might not able to reflect this phenomenon. Meanwhile, the distributions of average wind velocity component $u_x$ on the horizontal plane near the ground surface ($z=1.25b$) is shown in figure 3. The calculated data agree relatively well with the experimental values, except the value near the
building boundary. Based on these diagrams, the simulation approach is proved with good accuracy.

Regarding to the grid independence, three different mesh schemes all presents good accuracy and the simulated results show some differences only in the region quite near the building, which also means that the conducted grid arrangements are sufficient and can be employed for further study. Moreover, this paper focuses on the detailed flow characteristics around the building, thus the finer mesh scheme is adopted in the research.

![Figure 2](image1.png)  
**Figure 2.** Comparison of $u_x$ distributions along $z$-direction in vertical section $y=0$

![Figure 3](image2.png)  
**Figure 3.** Comparison of $u_x$ distributions along $y$-direction in horizontal section $z=1.25b$

3 Results of the flow field simulation around a single building

The simulation setup above is then adopted. The incoming wind speed at reference height is varied and set up as $u_r=0.1 \text{m/s}, 4.4 \text{m/s}, 10 \text{m/s}$ and $20 \text{m/s}$ respectively (see table 2).

| Table 2. The incoming flow conditions |
|---------------------------------------|
| $u_r$ (m/s) | $Re$ (-) |
| 0.1 | $0.054 \times 10^4$ |
| 4.4 | $2.40 \times 10^4$ |
| 10 | $5.40 \times 10^4$ |
| 20 | $10.8 \times 10^4$ |

3.1 Up-and-down flow

3.1.1 Streamlines of up-and-down flow around the building. From the streamline chart of $y=0$ plane (figure 4), the block effect of the building on the flow making obvious change on the flow field both the windward side and leeward side of the building.

It can be observed that the phenomenon of flow spilt occurs just before the building. The majority part of the airflow moves down to the bottom area and mixes with the incoming flow, forming reflux. In this region, the pedestrian are affected not only by the incoming flow but also reflux and the dive flow. Wind environment uncomfortableness should be considered.

On the other side, upward flow climbs to the roof height and then separation happens. When the incoming flow wind velocity is small, such as $u_r=0.1 \text{m/s}$ (figure 4a), two obvious irregular vortices formed above and behind the building, and the phenomenon of backflow also appear around the bottom of leeside. With the increase of wind speed, the flow above the roof reattaches again and one
large flow vortex is formed behind the building (figure 4b–d). The vortex impact range extents with the increase of wind speed. Additionally, the flow field behind the building is with relative low wind speed, in such case the pollution is not easy to diffuse.

**Figure 4.** Streamlines of relative mean velocity on the y=0 plane with incoming flow variations (a) \(u_r=0.1\text{ m/s}\), (b) \(u_r=4.4\text{ m/s}\), (c) \(u_r=10\text{ m/s}\), (d) \(u_r=20\text{ m/s}\)

3.1.2 \(u_z\) profile before the building. The phenomenon of up-and-down flow can be found from the \(u_z\) profile more obviously shown in figure 5. The values of velocity component \(u_z\) along the height are divided into evident positive and negative partitions and the demarcation points with different incoming flow almost coincident at \(z/b=1.5\), namely the height position \(3h/4\).

The flow reinforcements are found in the regions near the ground and roof. To be more explicit, in the region \(z/b>1.5\) (or \(z/h>0.75\)), the flow climbs up with increasing velocity to \(u_z/u_r=1.0\) (at the roof \(z/b=2.0\)) and then goes upward with gradually less values. On the other hand, in the region \(z/b<1.5\) (or \(z/h<0.75\)), the air dives and with the most significant enhancement at \(z/b=0.3\). Based on the \(u_z\) profile, three key height positions should be highlighted, respectively \(z/b=0.3, 1.5\) and 2.0.

3.1.3 \(u_z\) profile after the building. In figure 6, the values of velocity component \(u_z\) behind the building are shown. Besides, a significant speed loss area is observed. When the coming flow reference velocity is 0.1, which is low, the flow pattern behind the building is disorder (see figure 4a) and the velocity speed fluctuates round zero. As the incoming wind speed increases, the \(u_z\) values are positive within the scope of building height, namely the air flows from down to up, presenting the vortex behind the building is rotating clockwise (see figure 4b–d). The \(u_z\) value increases along the height and reaches a maximum at \(z/b=1.0\), then it weakens gradually till \(z/b=2.0\). Since the separation vortex appears on the roof, a slight fluctuation also turns up between 2.0 < \(z/b<2.5\).

**Figure 5.** \(u_z/u_r\) profile right before the building with incoming flow variations

**Figure 6.** \(u_z/u_r\) profile right after the building with incoming flow variations
3.2 Lateral flow

3.2.1 Streamlines of lateral flow around the building. Based on the horizontal view of flow field in figure 7, when the airflow approaches the construction, the stream flows to the both sides and reaches maximum speed at the edges of building (at $y/b=\pm0.5$), then gets over with expansion, cladding the shedding vortices. The wake vortices generate and move downstream with the evolution of time, but only one flame is shown here.

When the flow is laminar (such as $u_r=0.1\text{m/s}$), three groups vortices generate on the leeside. When the flow rate increases and turns to turbulent, two symmetric vortices can be observed and the vortex extension expands along with the increase of flow speed.

