

T4D/655: Modelling of flow past a building in urban heat island-induced flow

E Berbekar ^{1*}, G Kristof ¹, T Lajos ¹

*Budapest University of Technology and Economics, Department of Fluid Mechanics, Budapest, Hungary¹
berbekar@ara.bme.hu*

Abstract

When geostrophic wind is negligible, the ventilation of the city is induced by the Urban Heat Island Circulation (UHIC). The flow towards the city centre is limited to a layer above the surface, therefore, in case of an UHIC induced flow, the vertical velocity profiles differ significantly from the geostrophic wind. In consequence, the UHIC induced flow around buildings is expected to differ from the case, when an isothermal flow is present. Results of computational fluid dynamics (CFD) simulations and measurements verify this assumption. The paper focuses on the CFD and experimental modelling of flow field generated by UHIC and on differences of flow fields past a building model during different thermal stratifications. The experimental facility is the test section of a wind tunnel, open to the environment at one end. A heat source, modelling the city centre and generating circulation similar to the flow induced by an UHIC, is placed in the vicinity of the closed end of the box. The building model is placed in this flow, and velocity distribution is measured. By a slight modification to the facility, the isothermal flow can be modelled and velocity field past the building model can be measured and compared to the results from the thermally stratified experiment.

Keywords: urban heat island, isothermal flow, computational fluid dynamics, flow field around a building

1. Introduction

The phenomenon that the temperature in the city centre is higher than the temperature in the rural areas is called the Urban Heat Island (UHI). UHI can be characterised by its intensity, which is the temperature difference between the urban and rural areas. [1]

The UHI causes warmer winters, hot summers, increasing humidity and precipitation rate and a circulation in the city, which is called the Urban Heat Island Circulation (UHIC). The UHIC is caused by the temperature difference between the urban and rural areas. It can interact with the geostrophic wind, sea breeze, down flow from a hill, etc., but it can also cause the ventilation of the city alone.

The vertical velocity profiles of the flow induced by the UHIC and of the geostrophic wind differ significantly [2].

There are numerous ways that can be used to investigate the UHI [3]; such as field measurements, thermal remote sensing, computational modelling and small-scale modelling. When using small scale modelling, similarity criteria have to be matched, so that the same dimensionless equations and boundary conditions describe the full scale case and the model scale measurements.

Different techniques had been used to model the UHIC using small scale measurements. Beside outdoor scale-model experiments, heated boundary layer wind tunnels and heated water tanks can also be used to model an UHI. These methods are mainly used to validate computational models.

Numerous investigations have been carried out to determine the flow field around a single building. Field studies, wind tunnel measurements and computational fluid dynamics (CFD) models were used during these investigations. The knowledge of the flow field around a high-rise building is important to determine the dispersion of pollutants in an urban area. [4]

This paper presents a measurement arrangement that is capable of generating an UHIC-like flow. A building is placed into this flow to determine the flow field around it. With the experimental facility an isothermal flow can also be generated, which represents the ventilation of the city caused by the geostrophic wind and no UHI is present. With these two arrangements, the difference between the flow fields around a building in case of an UHIC and a geostrophic wind can be investigated.

2. Description of the experimental setup

The goal of the experiment is to induce an UHIC-like flow in laboratory conditions to examine the main parameters of the flow and investigate the flow around a single building placed in an UHIC. The same experimental facility is capable of generating an isothermal flow. Therefore the results obtained from the UHIC measurements can be compared with the flow patterns in the vicinity of the ground, and the flow field around a building, in case of an isothermal flow.

The experiments are carried out in a wind tunnel with a closed test section. The part of the wind

tunnel that is used to generate the UHIC-like flow can be seen in Fig.1.

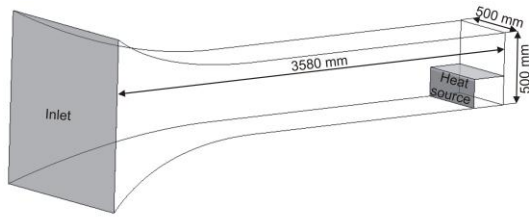


Fig 1. Measurement facility

When the isothermal case is measured, the wind tunnel operates as usual, the test section is open on both sides and the heat source placed inside is not operating. The flow is generated by the fan inside the wind tunnel.

When the UHIC is modelled, the fan of the wind tunnel is turned off and only one side of the test section is open, the one that leads to the converging duct. The other side of the test section is closed (right side of Fig.1). The downtown area is represented by a heat source, which is placed on the side of the test section, where the wind tunnel is closed. The flow is generated by the temperature difference, caused by a 2250 W heat source.

