2018 Building Performance Analysis Conference and
SimBuild co-organized by ASHRAE and IBPSA-USA
Chicago, IL
September 26-28, 2018

INTEGRATION OF CFD SIMULATIONS IN COMPUATIONAL DESIGN FOR HARNESSING THE NATURAL VENTILATION PERFORMANCE OF TYPICAL ATRIUM SPACES IN ATHENS, GREECE

Angelos Chronis[1], Foteini Stefopoulou[2], and Katherine Liapi[2]
[1]Institute for Advanced Architecture of Catalonia
[2]University of Patras

ABSTRACT

This paper investigates the integration of CFD in computational and parametric design and its potential in driving the optimization of the natural ventilation performance of buildings, using as a case study the courtyard spaces of the typical multistorey buildings in the historic center of Athens, Greece. The study employs a multi-scalar simulation approach and shape optimization techniques to assess the potential of using CFD to drive the design of retrofitted membranes that enhance the natural ventilation of the unused courtyards.

By parametrically modeling a membrane component on a representative courtyard case and using a Genetic Algorithm (GA) optimization along with a Fast Fluid Dynamics (FFD) CFD solver that allows for fast iterative CFD simulations, the study aims to produce localized retrofitting interventions to increase the air quality of the courtyards and the ventilation performance of the buildings.

INTRODUCTION

It is generally acknowledged that climate change due to gas emissions poses a significant threat to our wellbeing. Despite the continuous development of more sustainable methods for design and construction, buildings are still responsible for nearly half of the energy consumption and the CO2 emissions annually (UNEP 2016). Currently, heating energy demand is the main contributor of greenhouse emissions, but cooling energy demand which is mostly neglected is constantly rising and it is projected that by 2060 it will overcome that of heating (Peters, 2016). Taking into account the increasing cooling demand of hot developing countries in relation to colder developed ones, it is clear that more sustainable cooling strategies need to be undertaken.

In respect to the development of more sustainable design strategies, from a computer aided architectural design (CAAD) point of view, in the past decade there has been a continuous and rigorous development and integration in current design systems of building performance simulation (BPS) tools, including significant advancements in energy simulation, solar radiation and shading, to name a few. BPS tools are constantly incorporated in design processes, driving more sustainable buildings. However, similarly to the generally neglected cooling demands, cooling design strategies have not been as thoroughly investigated, also due to the complexity of the relevant computational fluid dynamics (CFD) simulations. CFD simulations are primarily used for assessing natural ventilation performance goals, but due to lack of integration in CAAD systems and their high computational demands, their use is restricted to mainly validation of design assumptions. To achieve though a significant optimization in the cooling performance of buildings, cooling strategies need to be addressed early on in the design process. Moreover, the wide use of computational optimization methods allows designers today to explore substantially larger design spaces. It follows that the integration of CFD with computational design methods would allow substantially greater optimization of cooling demands.

This paper investigates the integration of CFD in computational and parametric design and its potential in driving the optimization of natural ventilation performance in buildings, using as a case study the courtyard spaces of the typical multistorey buildings in the historic center of Athens, Greece. The study employs a multi-scalar simulation approach and shape optimization techniques to assess the potential of using CFD to drive the design of retrofitted membranes that enhance the natural ventilation of the unused courtyards.

By parametrically modeling a membrane component on a representative courtyard case and using a Genetic Algorithm (GA) optimization along with a Fast Fluid Dynamics (FFD) CFD solver that allows for fast iterative CFD simulations, the study aims to produce localized retrofitting interventions to increase the air quality of the courtyards and the ventilation performance of the buildings. The study's focus is mainly on assessing the integration of CFD in computational design and the feasibility of employing shape optimization techniques for CFD-driven geometry in buildings as well as on developing a methodology for employing shape optimization techniques for the given problem.

