Air Flow Visualisation Towards the Design of Breathing Skins

Manuel Muehlbauer[1], Nancy Y. Cheng[2], Mehrnoush Latifi Khorasgani [1], Jesse McCarthy[1] and Jane Burry[1]

[1]RMIT University
Melbourne, Australia
{manuel.muehlbauer,
jesse.mccarthy,
latifikhorasgani.mehrnoush,
jane.burry}@rmit.edu.au

[2]University of Oregon
Eugene, Oregon, USA
nywc@uoregon.edu

ABSTRACT

How can the geometric modification of surfaces affect airflow and enhance the thermal performance of ventilated facades? This study is an initial investigation into how surface articulation can affect air movement in a plenum between the shaped surface and the building, as a step toward the design of climate-specific ventilated facades, termed "Breathing Skins".

The research method describes how physical and digital procedures can play complementary roles in the understanding of complex environmental phenomena for architectural applications. Initial wind-tunnel tests measured the pressure change from the inbound side to the outbound side of a cavity with a variable shaped surface on one face. They allowed comparison of the frictional properties of a variety of folded surface forms, that could be used for façade siding or screens. Subsequently, the patterns were simulated with Computational Fluid Dynamics (CFD).

Results show that a solid sheet with concertina folds parallel to the wind revealed a greater loss in pressure and a Miura-Ori pattern a smaller loss in pressure than a flat sheet. These findings are shaping further research into the phenomenon by indicating useful avenues for the development of climate-modulating outer building skins.

Author Keywords

Computational fluid dynamics; folding surfaces; physical and digital simulation; patterned skins; ventilated facades.

ACM Classification Keywords

I.6.1 SIMULATION AND MODELING

1 INTRODUCTION

This research is a part of on-going research to investigate the potential of employing surface articulations or patterns for improving the thermal performance of building skins. It focuses on the consequences of a folded surface pattern on the movement of air close to the surface and within a cavity immediately behind the external layer of the Rear-Ventilated Façade. The primary intention is to modulate temperature and potentially provide both heating and cooling under different ambient conditions. Previous experiments [1,2] demonstrate that the surface shape can influence the passive heating of buildings' skins and alter the immediate external microclimate through self-shading.

Those studies were focused on developing empirical methods of investigating heat phenomena through engaging with the parallel physical and digital platforms along with employing a comparative scientific approach. They reveal the impact of surface articulations on reducing the amount of radiation received by the folded surfaces and changing amount of heat distribution on the double skin façade [2].

This study focuses on the application of folded continuous surfaces for improving the thermal performance of double skin facades by revealing the dynamics of air around them. This paper reports the results from the early experiments on observations and visualisations of the airflow in the cavity of ventilated facades by applying a similar hybrid physical-digital experimental method and framework.

2 BACKGROUND

Various experimental and numerical studies in engineering confirm that ventilated facades provide the opportunity of enhancing building energy efficiency by removing the heat inside wall plenums through air circulation and convection [3]. These thermodynamic and fluid dynamic studies reveal how the thermal improvement in such systems can be achieved by providing better air circulation inside the cavity.

They highlight the necessity of improving the cavity air flow to enhance the efficiency of the system. Summer Heat stagnation inside the cavity during summer can be prevented by altering the air density, leading air to flow upward, removing heat from the cavity [4]. This process reduces the amount of heat transmission from the cavity to the building [5].

Architectural studies explore different material applications for improving the efficiency of such systems [6]. However,

SimAUD 2016 May 16-18 London, UK
© 2016 Society for Modeling & Simulation International (SCS)

few studies have considered the impacts of parametric geometrical modification of surfaces within thermodynamic and fluid dynamic studies for architectural applications. Accordingly, our earlier research focused on heat conduction, convection and radiation in such building skins for their thermal performance optimisation through shape studies of surfaces [2, 7] and airflow measurement in porous screens systems, in which parameters such as surface roughness and porous shapes were manipulated parametrically.

The investigations were carried out in several software packages and physical platforms such as a small wind tunnel [2,8]. This current study is focused on the design of origami surfaces towards improving the thermal performance of ventilated facades by having the better understanding of the relation between the form of external shading devices and the airflow around them. The key aspect is to reveal the relationships between surface patterns and airflow changes around them that consequently affect heat exchange and convection in the cavity between the two layers of the façade.

2.1 Architectural Context

Understanding how a 3D surface relief pattern mounted on an exterior building wall can affect the airflow in the cavity behind it, requires review and evaluation of similar structures under different climatic conditions. In hot climates, a lightweight canopy is commonly used as a seasonal parasol, keeping direct sun off the roof of a building, and shading screens can be attached to the faces of an existing building. Opening the top of the cavity creates ventilation. We speculate that by manipulating geometries, we could maximize the chimney effect, potentially drawing cool air from an earth duct. For wet climates, shaped surfaces acting as rain screens could also benefit from enhanced cavity ventilation to reduce wood rot and fungal growth, a major source of asthma irritation.