3.2.2 $u_y$ profile before the building. In order to investigate the lateral flow quantitatively, the profiles of $u_y$ component right before the building at different horizontal positions on the condition of same incoming flow, such as $z/b=0.5, 1, 1.5$ and 2 are depicted in figure 8 and the profiles of $u_y$ component at $z/b=1$ on the condition of different incoming flow are shown in figure 9.

From figure 8, the symmetrical velocity distribution can be found with the condition $u_r=4.4\text{m/s}$. As the airflow gets close to the building, it divides into left and right halves and the $u_y$ values begin to increase from the symmetry axis. The maximum values appear at almost the same $y$ positon ($y/b=\pm0.5$). The $u_y/u_r$ values creep up to 0.8 to the great extent at $z/b=0.5, 1$ and 1.5, but 0.5 at the height of roof($z/b=2$). When the flow moves laterally, the $u_y$ data decrease to zero gradually.

From figure 9, although the incoming flow is varied with different speeds, the normalized values are almost coincident at the height $z/b=1$. That is, the flow field forwarding to the lateral sides of building is proportional to the incoming flow.

![Figure 7](image_url)

**Figure 7.** Streamlines of relative mean velocity on the $z/b=1$ plane with incoming flow variations (a) $u_r=0.1\text{m/s}$, (b) $u_r=4.4\text{ m/s}$, (c) $u_r=10\text{ m/s}$, (d) $u_r=20\text{ m/s}$
4 Conclusion

The three-dimensional flow field around a single building is analyzed with Fluent in this paper and the simulating approach is validated with experimental data. The flow characteristics are revealed and when the incoming flow approaches the building, it splits into up-and-down flow and lateral flow.

1) Right before the building, the up-and-down airflow generates at the demarcation point \( z/b=1.5 \). The majority part of the airflow moves directly and mixes with the incoming flow near the bottom, obtaining its most significant velocity enhancement at \( z/b=0.3 \). The left climbs up to the roof and the \( u_c \) component starts to increase and reaches the maximum at the height of roof. When the upward flow moves over the building, the vortex appears in the leeward side.

2) Lateral flow due to the building block also causes the rise of wind speed. The stream flows to the both sides symmetrically and reaches maximum \( u_t \) speed at the edges of building \((y/b=\pm 0.5)\). The \( u_t/u_0 \) values creep up to 0.8 to the great extent at \( z/b=0.5, 1 \) and 1.5, but 0.5 at the height of roof \((z/b=2)\). Then lateral flow gets over with expansion, cladding the shedding vortices.

Acknowledgement

The authors would like to acknowledge with great gratitude for the supports of the National Science Foundation of China(Grant No: 11272270, 51778551), the Science and Technology Plan Project of Xiamen(Grant No: 3502Z20161016), the Young Teachers Education Science Research Project of Fujian(Grant No: JA15384, JAT170410), the High-level Talent Project of Xiamen University of Technology(Grant No: YKJ14025R, YKJ15042R).

References

[1] Wu Y, Zhang X, Li H, Wang C, Nie G, Cheng Z 2011 Numerical simulation of the pedestrian level wind environment around high-rise buildings J. Zhengzhou Univ.(Nat. Sci. Ed.) 04 110-115
[2] Wu X, Li Q, Li Y 2014 Test on surface wind pressure distributions and wind load characteristics for complex shape high-rise building Journal of Architecture and Civil Engineering 01 76-82
[3] Zhao Y, Su H, Kang Y, Liu Z 2016 CFD Simulations of the wind environment of an industrial project in Wenzhou Building Energy Efficiency 02 125-128
[4] Shi X, Zhu Y, Duan J, Shao R, Wang J. 2015 Assessment of pedestrian wind environment in urban planning design Landscape and Urban Planning 140 17-28
[5] Li L, Zhang L, Liu G, Liu S, Zheng S 2015 Wind environment simulation of green building. Journal of Civil Architecture & Environment Engineering S2 215-218
[6] Neofytou P, Venetsanos A G, Vlachogiannis D, Bartzis J G, Scaperdas A 2006 CFD simulations of the wind environment around an airport terminal building Environmental Modelling & Software 21 520-524
[7] Mochida A, Tominaga Y, Murakami S, Yoshie R, Ishihara T, Ooka R 2002 Comparison of various k-ε models and DSM applied to flow around a high-rise building Wind and Structures 5 227-244
[8] Hussein A S, El-Shishiny H, 2009 Influences of wind flow over heritage sites: A case study of the wind environment over the Giza Plateau in Egypt Environmental Modelling & Software 24 389-410
[9] Blocken B, Carmeliet J. 2008 Pedestrian wind conditions at outdoor platforms in a high-rise apartment building: generic sub-configuration validation, wind comfort assessment and uncertainty issues Wind and Structures 11 51-70
[10] Yoshie R, Mochida A, Tominaga Y, Kataoka H, Harimoto K, Nozu T, Shirasawa T. 2007 Cooperative project for CFD prediction of pedestrian wind environment in the Architectural Institute of Japan Journal of Wind Engineering and Industrial Aerodynamics 95 1551-78
[11] Meng Y, Hibi K. 1998 Turbulent measurements of the flow field around a high-rise building Journal of Wind Engineering 76 55-64