BACKGROUND

CFD simulations have been used in various studies to assess and optimize the air quality of courtyard and atria in hot climates. CFD has been used to estimate the cooling loads of atria in hot and humid urban environments (Pan et al 2010) and to model the buoyancy driven natural ventilation of lightwells in high rise buildings (Farea et al 2015). Both of these studies are quite relevant to the one presented here, although their focus is more on the CFD modeling and calculation of cooling performance rather than using CFD as a design driver. CFD simulations have also been used in many occasions for passive down-draught evaporative (PDEC) cooling for office buildings for example (Robinson et al 2004), for optimizing the energy efficiency of façade systems with open atria (Bai et al 2015) and thermal comfort evaluation on building atria (Hussain and Oosthuizen 2013) to name just a few.

In our study, the focus is on optimizing the air quality of the currently unused courtyards of old multistorey buildings in central Athens by minimal retrofitting of membrane surfaces that would increase the air change and make these spaces more habitable while also enhancing the natural ventilation potential of the buildings. Retrofitting strategies as a means for optimization of energy consumption of buildings is also common ground, and given the current challenges of the construction industry in countries like Greece, retrofitting studies are increasingly proposed as sustainable alternatives to new constructions (Eliopoulou and Mantziou 2017).

METHODOLOGY

CFD Simulations

As a first step, CFD simulations for the prevailing summer wind direction have been performed on an extended surrounding area of a chosen typical block to assess the effect of the building massing on the airflow and to identify the different wakes caused by the massing to be later used to drive the design of the retrofitting components. The CFD simulations were run with the established Cham PHOENICS solver, through its Rhinoceros3D plugin, RhinoCFD to facilitate an integrated CAAD design approach. RhinoCFD allows designers to run CFD simulations natively in one of the most well established CAAD environments, thus it is chosen primarily for its integration, as well as its established accuracy of the solver.

The CFD simulations were run for the prevailing wind direction of the summer months (NNE) and for the average wind speed of 6 m/s, according to the statistical data of the Athens International Airport weather station. A logarithmic wind velocity profile with fixed mass flow was used for the inlet, and the boundary conditions were set to fixed pressure for the downwind domain faces and open sky (fixed pressure with diffusive links) for the upper face boundary and ground with an effective roughness height of 2.0E-04 representing the dense urban environment of the area. These boundary conditions are set by default when the WIND domain object is introduced in PHOENICS/RhinoCFD. The simulation domain was extended at 5 times the height of the area of interest for the upwind, lateral and upper boundaries and 8 times the height for the downwind boundary, as per simulation standards, to capture the wind flow with adequate accuracy.

This initial CFD study showed that airflow patterns at the higher parts of the courtyards could be harnessed to increase the air quality of lower parts with minimal retrofitting. The initial large scale CFD simulation of the area also shows the effect of the building's orientation with relation to the prevailing winds and the street canyons on the natural ventilation performance of the buildings, calling for a localized retrofitting that adapts to the local flow conditions of each courtyard (Figures 1&2).

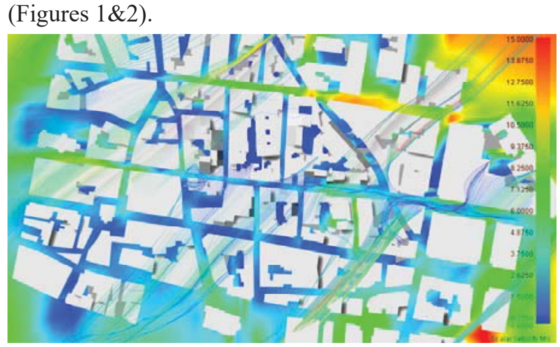

Figure 1 CFD Study of the Prevailing Wind Direction (NNE) – Velocity Contours and Streamlines Plan