In cold climates, a dark surface added to a sunny façade can collect heat and transfer it to the enclosed cavity between the added skin and the façade, so that warm air can be collected. Research on solar collectors is relevant because optimal flow in the cavity behind the solar radiation-absorbing surface is needed to facilitate the efficiency of the solar module. In the 1980's and 90's, Charles F. Kutscher of the National Renewable Energy Lab developed a solar collector that used a dark metal surface with perforations to allow a small volume of air to be slowly pulled by a fan through a heating chamber, known as an Unglazed Transpired Collector (UTC). This has been scaled up into the Conserval Solarwall (http://solarwall.com), a perforated metal second skin that is used for preheating intake air or crop drying.

Reviews of (UTC) research [9,10] examine the impact on efficiency of factors such as porosity, airflow rate, perforation pitch and diameter, wind approach velocity and material absorptivity and emissivity. While surface geometry is not directly addressed, Shukla [10] explains that the strongest factor influencing Heat Exchange Effectiveness (HEE) is the Reynolds number, created from the geometry and proportion of a flow cross-section. This research shows that the impacts of the surface geometry and perforations on flow efficiency have not been fully explored (Fig 1).

This study takes advantage of the fact that compared to a smooth plane; a textured surface causes more turbulence close to the plane, reducing drag. This study aims to explore how specific folding patterns affect airflow, and how we could adjust the folding and perforations for specific thermal comfort goals. This would be the first step in understanding how can we use folding and perforations 1) to maximize convective currents for cooling in hot climates and drying in wet climates and 2) to make UTC more effective for cold climates.

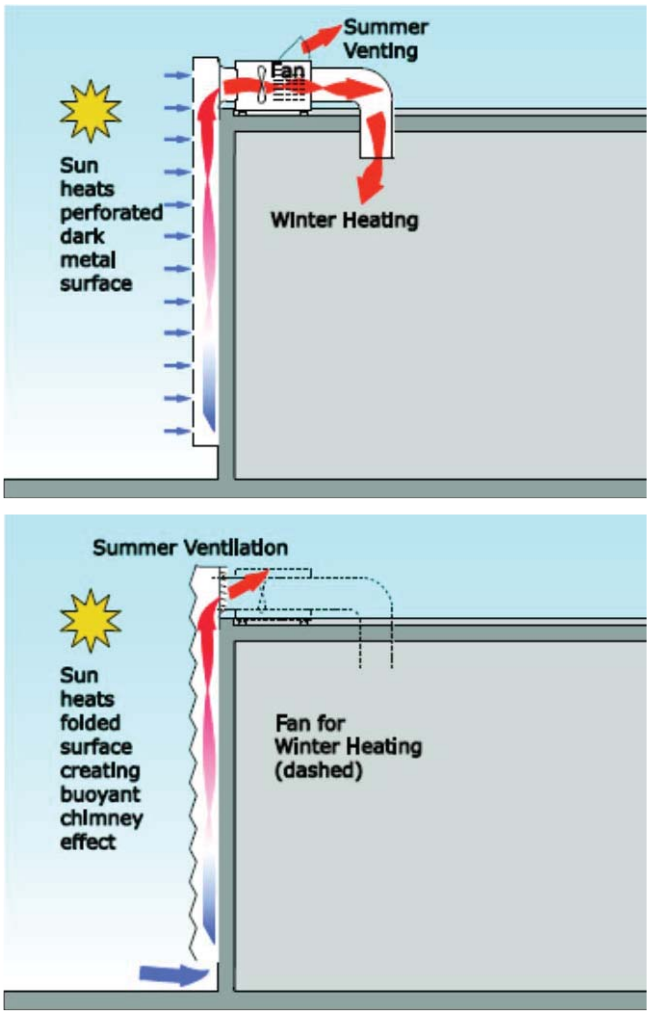

Figure 1. Active UTC wall & Passive continuous surface

4 METHODS

4.1 Analogue Experiment - Wind tunnel testing, gap cavity and geometry as explored variable

In order to understand how a folded screen mounted in front of an exterior building wall would affect air currents in the space between the screen and the wall, initial experiments were undertaken in an analogue wind tunnel. A number of experiments were done with both horizontal and vertical set up, to determine the effective parameters and further simplifications in the experiments.

