EVALUATION OF ANSYS-FLUENT MODEL AGAINST FIELD DATA IN THE FRAMEWORK OF THE VIEPI PROJECT

Annalisa Di Bernardino[1], Agnese Pini[1], Fabio Nardecchia[2], Elena Conigliaro[1], Paolo Monti[1], Giovanni Leuzzi[1] and Armando Pelliccioni[3]

[1] DICEA, University of Rome "La Sapienza", Via Eudossiana 18, Rome, Italy.
[2] DAEEE, University of Rome "La Sapienza", Via Eudossiana 18, Rome, Italy.
[3] INAIL-DIMEILA, Via Fontana Candida 1, Monteporzio Catone, Rome, Italy.

SAPIENZA UNIVERSITÀ DI ROMA

DICEA - DIPARTIMENTO DI INGEGNERIA CIVILE, EDILE ED AMBIENTALE

1 GOALS

- ✓ Numerical, high-spatial resolution investigation of turbulence in correspondence of urban environment
- ✓ Analysis of the effect of different anemological conditions on flow pattern in correspondence of buildings
- ✓ Comparison between numerical outputs and in-situ data
- ✓ Evaluation of the flow incident on the walls and windows of the building
- ✓ Quantification of the outdoor/indoor exchanges of air mass within a street canyon adjacent to the building of interest

2 INTRODUCTION

In last decades, the increase in urbanization has dramatically changed the urban morphology and climatology.

The most obvious consequences can be seen in an increase in energy consumption and gradual reduction of green areas, replaced with roads, large areas of concrete and large vertical surfaces. Locally, the presence of an urban area changes air temperature and humidity as well as the pattern and the structure of the wind regime, with complex flows within the streets and squares. The interaction between atmospheric boundary layer flows and buildings have been widely investigated in recent years.

At the micro-scale, Computational Fluid Dynamics (CFD) has become more attractive because of its ability to simulate realistically building arrangements and complex scenarios (Buccolieri et al., 2013; Blocken et al. 2011). Particular attention has generally been focused to the relation to the urban air quality due to their importance in many aspects, like environmental science, meteorology and wind engineering (Franke et al., 2007; Garau et al., 2018; Salvati et al., 2019).

In this paper, micro-scale CFD simulations of the wind flow within a large portion (nearly 1 km^2) of the Roman urban area have been carried out.

The study is part of the **VIEPI** (*Integrated Evaluation of Indoor Particulate Exposure*) project, whose main goal is the evaluation of the infiltration factors of particulate matter in indoor environment.

3 STUDY AREA AND METEOROLOGICAL CAMPAIGNS

❖ STUDY AREA

- Campus of University of Rome "La Sapienza", located in the center of Rome (red line in Figure).
- Area morphologically heterogeneous, with vegetation, buildings of different heights and complex geometry, heavily trafficked streets, high presence of pedestrians and crossings of primary importance for the city.

❖ FIELD CAMPAIGNS

Measurements carried out during 2018 (ongoing) in the framework of the VIEPI project.

Present study ➡ simulation #1: **21th April 2018 14 UTC** (daytime)
simulation #2: **21th April 2018 01 UTC** (nighttime)

❖ INSTRUMENTS

- **N.01 sonic anemometer** located on the roof of the building of interest (blue star in Figure) at about 28.5 m above ground level
- **N.01 sonic anemometer** located within the adjacent street canyon (yellow dot in Figure) at about 7 m above ground level, placed 0.80 m off the wall of the building
- **N.01 sonic anemometer** located within the adjacent street canyon (yellow dot in Figure) at about 16 m above ground level, placed 0.80 m off the wall of the building