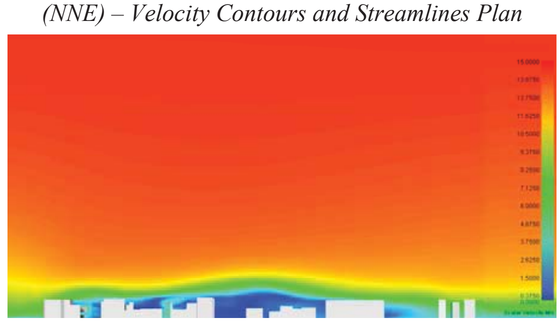

Figure 2 CFD Study of the Prevailing Wind Direction (NNE) – Velocity Contours Section

To achieve a localized response to the flow conditions and introduce a building element that can fully harness the ventilation potential, shape optimization techniques are employed to create an optimum form that will act as a wind catcher. The shape of this wind catcher membrane is driven by an iterative optimization study using CFD and a Genetic Algorithm. The shape of the membrane is driven by its performance in relation to maximizing the air change at the lower parts of the courtyard (up to 3 meters from the ground).

Fast Fluid Dynamics

At their current state, most CFD solvers that are exposed to CAAD environments have none or limited shape optimization capabilities. OpenFOAM, which is exposed to Rhinoceros3D through its plugin Grasshopper and the add-on Butterfly has only limited shape optimization methods, such as its integrated adjoint solver, which is also not exposed to Grasshopper. Other software, such as ANSYS Fluent have more extended optimization capabilities but the software is also not integrated with CAAD and has limited use in the architectural industry, also due to its very high cost. RhinoCFD, which was used for the context simulations has no optimization capabilities.

In this study, in order to employ shape optimization methods, we use a lower order, faster and not as accurate CFD solver (Stam 1999), based on a simplified, lower-order solver which is though still following the physics of the Navier-Stokes equation and which has been previously used in a number of studies and has been validated for a number of objectives (Zuo and Chen 2007, Chronis et al 2011, Karagkouni et al 2013, Waibel et al 2017).

As stated earlier, at its current state, the study does not aim to produce an accurate, to size, retrofitting solution, but mainly to assess the feasibility of employing shape optimization methods for natural ventilation in an integrated CAAD-CFD approach. The solver used is developed by the authors, based on the solver by Stam (1999), and it is integrated with Grasshopper, although for the purpose of this study it is running externally, as a Java application.

A validatoin of the FFD solver is beyond the scope of this paper which is focusing on the employment of shape optimization methods, based on the validity of previous work and given that the aim is to employ the shape optimization methods at an early design stage.

Genetic Algorithm

For the optimization part, a Genetic Algorithm (GA) is employed, in order to explore the maximum potential of the design space of the membranes. Other optimization methods, such as the gradient descent and simulated annealing methods can be employed within the same methodology. However for the purpose of this study the GA is considered the best fit, given the complex geometrical definition of the problem. The GA is also developed by the authors and also runs within the Java application. The encoding of the geometry on the genes is done per parameter (each parameter in the parametric definition is a gene) and the evolution is done with a multiple point crossover method and a 5% mutation rate. The fitness function was set as the sum of the velocity of all the cells up to 3 meters from the ground, in both 2D and 3D simulation models:

$$f = \sum_{i=0}^{n} u(i) + v(i)\ \{+w(i)\}$$

Where u, v {and w} are the x, y {and z} velocity of the wind for the n number of cells of the 2D and 3D cases respectively.

Multi Scalar Approach

In order to assess the potential and feasibility of employing shape optimization for the given problem, a multi-scalar approach was followed. Optimization runs were performed for both a 2-dimensional as well as a 3-dimensional representation of the problem, each with a different parametric description of the geometry, as well as with both a greater context area and a limited context area. These compromises were deliberate as the computational demand of such a feat is normally beyond the capabilities of a computer which is accessible by most designers. Most optimization runs were restricted to running for a maximum of 8 hours, to make their employment realistic in an architectural design scenario and specifically during the early stages of design.

Both the 2-dimensional and 3-dimensional optimization runs were performed using the same solver and GA, integrated in the Java applet, which allows for both domains to be set. All simulations and optimization were run on a Windows PC with 8 GB of memory and four Intel i7 processors at the speed of 2.4 GHz.