A transparent acrylic “drawer” system 28 cm (11”) wide x 61 cm (24”) long x 15.3 cm (6”) high, allowed for easy adjustment of cavity depth, with substitution of different screens to form the top surface parallel to the direction of wind travel. The surfaces were inserted into the drawer set up, sealed along the edges to allow airflow only over the surface and through the cavity.

The experimental configuration for the first series of experiments, using the same variable cavity depth system and card surfaces, was to mount the surface vertically, with a gap at the base and open to the top, and heat the surface from the front with a heat lamp (Fig. 2).

Temperature and pressure measurements were taken at the base and top of the cavity to determine whether there were detectable convection currents set up by the surface heating and whether the differential self-shading in different patterns influenced the measurable magnitude.

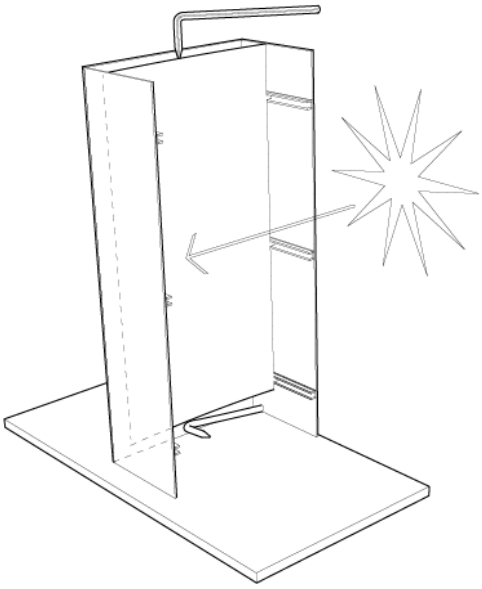

Figure 2. Vertical analogue experiment setup

Because convective buoyancy induced by solar radiation or heat lamps on a dark screen in a vertical position was too subtle to measure consistently, efforts were focused on examining flow through a horizontal chamber in a wind tunnel. Visualizing the flow using smoke and light sources (Fig. 3) helped identify the relevant parameters involved in the modelling turbulence in the cavity between screen and wall.



Figure 3. Smoke and light source under folded surface in horizontal setup

In the second series of experiments with horizontal setup (Fig. 4), the surfaces were laid horizontal in the mid flow of a wind tunnel, so that the influence of surface geometry on airflow could be examined independently from heat-induced effects.

The flow behind surfaces with different fold patterns (Fig. 5) was benchmarked against flat sheet panels by taking pressure measurements at the leading edge and at the exit point of the cavity for four different wind speeds.

The focus was a consistent experimental setup for comparison of different shapes, rather than for absolute metrics. First flat, then concertina folded, and then more complex folding patterns, created from white and black cardstock were tested.

Variables were: cavity depth for a particular surface shape (2.5, 5.0 and 7.5 cm or 1”, 2” and 3”), depth of folded surface for a particular folding, different surface patterns for consistent cavity and surface depth. The wind tunnel experiment was repeated several times, improving the data gathering technique each time.

The analogue experiments revealed the need for more continuous data and higher-resolution visualisation of the turbulence.

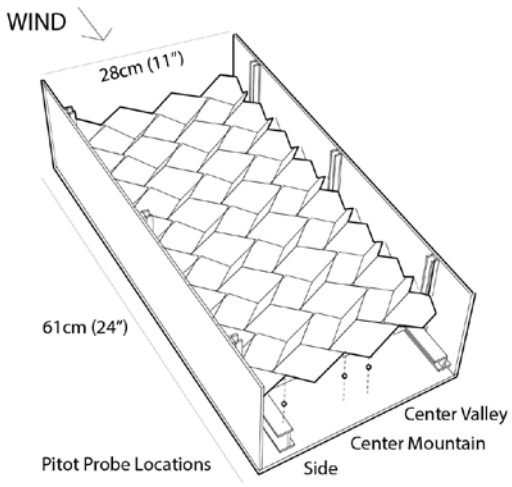

Figure 4. Horizontal analogue experiment setup

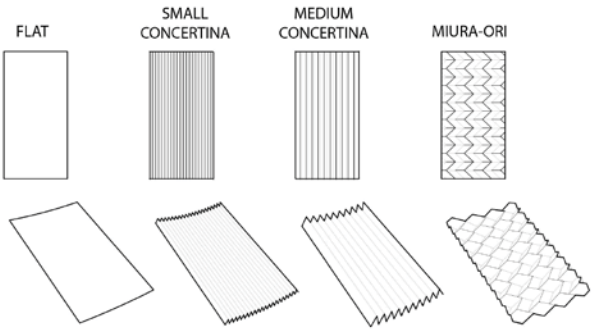

Figure 5. Effective typology

4.2 Computational Fluid Dynamics

Computational Fluid Dynamics (CFD) is a numerical tool utilised to simulate fluid flows and other fluid-related phenomena such as heat transfer, magneto-hydrodynamics, aero-acoustics, and Fluid-Structure Interactions (FSIs). It has typically seen widespread application in engineering disciplines involving fluid physics [13], but is now rapidly becoming a useful tool in the architectural design process [14-18].

