The 15th International Conference on Wind Engineering
Beijing, China; September 1-6, 2019

# Optimization of numerical parameters in CFD tools to improve natural ventilation assessment in complex urban area

Guillaume CANIOT [a], Stephane SANQUER [a], Gang HUANG [a]

[a] *Meteodyn, 33 bd Salvador 44800 Saint Herblain, Allende Nantes, France*

ABSTRACT: This paper presents a validation of a CFD numerical approach with optimized numerical parameters. Data from wind tunnel experiment is compared with numerical results on mean pressure coefficient. Numerical optimization investigates influence of inlet profile, mesh size and turbulence parameters. Suitable chosen numerical parameters can decrease discrepancies on pressure coefficient by 20 %.

KEYWORDS: CFD, pressure coefficient, numerical optimization, wind tunnel validation

## 1 INTRODUCTION

Wind condition creates high and low pressures on building's external faces. This gradient of pressure will drive wind inside building. Pressure distribution is first order of investigation when natural ventilation matters. Database of pressure coefficient is widely used inside dynamic thermal simulation software [1][2][3]. Those tables are convenient when geometry is simple and there is no interaction with neighbor buildings. Nevertheless, reality of an urban project is far from such ideal cases. More complex tools are required to assess mean pressure on building walls: wind tunnel or numerical simulation using computational fluid dynamic (CFD) can handle such task.

By using commercial code "Urbawind", velocity field in a complex area was compared with wind tunnel experiment with an average of 5% error on mean velocity [4].

Pressure coefficient on buildings with neighbor interaction is here performed using optimization of numerical parameters according to the correlation with the experimental results.

## 2 NUMERICAL APPROACH: METHODOLOGY

The CFD method consists of solving the Reynolds-Averaged Navier-Stokes equations on an unstructured rectangular grid with automatic refinement of the mesh near obstacles. The CFD tool delivers the tri-dimensional mean velocity field, the turbulence energy field and the mean pressure for each point in the domain.

In this paper, three points of numerical optimization will be discussed through wind tunnel data.

### 2.1 *Inlet profile*

The boundary conditions at the inlet represent the influence of the upstream that have been cut off by the computational domain. In [5], inflow should be defined at minimum 5 times the height of the tallest building where the mean velocity profile is derived from logarithmic profile corresponding to the upwind terrain with the correct roughness length z0. In Urbawind, there are four typical roughness profile defined with z0= 0.001, 0.05, 0.25, 0.7 meter. The turbulent kinetic energy constant profile is defined from the shear velocity u*.

### 2.2 *Grid convergence*

The quality of the mesh is evaluated thanks to grid convergence. Urbawind solves usually from coarse cell (minimum 2m resolution) through medium (minimum 1m resolution) to fine cell (minimum 0.5m resolution).

### 2.3 *Turbulence parameters*

The transport equation for the turbulent kinetic energy *k* contains a dissipation term ε deduced from the mixing-length theory.

$$\frac{\partial}{\partial x_i}\left[\rho \bar{u}_i k - \left(\frac{\mu_T}{\sigma_k}\right)\frac{\partial k}{\partial x_i}\right] = P_k - \varepsilon = \mu_T\left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i}\right)\frac{\partial \bar{u}_j}{\partial x_j} - \varepsilon \quad (1)$$

The turbulence viscosity $\mu_t$ is considered as the product of a length scale with a speed scale, which are both characteristic scales of the turbulent fluctuations.

$$\mu_t = \rho k^{1/2} L_T \text{and } \varepsilon = C_\mu \frac{k^{3/2}}{L_T} \quad (2)$$

The turbulent length scale $L_t$ varies linearly with the distance at the nearest wall d. The dependence of $L_t$ on the distance via a coefficient $C_l$ was defined in order to well reproduce the flows separation around typical building façades and roofs.

## 3 TEST CASE ON A SINGLE BUILDING

According to [6] on a single building, several couples of $C_l$ and $C_\mu$ have been evaluated according of reattachment length of two cases.
The results were summarized in the table 1. These results were compared with many numerical simulations and experiments [7] and [8].

Table 1. Comparison of Reattachment length of Numerical models and Exp. value (b is edge of the building)

| **Turbulence model** | **Reference** | **$X_R$/b** | **$X_{R\_TOP}$/b** |
|---|---|---|---|
| k-ε (Standard) | [7] | 2.7 | No separation |
| k-ε (Modified) | [7] | 3 to 3.2 | 0.52 to 0.58 |
| Differential stress model | [7] | 4.2 | >1 |
| LES | [7] | 1 to 2.1 | 0.50 to 0.62 |
| k-l ($C_L$=0.15, $C_\mu$=0.01) | UrbaWind | 1.5 | 0.60 |
| Experiment | [8] | 1.42 | 0.52 |

Validation of pressure coefficient was made with experimental data on Silsoe cube ([9],[10]).