It's worth also mentioning that the user interface of the Java applet allows the designer to visualize the CFD simulation as it's running, as well as to observe the fitness plot of the GA and the surface population, so that he can understand if both the optimization as well as the CFD run towards convergence (Figure 6).

Parametric Description of Geometry

The parametric description of the retrofitted membrane geometry was different in 2D and 3D although they both followed the same main principle, which is a funnel-looking shape that has the potential to drag air in the courtyard from all wind directions. At this stage the optimization is only done for the prevailing wind direction, for computational economy, but at a later stage, all wind directions can be evaluated within the same methodology.

For the 2D representation of the problem, the surface of the membrane was defined as two curves, which would form the cross-section of the membrane on the windward and leeward sides of the wind flow. The length (l), the height of each curve (h1, h2), the distance of the funnel in relation to the buildings surrounding the courtyard (d), as well as the curvature of the two cross-section curves (p1 & p2) were all encoded as genes of the Genetic Algorithm (Figure 3). The definition of the surface is deliberately topological, rather than purely geometrical, as would be the case if the points of the curves were defined by their x and y components, as the Genetic Algorithm converges faster with a topological definition and the evolution and mutation processes are more relevant to the problem.

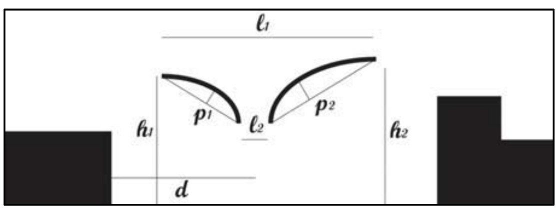

Figure 3 2D Representation – Parametric Definition of the Membrane as a 2D cross-section

For the 3D representation, the surface of the membrane was also defined by the dimensions and the position of the funnel in relation to the buildings, as previously explained, but also by a parametric conical surface that was varied at given intervals to produce ripples on the surface that can capture the wind from different directions and drag it into the courtyard with increased speed. The parametric variations were encoded as an array of parameters {p(i)} that offset the conical geometry at the set intervals (Figure 4). A number of different variations of intervals and deformations were developed to be tested through the GA and FFD optimization process.

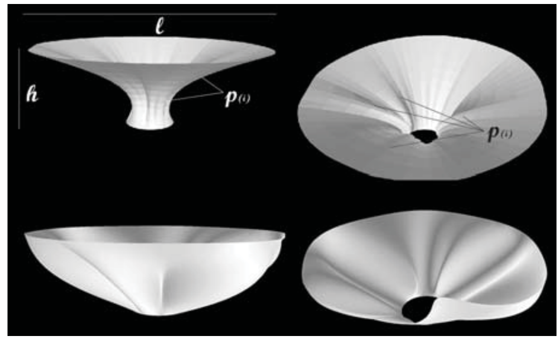

Figure 4 3D Representation –Parametric Ripples generate different parametric variations of the membrane

RESULTS

2D Representation with Limited Context

The first experiment, with the 2D representation and a limited context had the most promising results, as the GA clearly converged to specific shapes and performance gains. The resolution of the domain was high enough to capture vortices and airflow patterns, although these remain to be validated with more accurate solvers. Nevertheless, with the accuracy of the given solver, the GA clearly converges to a global optimum and a single solution for the optimization of the membrane despite the high mutation ratio of the GA. Figure 5 shows the simulation environment, where the captured flow is clearly seen as well as the GA population representation, where at the bottom 2 rows the converged solution is dominating the fittest members. Figure 6 shows the fitness plot, where the convergence to a global optimum is also visible, with just the 5% mutation rate offsetted from the performance range.

The 2D optimization runs were also the fastest of the set, with a runtime of a couple of hours being enough for convergence of the GA. This first test's results were quite promising leading to the conclusion that at least with the given accuracy compromises, the GA and FFD are able to provide a specific answer to the defined problem.