The core purpose of any CFD code is to arrive at a numerical solution of the governing equations of fluid dynamics over a discretised domain – commonly known as a mesh, representing the geometry and flow conditions of interest. A key advantage of CFD is the ability to interface with CAD packages, which can permit quick parameterisation of geometric variables of interest while allowing a wide exploration of the design space in a relatively small amount of time.

Turbulence is present in almost every realistic flow in nature and may be either modelled or resolved in CFD. Resolving the turbulent flow structures in a representative flow requires both stringent grid resolution requirements, and depending on the method by which the turbulence is resolved, orders of magnitude higher computing expenditure than if the turbulence were modelled using a suitable turbulence model. Refer to such works as Wilcox [11] for a more elaborate explanation of turbulence modelling.

4.3 Preliminary CFD Analysis

CFD analyses utilising a Reynolds-Averaged Navier-Stokes (RANS) model were performed to correlate with the analogue experiments. They replicated the horizontal analogue setup to visualise the air pressure and velocity, again ignoring the buoyancy impacts for simplicity and consistency. The CFD code employed was ANSYS® Fluent, which is commercially available and has been extensively validated for many canonical flows including those with turbulence and heat transfer mechanisms. Full-scale CAD geometries of the wind-tunnel specimens were meshed with structured grids that were each tested for mesh independence of results. The boundary conditions for the flat-plate setup are shown in Fig. 6, with the flat plate being replaced by the folded or concertina plate in the other two cases. Reynolds numbers for the CFD were similar to the 5 cm cavity depth experiment at 4 m/s.

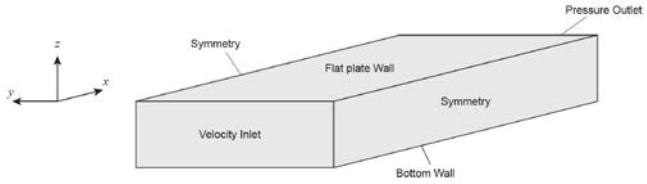

Figure 6. Boundary conditions for flat-plate setup

At this stage of the investigation, the purpose of employing CFD was to rapidly explore the design space through parameterisation of the CFD models. Three geometries – flat sheet, concertina and Miura-Ori pattern that had been tested in the wind tunnel were then simulated with CFD.

Here the realizable k-ε RANS turbulence model was chosen for its good performance in wall-bounded flows with pressure gradients [12], and was used to predict the turbulence levels throughout the computational domain. Because heat transfer mechanisms were not taken into account in this stage, only the flow mass and momentum equations are solved.

5 RESULTS

5.1 Reporting Results of Analogue Experiment

Horizontal pressure drop measurements taken at 5.0 cm screen height reveal that the concertina folds create a great deal of friction in the chamber, whereas the Miura-Ori pattern creates less friction than a flat screen (Fig. 7).

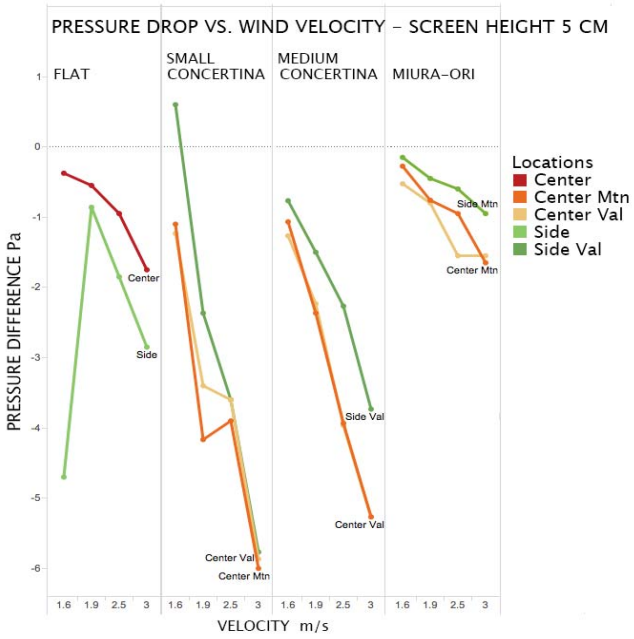

Figure 7. Analogue results – Pressure drop vs. wind velocity

Outside of the pressure measurements, working in the wind-tunnel provided an intuitive sense of turbulence. Smoke pencil trails revealed qualitative aspects of the airflow at the chamber entrance. Due to smoke dispersion, a different configuration would be needed to capture downstream behaviour.