After the flow of the two cases is measured, a building model ($W = 40$ mm wide, $L = 40$ mm long and $H = 100$ mm high) is placed on the test section of the wind tunnel to determine the difference between the flow fields around a building in case of an UHIC and an isothermal flow. During the measurements, the Reynolds number was 2000 and the Richardson number was 15.

The velocity measurements were carried out with a 2D Laser Doppler Anemometry (LDA) system.

3. Description of the CFD model

Before the measurements, a CFD simulation was carried out to investigate, whether the facility is capable of generating an UHIC-like flow. Therefore the geometry of the model was the same as described during the description of the wind tunnel experiment setup (Fig.1).

ANSYS Fluent was used to carry out the simulations. The Reynolds Averaged Navier-Stokes Equation and the Energy Equation were solved. For modelling the turbulence, both the Standard and the Realizable $k-\epsilon$ model were used. Time-dependent simulations were carried out and the buoyancy effect was modelled by the Boussinesq approximation.

Both the UHIC case and the isothermal case were investigated using CFD. In the UHIC case, the inlet of the converging duct is set to constant pressure boundary condition and the heat source was modelled as a porous zone. In the isothermal case, the inlet of the converging duct is set to be a velocity inlet, and the outlet is defined as a pressure outlet.

4. Results and discussion

4.1 Results of the CFD simulations

The CFD simulations were carried out to investigate, whether the measurement facility is capable of generating an UHIC-like flow, and to predict the results of the measurements.

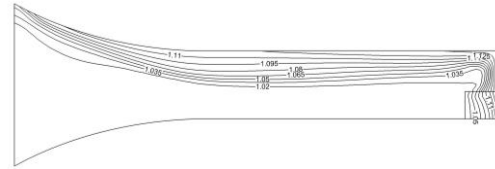


Fig 2. Dimensionless temperature distribution

The dimensionless temperature distribution of the test section was investigated (Fig.2), because the UHIC-like flow is generated by the temperature difference. The ambient reference temperature was used to calculate the dimensionless temperature values.

In Fig.2 it can be seen that the heat rises above the heat source and exits the test section through the converging duct. On the lower region of the test section, ambient air flows in the direction of the heat source. The velocity vectors of this circulation, which is similar to the UHIC are shown in Fig.3.

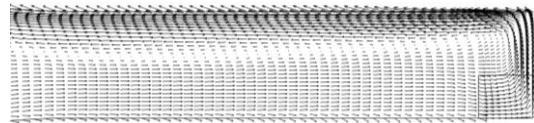


Fig 3. Velocity vectors of the UHIC-like flow

The temperature distribution in case of the isothermal flow is not shown, because the temperature is the same in every grid cell.

Since according to the CFD simulation, the measurement facility is capable of generating an UHIC-like flow, simulations were carried out to investigate the flow field around a building during different thermal stratifications.

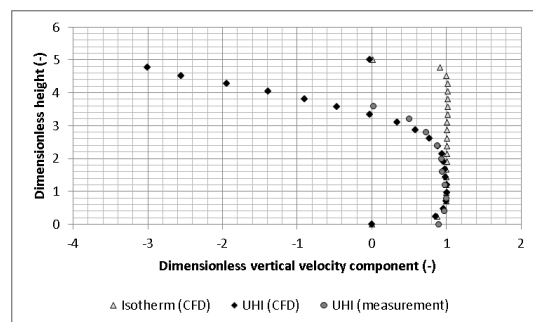


Fig 4. Vertical velocity profiles

The vertical dimensionless velocity profiles shown in Fig.4 are representing the approaching flow, which occurs before the flow reaches the building model, in a distance of $1.5H$ from the building. The reference velocity is the velocity of the approaching flow in the height of the building. It can be seen in Fig.4 that when modelling an UHIC-like flow, the vertical velocity profile of the

approaching flow looks similar to the velocity profile that occurs when an UHI is present in a city.

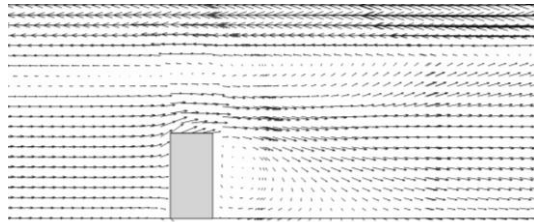


Fig 5. The flow field around a building in the UHIC case taken from the CFD simulation

In Fig.5 the flow field around a building can be seen in the UHIC case as the result of the CFD simulation. It can be compared to the flow field around the same building in case of an isothermal flow, which can be seen in Fig.6.