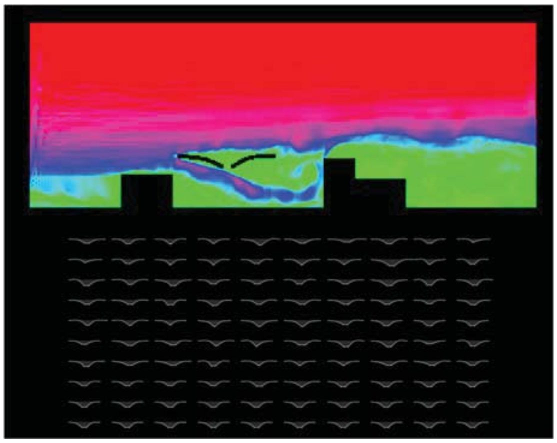

Figure 5 2D Representation with Limited Context – CFD Simulation and Surface Population

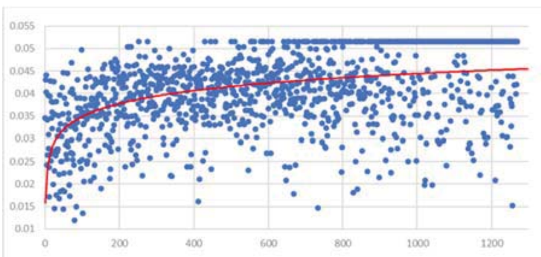

Figure 6 2D Representation – Fitness Plot

2D Representation with Extended Context

The second experiment was again done with the 2D representation but with an extended context which covered at least 2 bloks surrounding the area of interest. The GA again clearly converged to a global optimum, with a specific shape and performance gain and although the resolution of the solver was high enough to capture the flow in the overall area, the specifics of the flow inside the courtyard were not accurately resolved. This was obvious both in the simulation visualization as well as supported by the resulting surface which acted more as a blockage element, rather than directing the flow inside the courtyard as can be seen in Figure 7. Figure 8 shows how the fitness plot is indeed converging to a global optimum in both runs.

One of the main reasons that we assess to be affecting these results is the uniform orthogonal grid of the solver, that doesn't allow a refinement region inside the courtyard, therefore not allowing an accurate enough simulation to capture the flow patterns caused by the membrane in the courtyard and thus treating the membrane mostly as an obstacle boundary. Although the optimization runs were able to converge within the defined timeframe, a much higher grid resolution was avoided, as the computational demand rised exponentially. In both of the 2D cases though, the optimization is considered to be useful as a design tool because it can clearly identify trends within the solution space defined. The interaction between the GA/CFD framework and the designer is considered useful as it is run within a normal design cycle.

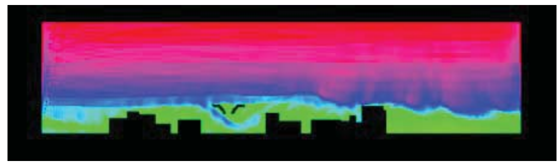

Figure 7 2D Representation with Extended Context – CFD Simulation

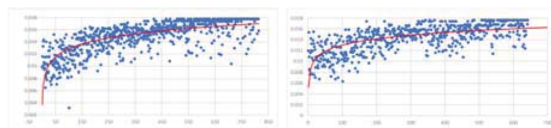

Figure 8 2D Representation with Extended Context – Fitness Plots

3D Representations

The last experiments were done with the 3D representation and both an extended context (only one block around the geometry) and a limited one (no surrounding geometry) but they were not able to converge within the limited time frame. The experiments done with no context would be able to converge to local optima and specific surface forms within a larger time frame, but the inclusion of a 3D context of buildings surrounding the membrane made it impossible for the GA to converge as the computational demand was too high (Figures 9 and 10).