5.2 Reporting Results of Preliminary CFD

The pressure distribution contours for the geometries are shown in Fig. 8-10, along with the calculated pressure drop. The gauge pressure is plotted, which is defined by:

$$P_{gauge} = P_{absolute} - P_{atmopsheric},$$

where P_{gauge} is the gauge pressure, $P_{absolute}$ is the local absolute static pressure and $P_{atmopsheric}$ is the atmospheric pressure.

Immediately noticeable is the disparity in pressure distribution with the Miura-Ori geometry, as compared to the other two geometries. This is due to the variation in cross-sectional area along the length of the geometry, which changes the pressure gradient of the flow. As the cross-sectional area decreases, the flow velocity increases and subsequently the static pressure decreases. The opposite is true when the cross-sectional area increases. This becomes an important consideration when introducing porosity into the geometry; depending on location, outside air may either be entrained into the duct, which would occur in areas where the local static pressure is lower than atmospheric, or air could be expelled from the duct in places where the local static pressure is greater than atmospheric pressure.

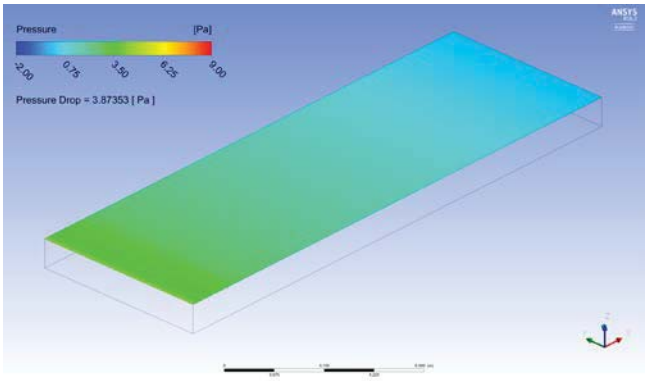

Figure 8. Pressure distribution contours for flat sheet

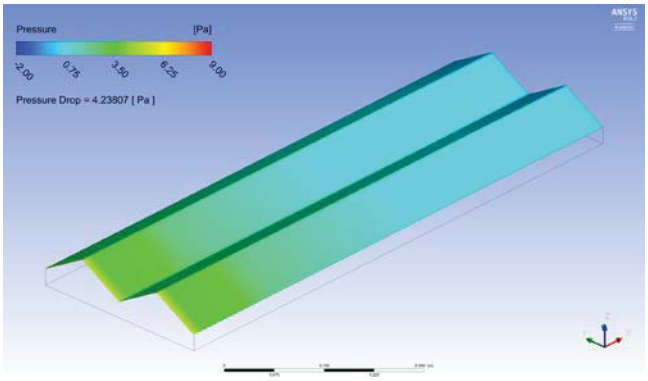

Figure 9. Pressure distribution contours for concertina

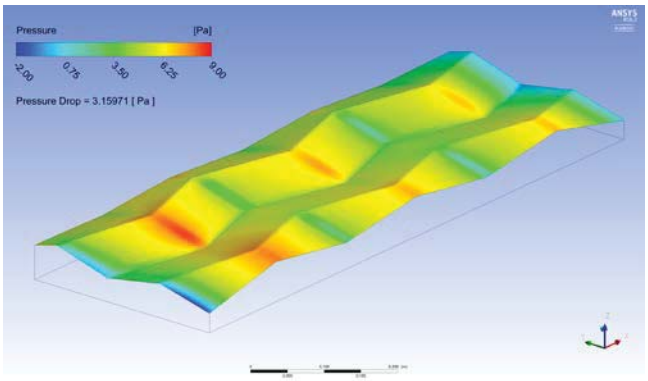

Figure 10. Pressure distribution contours for Miura Ori

Interestingly, the medium concertina plate exhibited the greatest pressure drop. This is most likely due to the increased viscous losses occurring near the apexes of the geometry. Conversely, the Miura-Or shows the least pressure drop, likely due to the varying cross-sectional area of the duct, preventing boundary-layer development. This is a somewhat similar phenomenon that is exploited with golf balls; that is, the dimples delay boundary-layer growth and subsequent separation, reducing drag and allowing for a longer trajectory. This result is visualised in Fig. 11-13,

which compares the velocity distributions in the three geometries. It can be seen that the Miura-Ori geometry does not allow for steady growth of the wall boundary layers, in contrast with the flat plate and medium concertina geometries.

