Microclimate on building envelopes: wind tunnel and computational fluid dynamic analysis of basic and complex geometries

C. Hershcovich, R. van Hout, V. Rinsky and M. Laufer, Y. J. Grobman

Technion, Israel Institute of Technology
Haifa, Israel
cheli@campus.technion.ac.il, rene@technion.ac.il, rinsky@tx.technion.ac.il, mjlaufer103@gmail.com, yasha@technion.ac.il

ABSTRACT

This paper presents the results of an investigation into the potential of complex geometry to create a microclimate by changing the airflow on building façades.

The current stage of the research focused on developing a set-up and a methodology for examining and comparing the results of a physical wind tunnel test to those of CFD (computational fluid dynamics) simulations. A series of chosen façade geometries was examined using CFD simulations and physical wind tunnel simulations. The numerical simulations were validated by the experimental data. Based on the best performing geometries of the first examined series, new geometries will be developed and tested towards a better understanding of the façade's geometry contribution to the building's thermal performance.

Author Keywords

microclimate; building envelope; thermal insulation; computational fluid dynamics; complex geometry.

ACM Classification Keywords

I.6.1 SIMULATION AND MODELING

1 INTRODUCTION

The traditional approach to the design of building envelopes is based on a combination of layers that combine mass and material properties and create the thermal barrier from the outside. As opposed to the complex cellular structure of natural skins, which use their geometry to increase thermal performance, traditional building envelopes are typically based on flat orthogonal geometry, repetition, limited functions and structural homogeneity.

The following paper presents the results of research that examines the potential to develop a multifunctional building façade that employs complex geometry to create a microclimate on the building façade, which will contribute to its thermal performance.

The early stages of the research developed a theoretical framework for a shift toward building envelopes that are based on complex geometry and validated the potential of this approach with 2D computational fluid dynamic (CFD) simulations (Grobman, 2013)(Grobman and Elimelech, 2015). The current stage of the research focuses on the development and optimisation of complex cellular building envelope geometry. It employs and compares results from physical wind tunnel experiments and 3D CFD simulations.

This paper opens with a summary on architectural precedents, existing literature on complex geometry in building envelopes and the results of the previous stages of the research. It then presents the set-up and the methodology of the current stage of the research. The final section discusses the results, the conclusions of the current examinations and the directions worth pursuing in future stages of the research.

2 BACKGROUND AND PRELIMINARY RESEARCH

A precedent and literature review on microclimate and complex façade geometries in architecture was performed in a preliminary research (Grobman 2013) (Grobman and Ellimeleh 2015). Architectural precedents in this field can be divided into two types. The first focused on the esthetical aspects of the façade geometry and did not try to postulate better performance because of the facades geometry. Notable examples are the buildings of Antoni Gaudi, Eladio Dieste and Frank Gehry. The other type of designs focused on the material and morphological aspects of the façade to increase the building envelope's performance. Notable examples are the Beijing National Aquatic Center by PTW Architects and KOL/MAC Architecture's INVERSAbrane building envelope (Grobman and Neuman 2011). Academic research in this field that was reviewed in the preliminary research focused on green roofs and the effects of vegetation (Aydogan 2012), and bio inspired building facades (Gruber and Gosztonyi 2010, Knippers and Speck 2012, Badarnah, Nachman Farchi, and Knaack 2010, Laver et al. 2008).

The preliminary research developed a framework for a shift towards using the geometry of the building envelope to create a microclimate on the building façade. It also validated the potential of basic building envelope geometries to generate a micro-climate on the façade (Grobman and Ellimeleh 2015).

The preliminary research examined only a small number of basic façade geometries. It called for further examination of various types of geometries in order to better define the connection between microclimate and façade geometry. It also suggested that CFD results should be validated using wind tunnel tests with physical models due to the complex nature of air flow on facades.

3 METHODOLOGY

The current stage of the research focused on examining the microclimate developed on various envelope geometries and the potential impact of the microclimate on the thermal performance of the building envelope. The research methodology consisted of two stages. The first consisted of a comparative examination of similar chosen façade geometries in physical wind tunnel and CFD analysis. The aim of this stage is to validate the numerical simulations by experimental data and to select the best performing envelope geometries for further development and optimization. Following successful results in the first stage, the second stage of the research will continue developing the best performing geometries from the first stage using the CFD analysis tested in the first stage. The tile geometries will be initially optimized for thermal behaviour and later also for self-shading to decrease solar radiation. The final aim of the research is to develop a building envelope tile or series of tiles with increased thermal performance, which is generated by the envelope's tile geometry.