As it can be seen in Figure 9, which shows the simulation run along with the GA population representation, the forms of the 3D membrane do not converge to a single shape. The inclusion of a surrounding context is however inevitable in order to draw meaninful conclusions for a design problem, thus the 3D optimizaton is, at this stage, not meaninful for the designer, which poses a limitation to the method in its current state. As with the 2D larger domain, it is clear that the uniformity of the grid poses a significant drawback on the FFD simulations as it does not allow fine tuning of the domain to increase the resolution near the area of interest. As discussed later, furher development of the solver that would allow refinement areas for the grid would allow much higher resolutions to be achieved near the surfaces.

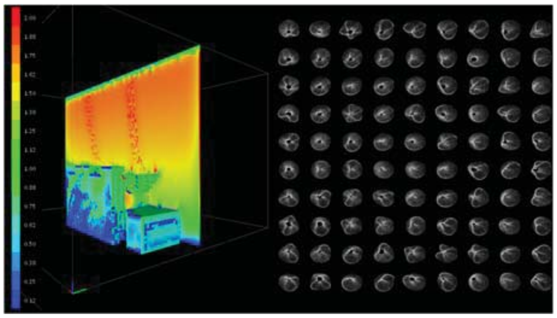

Figure 9 3D Representation with Context – Simulation and Population

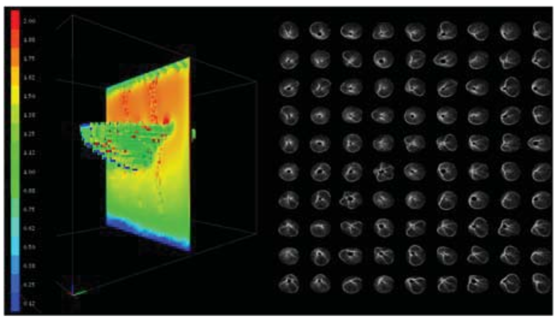

Figure 10 3D Representation with no Context – Simulation and Population

DISCUSSION

Although these experiments can only serve as a first step in assessing the feasibility of employing shape optimization methods for the natural ventilation of courtyards and in an early stage architectural workflow, they nevertheless show that there is potential in integrating shape optimization in the architectural design workflow. They also clearly demonstrate that the current state of integration of CFD and shape optimization in CAAD is limited and further work needs to be done for a methodology to be developed.

Given the results, we assess that 2D optimization problems can be addressed, through the proposed methodology, although caution needs to be taken in the representation of the problem and the definition of an appropriate domain size. From the 2D optimization experiments, we conclude that an inevitable further step for the development of the solver would be the addition of refinement areas. Regarding 3D optimization problems, with the current state of the solver and the GA approach taken, we assess that they can not be sufficiently addressed, especially in relation to natural ventilation problems.

Given the limitations of the method, we assess that at its current state, it can only be useful to address targeted design problems that can be represented in 2D and within a smaller context area. Once refinement areas are introduced, larger 2D areas and 3D representations can be addressed. Nevertheless, further work needs to be done both in 2D and 3D representations, in order to validate the merits of using FFD and GAs for natural ventilation problems. As previously stated, the validation of the results of the optimization process are not in the scope of this paper, as it is focused on the feasibility of employing shape optimization methods and it assumes the accuracy of the solver as defined by previous work mentioned earlier. However, at its next step, the study aims to repeat these experiments with different CFD solvers and integrated optimization methods that are currently becoming available in CAAD environments.

CONCLUSION

This paper investigated the integration of CFD in computational and parametric design and the feasibility of using shape optimization methods to drive the shape of retrofitted membranes for the natural ventilation of courtyard spaces of the typical multistorey buildings in the historic center of Athens, Greece. By running a set of experiments in both 2D and 3D representations of the problem, and using a multi-scalar approach, the study concludes that shape optimization methods are still not fully feasible in an integrated CAAD design workflow and further work needs to be done to employ them in a meaningful way.

2D optimization is feasible up to a degree, but caution needs to be taken in the set up of the domain. Nevertheless, we consider this study as a promising first step towards employing shape optimization within an early stage architectural design framework.