## 4 TEST CASE ON COMPLEX CASE

The project ORCHIDEE fund by French environmental and energy management agency (ADEME) evaluates thermal comfort inside and outside buildings. One part of the project consists in validation of the numerical results from simulation of the pressure coefficient. The pilot project will be focus on a district located in Saint-Denis on Reunion Island [11].
The model of the district consists of 21 buildings over 250m x 250m with a moderate slope of 2% but locally can rise up to 8.5%.

The experimental wind simulation is carried at Eiffel Wind Tunnel in Paris. About 15 points by building are defined to measure pressure the two-floor building and roof which represents more than 300 pressure taps in total. 16 wind directions (step of 22.5°) are performed in the wind tunnel. Roughness at the inlet is generated through building block with an average of 0.5 m aerodynamical roughness. The model of the project is shown with topography on the left of Figure 1 with an example of the pressure coefficient in the right side.

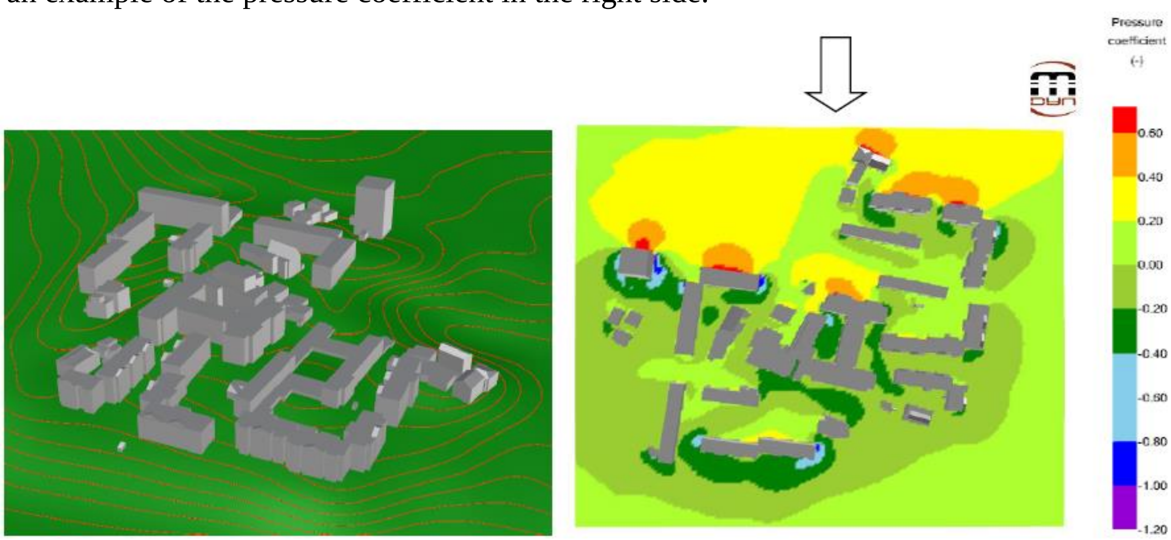


Figure 1 – CAD of the project and example of wind computation map (Pressure coefficient)

Correlation factor will be used to characterize difference between numerical and experimental result. Correlation factor ranges from 0 to 1 where 0 means both variables are not correlated linearly whereas 1 means that both variables are ideally correlated linearly. Figure 2 shows correlation between Urbawind and wind tunnel result of pressure coefficient of prevailing wind directions.