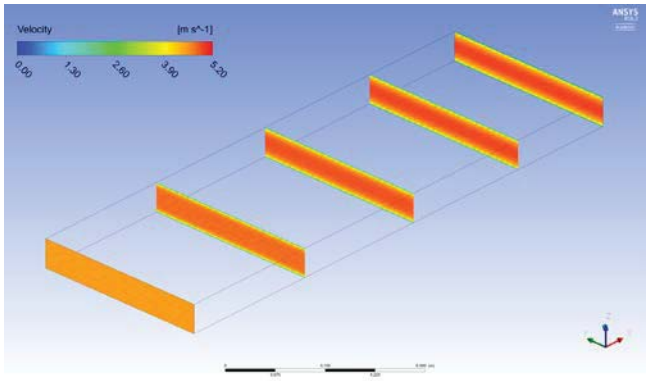

Figure 11. Velocity distribution for flat sheet

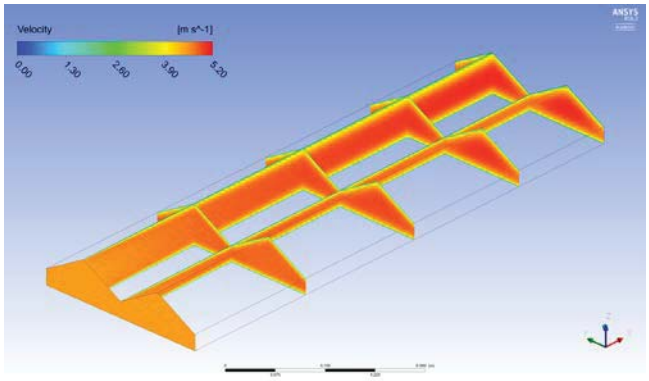

Figure 12. Velocity distribution for concertina

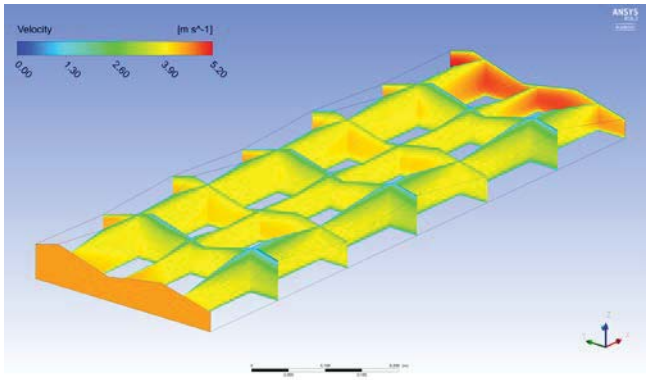

Figure 13. Velocity distribution for Miura-Ori

6 DISCUSSION

The range of experiments provided insight into how the analogue and digital tools complement each other.

The small wind tunnel used is not as accurate as a large-scale wind tunnel experiment, but it allows designers to get some quick initial insights into airflow. An accessible wind tunnel and heat lamp allow sensory perception of environmental phenomena such as lift and heat absorption that might not be evident in a digital simulation. But the analogue experiments accrued some measurement errors. For example, closer observation revealed that the outbound measurements for the 2.5cm height were affected by the location of a support bar, so the 5.0 cm measurements were more reliably compared. In addition, the reliability of the measurements might have been affected by the close proximity of the wind-tunnel walls.

6.1 Limitations of the study - CFD as Approximation

CFD provided insights on the turbulence pattern inside the façade cavity. It is important to recognise that at its core, CFD is a mathematical model that solves the governing equations of fluid dynamics, which in turn are subject to conservation laws of fluid physics. But a mathematical model is only as good as its inputs and simplifying assumptions. One of the major assumptions made in the realizable k-ε turbulence model (and two-equation turbulence models in general) is that the turbulence viscosity is isotropic, which is not strictly true [20]. However, this assumption still holds well for a number of technical flows, such as the wall-bounded flows analysed in the present study. A synopsis of the relative complexity of CFD models is summarised in Fig. 14.

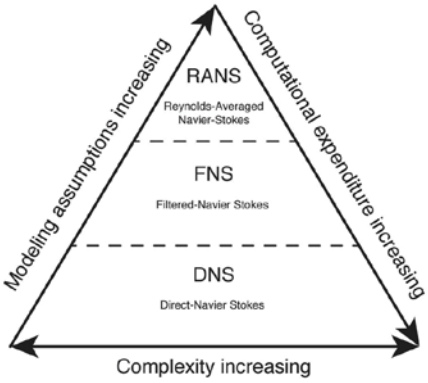

Figure 14. Complexity of Navier-Stokes Equations

Here, a RANS model was employed to visualise the flow-field within the three plate designs. However, in future studies, where detailed analyses of turbulent structures in a particular facade design are required, a turbulence resolving CFD model such as an FNS model will be used.