ACKNOWLEDGMENT

This project has received funding from the European Union’s Horizon 2020 research and innovation programme under the Marie Sklodowska-Curie grant agreement No 642877.

REFERENCES

Bai, Guiping, Guangcai Gong, Chuck Wah Yu, and Ouyang Zhen. 2015. “A Combined, Large, Multi-Faceted Bulbous Façade Glazed Curtain with Open Atrium as a Natural Ventilation Solution for an Energy Efficient Sustainable Office Building in Southern China.” Indoor and Built Environment 24 (6): 813–32. doi:10.1177/1420326X15602048.

Chronis, A., A. Turner, and M. Tsigkari. 2011. “Generative Fluid Dynamics: Integration of Fast Fluid Dynamics and Genetic Algorithms for Wind Loading Optimization of Free Form Surfaces.” SimAUD2011, no. Malkawi 2004: 79–85.

Eliopoulou, Eftychia, and Eleni Mantziou. 2017. “Architectural Energy Retrofit (AER): An Alternative Building’s Deep Energy Retrofit Strategy.” Energy and Buildings 150. Elsevier B.V.: 239–52. doi:10.1016/j.enbuild.2017.05.001.

Farea, Tareq Gaber, Dilshan Remaz Ossen, Saqaff Alkaff, and Hisashi Kotani. 2015. “CFD Modeling for Natural Ventilation in a Lightwell Connected to Outdoor through Horizontal Voids.” Energy and Buildings 86. Elsevier B.V.: 502–13. doi:10.1016/j.enbuild.2014.10.030.

Hussain, Shafqat, and Patrick H. Oosthuizen. 2012. “Numerical Study of Buoyancy-Driven Natural Ventilation in a Simple Three-Storey Atrium Building.” International Journal of Sustainable Built Environment 1 (2). Elsevier Ltd: 141–57. doi:10.1016/j.ijsbe.2013.07.001.

Karagkouni, Chrysanthi (Sandy), A.F.G. Schieck, Martha Tsigkari, Angelos Chronis, Ava Fatah gen Schieck, Martha Tsigkari, and Angelos Chronis. 2013. “Façade Apertures Optimization: Integrating Cross-Ventilation Performance Analysis in Fluid Dynamics Simulation.” In SimAUD 2013: Symposium on Simulation for Architecture and Urban Design, 45:16:1--16:9.

Pan, Yiqun, Yuming Li, Zhizhong Huang, and Gang Wu. 2010. “Study on Simulation Methods of Atrium Building Cooling Load in Hot and Humid Regions.” Energy and Buildings 42 (10). Elsevier B.V.: 1654–60. doi:10.1016/j.enbuild.2010.04.008.

Peters, Toby, and David Strahan. 2016. “The Cold Economy – Why ? What ? How ? May 2016,” no. May: 1–15.

Robinson, D., K. J. Lomas, M. J. Cook, and H. Eppel. 2004. “Passive down-Draught Evaporative Cooling: Thermal Modelling of an Office Building.” Indoor and Built Environment 13 (3): 205–21. doi:10.1177/1420326X04043816.

Stam, Jos, and Jos. 1999. “Stable Fluids.” In Proceedings of the 26th Annual Conference on Computer Graphics and Interactive Techniques - SIGGRAPH ’99, 121–28. New York, New York, USA: ACM Press. doi:10.1145/311535.311548.

United Nations Environment Programme (UNEP). 2016. The Emissions Gap Report 2016. United Nations Environment Program. doi:ISBN 978-92-807-3617-5.

Waibel, Christoph, Lukas Bystricky, Aytaç Kubilay, Ralph Evins, and Jan Carmeliet. 2018. “Validation of Grasshopper-Based Fast Fluid Dynamics for Air Flow around Buildings in Early Design Stage.” Accessed February 19. doi:10.26868/25222708.2017.582.

Zuo, Wangda, and Yan Chen Qingyan. 2007. “Validation of Fast Fluid Dynamics for Room Airflow.” Building Simulation 2007, no. 1972: 980–83.