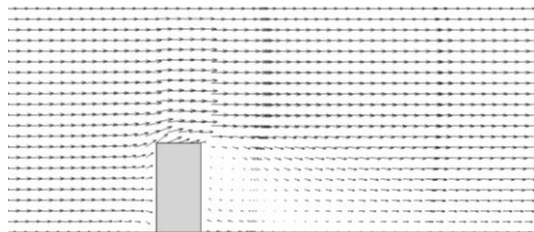


Fig 6. The flow field around a building in the isothermal case taken from the CFD simulation

As Fig.5 and Fig.6 suggest, the shape and size of the separation bubble behind the building differs in the two cases.

4.2 Results of the measurements

Small scale measurements were carried out to investigate the flow field around a building during different thermal stratifications. Two cases were investigated: one where an UHIC-like flow was generated and another with isothermal flow. The dimensionless velocity profile measured in case of an UHIC-like flow is shown in Fig.4. The measured profile is in good agreement with the dimensionless velocity profile that occurred in the CFD simulations.

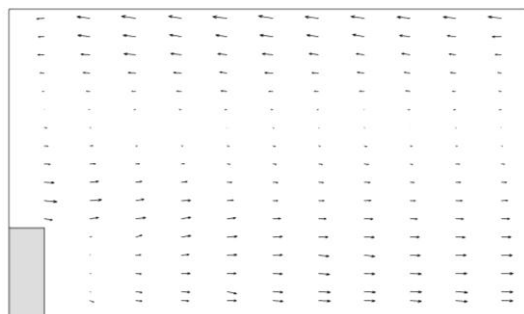


Fig 7. The measured flow field behind a building in the UHIC case

In Fig.7 the flow field behind the building can be seen during thermally stratified conditions (UHIC case). In Fig.8 the flow field around the building in

case of an isothermal flow can be seen. Fig.7 and Fig.8 supports the findings of the CFD simulations that the separation bubble behind the building differs, depending on the thermal stratification.

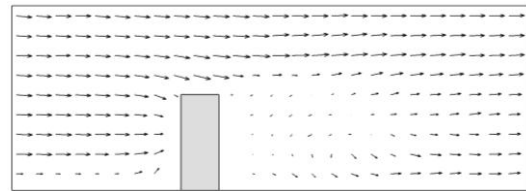


Fig 8. The measured flow field around a building in the isothermal case

5. Conclusion

Based on the results of CFD simulations and measurements, the facility presented in this paper is capable of modelling an Urban Heat Island Circulation. The difference between an UHIC- and a geostrophic wind-induced flow can be examined using this facility.

The flow field around a building during different stratifications was investigated and was found different, when an UHIC-like flow is present from the case of an isothermal flow. The importance of this phenomenon lies in the findings of previous scientists, that the ventilation of a city and the pollution dispersion around a building is effected by the flow field.

In the future, different CFD models are planned to be used that model the separation bubble behind buildings more accurately. [5] Moreover, further measurements are planned to be conducted for input data and model validation.

6. Acknowledgements

The work reported in the paper has been developed in the framework of the project „Talent care and cultivation in the scientific workshops of BME" project. This project is supported by the grant TÁMOP-4.2.2.B-10/1--2010-0009.

7. References

1. Oke, T. R., (1982). The energetic basis of the urban heat island. *Quarterly Journal of the Royal Meteorological Society*, vol. 108.
2. Lu, J., S. P. Arya, W. H. Snyder and R. E. Lawson, (1997). A Laboratory Study of the Urban Heat Island in a Calm and Stably Stratified Environment. Part II: Velocity Field. *Journal of Applied Meteorology*, vol. 36, no. 10, pp. 1392–1402.
3. Cermak, J., (1996). Thermal effects on flow and dispersion over urban areas: Capabilities for prediction by physical modeling. *Atmospheric Environment*, vol. 30, no. 3, pp. 393–401.
4. Blocken, B. and J. Carmeliet, (2004). Pedestrian Wind Environment around Buildings: Literature Review and Practical Examples. *Journal of Building Physics*, vol. 28, no. 2, pp. 107–159.

5. Tominaga, Y., A. Mochida, S. Murakami, and S. Sawaki, (2008). Comparison of various revised k- ϵ models and LES applied to flow around a high-rise building model with 1:1:2 shape placed within the surface boundary layer. *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 96, no. 4, pp. 389–411.