6.2 Major Findings - Parameters and decision variables defined and explored

During the series of experiments, we explored different wind velocities, cavity depth and surface geometries as decision variables. With typical Reynolds Numbers for experiments being around 9,500, fully turbulent flows are expected inside the cavity for all three categories of geometry. Furthermore, the optimal façade cavity was determined experimentally at about 50mm for the flat sheet

and at the average cavity depth for the other geometries respectively.

First insights from the preliminary CFD study reveal the potential for strategic positioning of perforation in low-pressure areas with potential flow entrainment into the façade cavity. The Muria-Ori pattern showed the lowest pressure drop and will be further investigated, as the air can pass through the cavity less restrained in case of forced convection. This case is likely, regarding the position of the façade, where the top edge aligns with the leading edge of the façade. The impact of this position and inlet/outlet conditions are different geometric decision variables that will be explored in further experiments. Based on the results it is predicted that applying a folded pattern on the inner layer of the cavity could also change the results – one area for further study.

7 CONCLUSION

The described experiments show ways to study how surface geometry influences the flow of air in a plenum between the surface and the building. Both wind tunnel and CFD studies show that shape of surfaces significantly affects airflow: the impacts can be measured in both low fidelity analogue airflow testing and high fidelity digital simulation. Specifically results showed that a solid sheet with concertina folds parallel to the wind revealed a greater loss in pressure and a Miura-Ori pattern a smaller loss in pressure than a flat sheet. They provide initial evidence for developing optimisation strategies for such variables as cavity depth, surface shape, and angles of folding.

The work also illustrates the value of a cycle of analogue and computational experimentation, each informing the development of the other without the need to close couple numerical results in each context.

The results have provided the evidence and direction for further refined studies, progressively adding to the complexity of the models by introducing different materials, surface finish, and controlled and differentiated perforation for porosity, as well as examining how the sun's heat changes the flow dynamics. The Computational Fluid Dynamics environment will continue to provide an extensive iteration and versioning cycle as a means to narrow down the most fruitful areas for further analogue experimentation.

8 FUTURE WORKS

The presented research is a significant step in solving the question of how turbulence in the cavity of ventilated facades influences the thermal convection inside the cavity. Multi-Criteria Optimization used in various engineering applications [21-23] has the potential to further improve the design of façade surfaces.

For example, in a holistic optimization strategy, the façade surface geometry could be targeted towards summer interior comfort through self-shading and cavity ventilation, along with sheltering passing pedestrians from excessive wind (Fig.15).

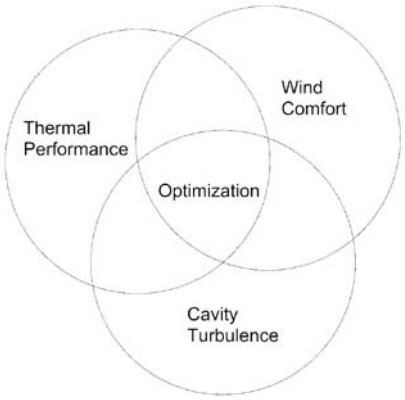

Figure 15. Venn-diagram of optimization strategy

Currently, more complex morphologies than isolated buildings in an open field, such as, surface effects and porosity influencing turbulence, generally require detailed, computationally heavy and time-consuming simulations [7]. Future directions for the research include investigating the development of lightweight CFD simulation to accomplish meaningful results for façade optimization processes in early design.

9 ACKNOWLEDGEMENT

Thanks to John Breshears of Architectural Applications; and G.Z. Brown, Kevin Van den Wymelenberg, Jeff Kline and Dale Northcutt of the Univ. of Oregon Energy Studies in Buildings Lab for consulting advice and Jesse Elliott, Marziah Rajabzadeh, Kristin Potter and Francisco Toledo for assistance on the wind-tunnel testing and graphics visualization. Support from the Global Oregon Faculty Collaboration Fund catalyzed this research project.

We would like to acknowledge the support of the Australian Research Council for the project: Integrating architectural, mathematical and computing knowledge to capture the dynamics of air in design which has provided the context and funding for this research.