3.1 Flow field

Here, the flow impinging on a building envelope was modeled by a turbulent jet impinging on smooth and sculptured flat surfaces using a combination of numerical simulations and experiments. The physical experiments were conducted at the Environmental Multi-Phase Flow Laboratory at the Technion (Figure 1). An axisymmetric, turbulent air jet was created by installing a converging nozzle (contraction area ratio of 100:1) at the end of a small open loop wind tunnel. The exit diameter of the jet was D = 20 mm, and exit jet velocities were U_j = 1, 5 and 10 m/s, corresponding to 1, 3 and 5 Beaufort. Jet Reynolds numbers were Re = $U_j D/\nu$ = 1,300, 6,260 and 12,354, where ν is the kinematic air viscosity. The surfaces were installed at H = 10 cm (H/D = 5), where H denotes the distance measured from the nozzle exit. Plate dimensions were ten times those of D while modelled sculptured hole diameters were smaller than D. Flow field measurements were performed using Particle Image Velocimetry (PIV). The PIV system (LaVision GmbH) consisted of a CCD camera (2048x2048 pixels), an Nd:Yag laser (200mJ/pulse), laser sheet optics and acquisition/processing software (DaVis 8.1). A laser sheet (~ 1mm thickness) was created using a cylindrical lens and planar measurements were acquired at 5Hz. At each Re and plate configuration, 450 instantaneous vector maps were acquired having a field of view (FOV) of about 100x100 mm^2. Multi-pass data processing was performed starting with an interrogation window size of 64x64 pixels that was reduced to 8x8 pixels at 50% overlap resulting in a vector spacing of 0.2 mm. In between passes, outlier detection was performed and the data was smoothed using a 3x3 Gaussian spatial filter (Raffel et al. 2007).

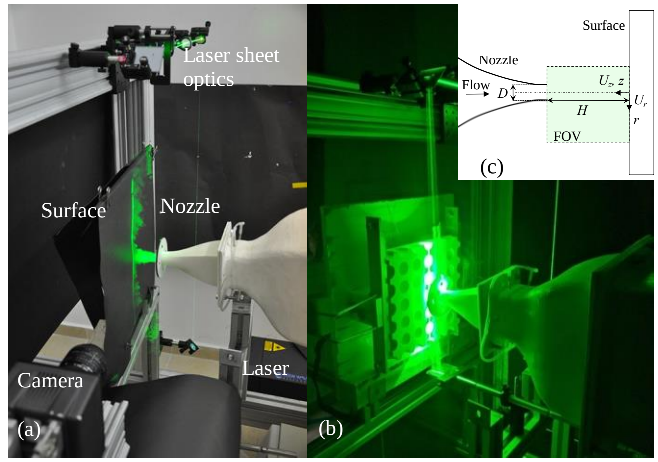

Figure 1. Picture of the (a) experimental PIV setup, (b) flow visualization on sculptured surface and (c) FOV and employed coordinate system

The computer-based simulation employed the commercially available Star-CCM+ 11 CFD package. A 3D simulation was set up with a flat impinging surface. The fluid parameters and jet geometry were modelled to match the experiments. The numerical simulations were based on the Reynolds Averaged Navier-Stokes (RANS) equations that were solved using the V2F model (e.g. Behnia et al. 1998) employing a slightly modified k-ε turbulence model and an additional transport equation for the variance of the wall-normal velocity fluctuations. Note that the V2F model was chosen since it has previously shown to offer the most accurate results for impinging jet problems excluding the overly "expensive" Direct Numerical Simulation and Large Eddy Simulation (DNS/LES) methods (Zuckerman and Lior 2005).

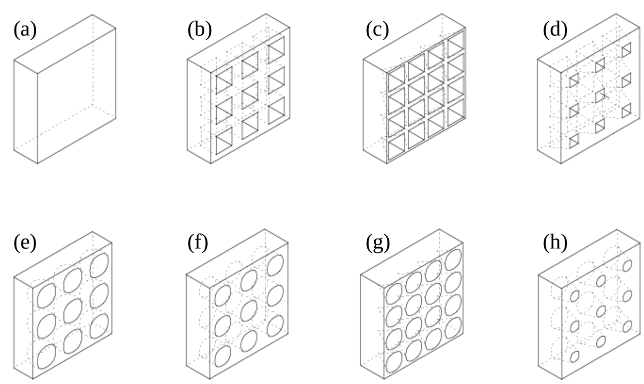

Figure 2. Test tiles: (a) – plane wall; (b)-(d) – rectangular cells; (e)-(h) – circular cells

In general, a jet flow impinging on a wall results in a stagnation region and a fast developing wall jet, radially extending from the stagnation point. We choose the impinging jet since it is a well-researched flow that may model some of the flow characteristics around a building envelope exposed to atmospheric winds. This setup allows both for the study of the stagnation point heat transfer characteristics as well as in those regions where the flow is parallel to the wall.