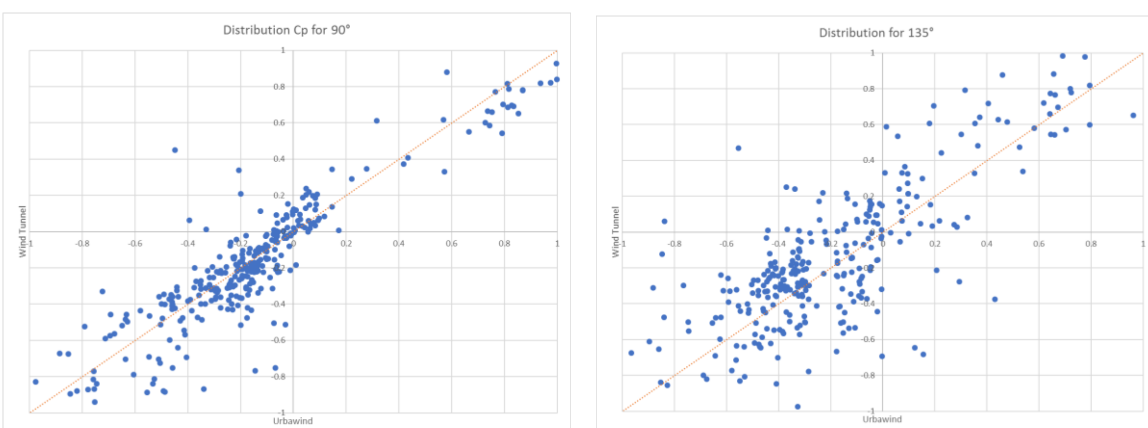


Figure 2 – Correlation factor between wind tunnel and simulation results for 90° (=0.90) and 135° (=0.79)

## 4.1 *Inlet profile*

A computation with CFD using log. profile as inlet profile provides a correlation factor of 0.69. To create similar inlet conditions than in wind tunnel, a generator of CAD geometry with random block is created. The comparison of the inlet profile (mean speed and turbulence intensity) provides good agreement between wind tunnel experiment and wind simulation. The factor correlation rises to 0.79 with this configuration i.e an improvement of +15% is achieved.

## 4.2 *Grid convergence*

By default, in Urbawind meshes, the smallest cell size is defined with 0.5m (fine mesh), 1.0m (medium mesh) and 2m (fine mesh). Considering the coarse mesh as a reference, correlation factor increases by 6.5% or 7.8% respectively with medium or fine cell size.

## 4.3 *Turbulence parameters*

Over this complex project with various aerodynamical effects, 48 couples of $C_\mu$ and $C_l$ are evaluated including $C_\mu$ =0.01 and $C_l$ =0.15. From the standard turbulence parameters $C_\mu$=0.09 and $C_l$ =0.36 and the optimum turbulence parameters found to be equal to $C_\mu$=0.05 and $C_l$ =0.15, the correlation factor is increased by 4%. Note that there is very low influence for $C_l$=0.15 for all $C_\mu$.

# 5 CONCLUSION

The paper presents a numerical optimization to assess pressure coefficient on facade in a CFD tools. Suitable chosen parameters can expect up to 20% increase of the accuracy of computation with respect to standard setup. User should be careful to use appropriate inlet profile, mesh accuracy and turbulence parameters.

# 6 REFERENCES


1 Liddament, M. W., 1986, Air Infiltration Calculation Techniques, An Applications Guide, AIVC.

2 Clarke J, 1990, Hand J, Strachan P, A building and plant energy simulation system. Energy Simulation Resarch Unit, Department of Mechanical Engineering, University of Strathclyde.

3 Eurocode 1: Actions on structure – Part 1-4 : general actions – Wind Actions, EN 1991-1-4, CEN/TC 250

4 Caniot, G., Li, W., & Dupont, G. (2011, July). Validations and applications of a CFD tool dedicated to wind assessment in urban areas. In 13th International Conference on Wind Engineering.

5 Franke, Jörg, ed. Best practice guideline for the CFD simulation of flows in the urban environment. Meteorological Inst., 2007.

6 Sanquer S., Caniot G., Wind-induced pressure coefficients on buildings dedicated to air change rate assessment with CFD tool in complex urban areas, In Proceedings of the 36th AIVC-(pp. 23-24), Madrid 2015

7 Tominaga, Y., Mochida, A., Murakami S., Sawaki S. (2008) Comparison of various revised k-epsi models and LES applied to flow around a high-rise buildings model with 1:1:2 shape placed within the surface boundary layer. Journal of Wind Engineering and Industrial Aerodynamics, 96, pp 389-411

8 Meng T., Hibi K. (1998). Turbulent measurements of the flow field around a high-rise building. Journal of Wind Engineering, Jpn, 76, pp 55-64

9 Richards P.J., Hoxey R.P., Connel B.D., Lander D.P (2009), Wind tunnel modelling of the Silsoe cube, Journal of wind engineering & industrial aerodynamics, 95, pp 1384-1399

10 Costola D., Blocken B., Hensen J.L.M (2009), Overview of pressure coefficient data in building energy simulation and airflow network programs, Building and Environment, 44, pp 2027-2036

11 Caniot G., Sanquer S., Blanchard B., Assessment of natural ventilation in urban area: Comparison between CFD and Wind Tunnel Building Simulation 2019 - To be published