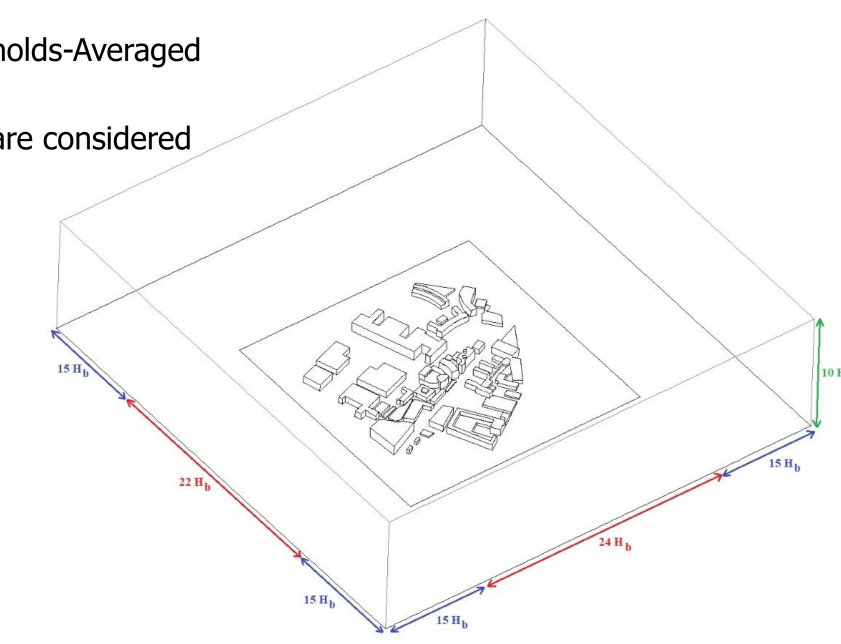
4 PHYSICAL MODEL

❖ NUMERICAL MODEL

- Standard k-ε turbulence closure based on the 3D Reynolds-Averaged Navier-Stokes (RANS)
- Boussinesq's approach, radiation model and PISO scheme are considered

❖ DOMAIN DIMENSIONS

H_b=25 m ➡ height of the investigated building
H_{max}=35 m ➡ maximum height of the buildings



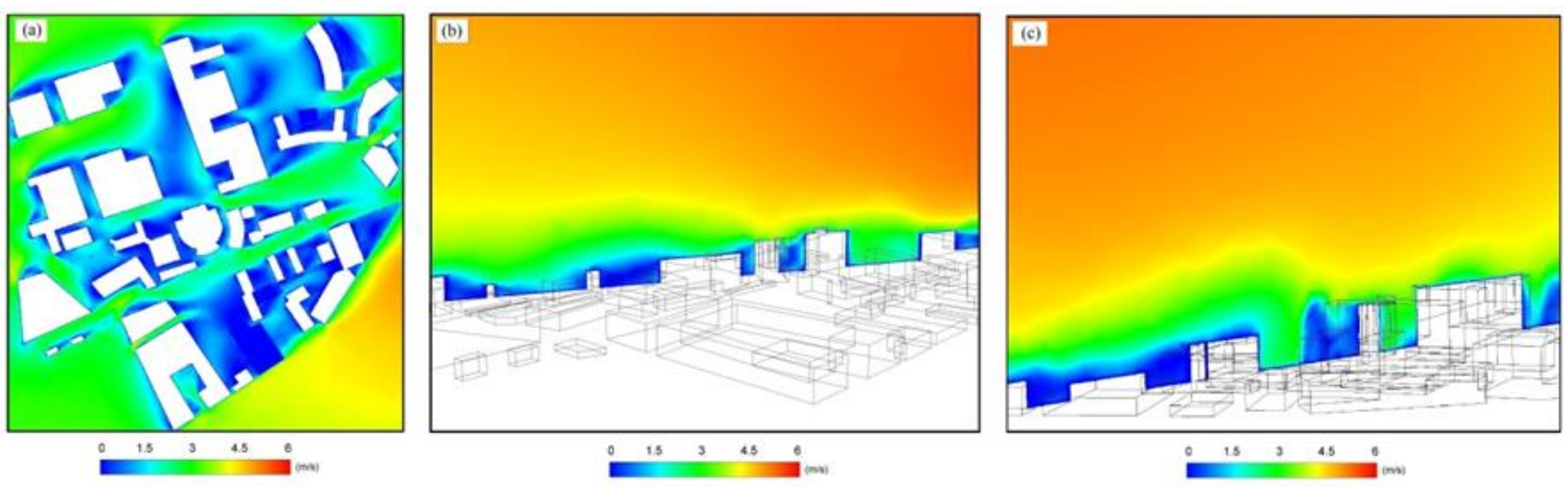
- ➢ Upstream/downstream extension of the domain ➡ 15 H_b
- ➢ Lateral extension of the domain ➡ 15 H_b
- ➢ Vertical extension of the domain ➡ 10 H_{max}