3.2 Envelope geometry

Two different sets of geometries were tested. The first set consisted of repetitive basic geometries, similar to the geometries that were examined in the 2D CFD simulation in the early stage of the research (Grobman and Elimelech, 2015) (figure 2).

The second set of geometries was based on selected 3D scans of cacti. The idea was to examine whether the cacti external geometry helps to develop a microclimate in a way that contributes to its thermal performance. From numerous cacti that were 3D scanned, 3 cacti with different geometry were selected. The 3D scans were rationalized and employed on a flat surface while keeping the exact proportions of the cactus envelope (see figure 3). Similar to the preliminary research, all simulations in this stage were performed on concrete walls, 30 cm thick, which is the typical width used for load-bearing concrete walls in buildings. Wind tunnel simulations were performed on 3D printed models made of gypsum that were printed using a Zcorp printer in 1:25 scale (Figure 3).

results involving sculptured plates, we anticipate that in future studies the relatively fast CFD simulations can be used to select the most promising results and geometries and wind tunnel experiments will only be used for validation purposes.

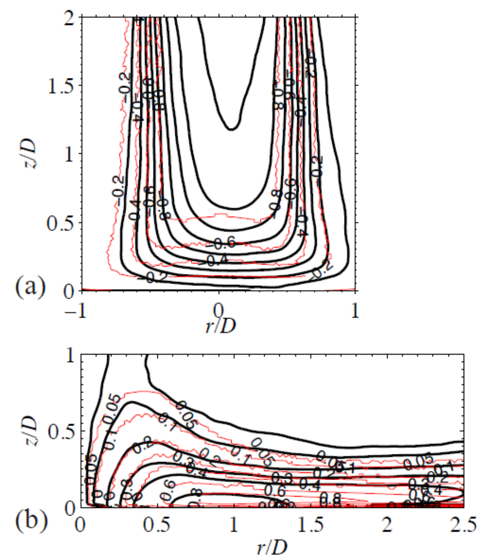

Fig. 4. Comparison between numerical simulations and experiments (Re = 6,260, U_j = 5 m/s). (a)$<U_z>/U_j$. (b) $<U_r>/U_j$. Experiments: Black solid iso-contours ; Simulation: Red solid iso-contours. Depicted contour levels relate to simulations but equally apply to experiments.

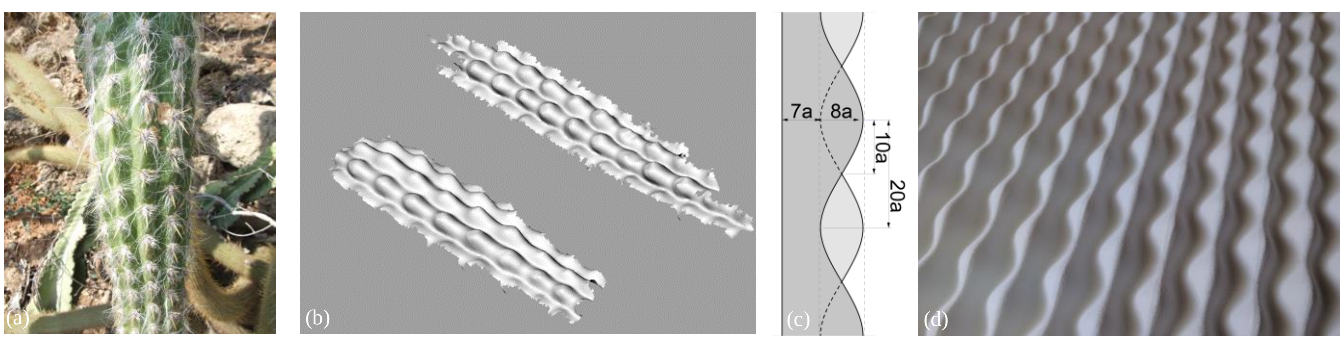

Fig. 3. Example of one examined cactus. (a) – type: Oero cactus; (b) – 3d model of Oero cactus created by a 3d scan; (c) – tile section, defined according to the exact proportions of the scan; (d) – a 3d printed gypsum model representing the cactus proportions.

4 PRELIMINARY RESULTS

As an initial step the measured average flow field of a turbulent jet impinging on a flat, smooth surface was compared to the CFD simulations. An example is provided in Figs. 4a and b that depict iso-contour levels of the normalized, ensemble averaged axial velocity field, $<U_z>/U_j$, and the radial velocity, $<U_r>/U_j$, respectively. Note that the $<U_z>$ is significant mainly in the incoming circular jet while $<U_r>$ is significant near to the wall in the radial direction. As observed, some differences exist but overall the comparison between the simulations and the experiments is satisfactory. Although these are intitial results and they will need to be further corroborated against

In addition, Fig. 5 shows the contour plots of the normalized Reynolds shear stress, $<u_r u_z>/U_j^2$, for a smooth target surface and a sculptured one (Fig. 2e). Here u_r and u_z are the velocity fluctuations in the r and z direction, respectively. Since $<u_r u_z>$ is directly associated with momentum transfer and can also be related to the heat transfer by the Reynolds analogy, differences in its distribution will have an effect on the heat transfer characteristics. While the distribution of $<u_r u_z>$ is similar in the incoming jet, the influence of the holes can be discerned in the insets in Fig. 5. The CFD simulation will be used in the next stage of the research to simulate and calculate the expected heat transfer or thermal insulation performance of the examined geometries and a comparison with the experiments will be made based on the Reynolds analogy.