REFERENCES

1. Williams, M. et al. A Physical and Numerical Simulation Strategy to Understand the Impact of the Dynamics in Air for the Design of Porous Screens, *SimAUD '15 Proceedings of the Symposium on Simulation for Architecture & Urban Design*, 12 April 2015, 43-50.
2. Khorasani, M. L.; Burry, J.; Salehi, M. Thermal performance of patterned facades-studies on effects of patterns on the thermal performance of facades, *Proceedings of the 32nd eCAADe Conference – Fusio, Volume 1*, Netherlands, 10-12 September 2014, 267-276.

3. Lai, C. M., & Lin, Y. P. Energy saving evaluation of the ventilated BIPV walls. *Energies, 4(6),* 14 June 2011, 948-959.

4. Di Maio, F., & Van Paassen, A. H. C. Simulation of temperature and air flow in a second skin façade. *Proceedings of Roomvent 2000, Air Distribution in Rooms: Ventilation for Health and Sustainable Environment, Volume 2,* Oxford, Elsevier, Reading, UK, held 9-12 July 2000, 943-949.

5. Gan, G. Simulation of buoyancy-induced flow in open cavities for natural ventilation. *Energy and Buildings, 38(5),* 2 August 2005, 410-420.

6. Bechthold, M., Kane, A. & King, N. Ceramic Material Systems. *Architecture and Interior Design.* Berlin, Basel: Birkhäuser, 10 March 2016.

7. Burry, J., Salim, F., Williams, M., Anton Nielsen, S., Pena de Leon, A., Sharaidin, K., & Burry, M. Understanding heat transfer performance for designing better facades. *ACADIA 13: Adaptive Architecture Proceedings of the 33rd Annual Conference of the Association for Computer Aided Design in Architecture,* Delaware, United States, 24-26 October 2013, 71-78.

8. Moya Castro, R. Wind analysis in the early design stage: an empirical study of wind visualisation techniques for architects, Doctor of Philosophy (PhD), Architecture and Design, RMIT University, March 2015.

9. Leon, M. A.; Kumar, S. Mathematical modeling and thermal performance analysis of unglazed transpired solar collectors. *Solar Energy, Vol.81(1),* January 2007, 62-75.

10. Shukla, A.; Nkwetta, D. N.; Cho, Y.J.; Stevenson, V.; Jones, P. A state of art review on the performance of transpired solar collector. *Renewable and Sustainable Energy Reviews, Vol.16(6),* August 2012, 3975-3985.

11. Wilcox, D. C. Turbulence modelling for CFD. 2nd ed. DCW Industries. La Cañada, ISBN 0963605100, 1998.

12. Shih, T.-H.; Liou, W. W.; Shabbir, A.; Yang, Z.; Zhu, J. A new *k-ε* eddy-viscosity model for high Reynolds number turbulence flows - Model development and validation. *Computers Fluids,* 24(3)., March 1995, 227-238.

13. Tu, J.; Yeoh, G. H.; Liu, C. Computational fluid dynamics: a practical approach. Butterworth-Heinemann, 2013.

14. Kaijima, S.; Bouffanais, R.; Willcox, K.; Naidu, S. Computational Fluid Dynamics for Architectural Design. *Architectural Design, Vol.83(2),* 12 March 2013, 118-123.

15. Chen, Q. Using computational tools to factor wind into architectural environment design. *Energy and Buildings, 36(12),* 2004, 1197–1209.

16. Chen, Q.; Jelena., S. Application of cfd tools for indoor and outdoor environment design. *International Journal on Architectural Science, 1(1),* 2000, 14–29.

17. Jal, E. N. Applying cfd to environmental flows. Technical report, Connell Wagner Pty Ltd, South Melbourne, Victoria Australia, December 2003.

18. Kirkegaard, P. H.; Hougaard, M.; Staerdahl, J. On Computational Fluid Dynamics Tools in Architectural Design. Technical Report 56, Department of Civil Engineering, Aalborg University, 2008.

19. Capobianci, M.; Irvine, Jr. T. F. Triangular Ducts, Flow and Heat Transfer, *International encyclopedia of heat & mass transfer,* CRC Press, Boca Raton, 1997.

20. Hinze, J. O. Turbulence. McGraw-Hill Publishing Co, 1975.

21. Preisinger, C.; Vierlinger, R.; Heimrath M. Strategies for Combining Structural Analysis and Parametric Geometries, in possession of the author, 15 January 2015.

22. Yang, D.; Sun, Y.; Turrin, M.; Buelow, P. von; Paul J. Multi-objective and multidisciplinary design optimization of large sports building envelopes: a case study, *Proceedings of the International Association for Shell and Spatial Structures (IASS),* 2015.

23. Danhaive, R. A.; Mueller, C. T. 'Combining parametric modeling and interactive optimization for high-performance and creative structural design, *Proceedings of the International Association for Shell and Spatial Structures (IASS),* 2015.