Numerical domain dimensions ➡ 1297 x 1345 x 350 m^3

	Mesh A	Mesh B	Mesh C
Number of cells	≈1.6·10^6	**≈2.5·10^6**	≈4.7·10^6
Mesh interval size (m)	3 - 30	**0.25 - 25**	1.5 - 20
Δ (%)	5.91	**-**	0.56

4 RESULTS

Simulation #1: 21th April 2018 14 UTC



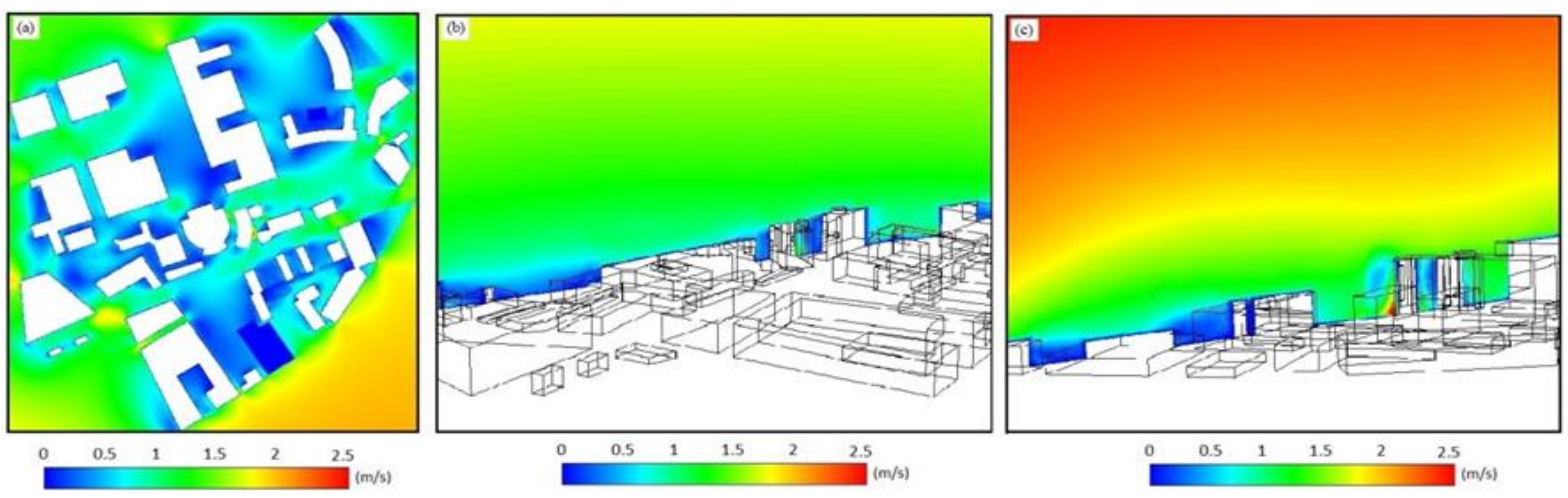
(a) Horizontal field of wind velocity at 28.5 m above ground level.
(b) Wind velocity magnitude in the vertical plane parallel to the building wall.
(c) Wind velocity magnitude in the vertical plane perpendicular to the building wall.