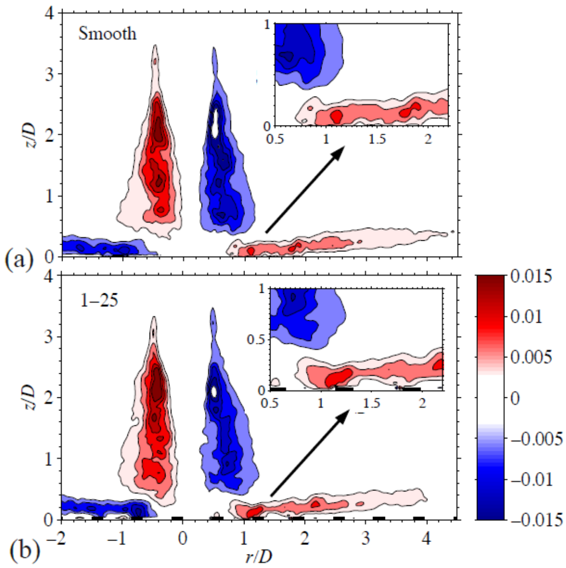

Fig. 5. Measured spatial distribution of the normalized Reynolds shear stress. (a) smooth plate, (b) plate with hemispherical dimples having a diameter of 10 mm (Fig. 2e). The impenetrable wall between the holes is indicated by thick black bars at $z/D = 0$ in (b).

5 CONCLUSION

This paper presented the initial results of a second stage of research that examined the potential of using geometry parallel to material to increase the building envelope's performance. The current stage focused on developing a set-up and a methodology for examining and comparing the results of a wind tunnel tests to those of CFD simulations. Initial results in the wind tunnel validated the correlation between both test paving the way to expand the CFD tests in future research towards examining, comparing and optimizing building envelope geometry contribution to the thermal performance of buildings.

ACKNOWLEDGMENTS

The research was supported by the Israeli Ministry of Construction and Housing - grant number 2022402/2015.

REFERENCES

1. Aydogan, A. (2012). Building-integrated active modular phytoremediation system (Doctoral dissertation, Rensselaer Polytechnic Institute). http://gradworks.umi.com/35/30/3530027.html.
2. Behnia, M., Parneix, S., & Durbin, P. A. (1998). Prediction of heat transfer in an axisymmetric turbulent jet impinging on a flat plate. International journal of heat and mass transfer, 41(12), 1845-1855.
3. Badarnah, L., Farchi, Y. N., Knaack, U., Carpi, A., & Brebbia, C. A. (2010). Solutions from nature for building envelope thermoregulation. Design & Nature V: Comparing Design in Nature with Science and Engineering, A. Carpi, CA Brebbia, Editors, 251-262.
4. Grobman, Y. J., & Neuman, E. (2011). Performalism: form and performance in digital architecture. Routledge.
5. Grobman, Y. (2013). Cellular Building Envelopes, in: Chakrabarti, A., Prakash, R.V. (Eds.), ICoRD'13, Lecture Notes in Mechanical Engineering. Springer India, pp. 951–963.
6. Grobman, Y. J., & Elimelech, Y. (2016). Microclimate on building envelopes: testing geometry manipulations as an approach for increasing building envelopes' thermal performance. Architectural Science Review, 59(4), 269-278.
7. Gruber, P., & Gosztonyi, S. (2010). Skin in architecture: towards bioinspired facades. WIT Transactions on Ecology and the Environment, 138, 503-513.
8. Knippers, J., & Speck, T. (2012). Design and construction principles in nature and architecture. Bioinspiration & biomimetics, 7(1), 015002.
9. Laver, J., Clifford, D., & Vollen, J. (2008). High performance masonry wall systems: principles derived from natural analogues. WIT Transactions on Ecology and the Environment, 114, 243-252.
10. Raffel, M., Willert, C. E., Wereley, S. T., Kompenhans, J. (2007). Particle Image Velocimetry, A practical guide. Second Edition Springer-Verlag Berlin Heidelberg, pp. 460.
11. Zuckerman, N., & Lior, N. (2005). Impingement heat transfer: correlations and numerical modeling. Transactions of the ASME-C-Journal of Heat Transfer, 127(5), 544.