Comparison between CFD results and observations collected by the three anemometers.
u, v and w represent the zonal, meridional and vertical components of the wind velocity, respectively.

		u (m/s)	**v** (m/s)	**w** (m/s)
z=7 m AGL	CFD	-0.0831	0.2713	1.0229
	Observations	-0.1014	-0.1907	0.4138
	Δ	0.0183	0.4620	0.6091
z=16 m AGL	CFD	-0.0516	0.6526	0.6623
	Observations	-0.0519	0.0380	0.4302
	Δ	0.0003	0.6146	0.2320
z = 28.5 m AGL	CFD	2.8323	2.2011	0.3888
	Observations	3.0376	2.3423	0.3726
	Δ	0.2053	0.1412	0.0162

- ➢ Quite correct results in comparison with the anemometer placed on the roof of the building (error less than 8% in each component)
- ➢ Larger error within the street canyon adjacent to the building. In particular, the model does not capture the direction of meridional component of the wind

Simulation #2: 21th April 2018 01 UTC



(a) Horizontal field of wind velocity at 28.5 m above ground level.
(b) Wind velocity magnitude in the vertical plane parallel to the building wall.
(c) Wind velocity magnitude in the vertical plane perpendicular to the building wall.

Comparison between CFD results and observations collected by the three anemometers.
u, v and w represent the zonal, meridional and vertical components of the wind velocity, respectively.

		u (m/s)	**v** (m/s)	**w** (m/s)
z=7 m AGL	CFD	0.0344	-1.3620	-0.0551
	Observations	0.3842	0.3823	0.0685
	Δ	0.3498	1.7443	0.1236
z=16 m AGL	CFD	0.0192	-0.3525	-0.2297
	Observations	-0.1825	-0.2370	0.5674
	Δ	0.2017	0.1155	0.7971
z=28.5 m AGL	CFD	-0.7757	-0.8629	-0.0095
	Observations	-0.8811	-0.9596	0.0477
	Δ	0.1054	0.0967	0.0572

- ➢ Typical canyon effect can be observed: the wind, blowing from North East, channeled between buildings and the velocity increases at the center of the domain
- ➢ The meridional (i.e. the component almost parallel to the street canyon) component of the velocity shows high values, especially near the ground. This effect decreases as the height increases.
- ➢ Comparison gives good agreement above the rooftop while, within the canyon, the error increases.

5 CONCLUSIONS

Numerical results have been compared to field measurements carried out near the building of interest.

To investigate the influence of meteorological condition on the flow patterns in the area, two anemological situations were studied. The daytime simulation (21th April 2018, 14 UTC) has provided results in agreement with field measurement. In the nighttime case (21th April 2018, 01 UTC), differences between numerical results and measurements are more evident and larger errors occur.

These results must be seen as preliminary tests. In fact, more accurate simulations are needed to better reproduce the flow, especially within the canyons, where the high complexity of the geometry involves the formation of complex structures that are not correctly reproduced in the present work.

REFERENCES

- Blocken B., W. D. Janssen and T. van Hooff, 2011: CFD simulation for pedestrian wind comfort and wind safety in urban areas: General decision framework and case study for the Eindhoven University campus. Environ. Model. Softw., 30, 15-34.
- Buccolieri, R., F. Sartoretto, A. Giacometti, S. Di Sabatino, L. Leo, B. Pulvirenti and H. Wigö, 2010: Flow and pollutant dispersion within the Canal Grande channel in Venice (Italy) via CFD techniques. 13th International Conference on Harmonisation within Atmospheric Dispersion Modelling for Regulatory Purposes, HARMO 2010, 1-4 June 2010, Paris, France (pp. 760-764).
- Fluent Inc., 2006. Fluent 6.3 User's Guide. Fluent Inc., Lebanon.
- Franke, J., A. Hellsten, H. Schlünzen and B. Carissimo, 2007: Best practice guideline for the CFD simulation of flows in the urban environment. COST Action 732, Quality Assurance and Improvement of Meteorological Models.
- Garau, M., M.G. Badas, S. Ferrari, et al. (2018) Turbulence and Air Exchange in a Two-Dimensional Urban Street Canyon Between Gable Roof Buildings. Boundary-Layer Meteorol., 167, 123–143.
- Salvati, A., P. Monti, H. Coch Roura and C. Cecere, 2019: Climatic performance of urban textures: analysis tools for a Mediterranean urban context. Ener. Build., 185, 162–179.