Evaluating the capability of EnergyPlus in simulating geometrically complex Double-Skin Facades through CFD modelling

Salma El Ahmar[1], Antonio Fioravanti[2]
[1,2]Sapienza University of Rome
[1,2]{salma.elahmar|antonio.fioravanti}@uniroma1.it

This paper represents a preliminary investigation into the appropriateness of using EnergyPlus as a simulation tool for Double-Skin Façades (DSFs) that are considered geometrically complex. It builds upon previous research conducted by the authors in which a DSF was designed and simulated for an existing office building in Cairo. For this verification, the DSF was simulated once more using Computational Fluid Dynamics (CFD) to evaluate the accuracy of the previously obtained results. The cavity temperature and the volume flow rate of the airflow provided by EnergyPlus are compared with those obtained by OpenFOAM CFD software. The results give a credible indication of the reliability of EnergyPlus and encourages further investigations. The strengths and limitations of each software are discussed.

Keywords: *Double-Skin Facades, Complex geometry, EnergyPlus, CFD*

INTRODUCTION

With the development of computational design tools, architects are becoming increasingly interested in exploring more complex geometric configurations in their research and practice. Similarly, they are also becoming more interested in the environmental performances of such complex geometries. These interests are seen in façade design which plays an important role in the thermal comfort of interior spaces of buildings. Double skin facades (DSFs) in particular are popular for both aesthetic and environmental reasons, and are the focus of this research.

However, despite the advancement of the design tools that facilitate parametric explorations and evolutionary form finding for example, the simulation software commonly used by architects in the design phase to evaluate and choose among different solutions are not as advanced and not always appropriate to the problem at hand. This is particularly relevant in the case of DSFs in which many physical phenomena occur and interact with each other, such as heat transfer through radiation, conduction and convection (Poirazis, 2006). This affects the final performance of the DSF on which the architect bases his or her decision.

EnergyPlus is among the widely-used building energy simulation tools, and it is easily linked with parametric design software such as Grasshopper for Rhino, due to the presence of many plugins. However, the appropriateness of EnergyPlus is debatable (Zhang, et al., 2013; Sabooni, et al., 2012; Kim and Park, 2011) for modelling flat DSFs, let alone geometrically-complex ones. This represents the main problem addressed in this paper. Computational Fluid Dynamics (CFD) software are more suitable for such problems, however they are often not used in early design phases as they need much more time and expert knowledge for their use.

AIM AND METHOD

This paper continues earlier investigations (El Ahmar & Fioravanti, 2015) of a folded DSF with perforated surfaces that attempted to decrease cooling loads of an existing office building in Egypt. The geometry of the façade was optimized using evolutionary algorithms in Grasshopper, and the fitness function was based mainly on results provided by EnergyPlus through the plugin ArchSim. The results were expected to contain inaccuracies due to limitations of EnergyPlus as will be discussed. This paper represents a preliminary verification phase of previously obtained results. The main objective is to know the degree of inaccuracy of EnergyPlus and whether it can be relied on in simulating the temperature and airflow inside the cavity of a DSF that is considered geometrically complex.

The paper starts with a brief overview of the software used in general to simulate the thermal performance of DSFs, highlighting the main approach used for calculations, their advantages and limitations. Then for this investigation a comparison is performed, in which the temperature and airflow values inside the DSF cavity are simulated using OpenFOAM which is an open-source software for CFD, then the results of these simulations are compared to those previously obtained using EnergyPlus. Another advantage of this comparison is getting a deeper understanding of the behaviour of the proposed folded DSF in terms of its temperature distribution and airflow patterns which were not possible to know earlier.

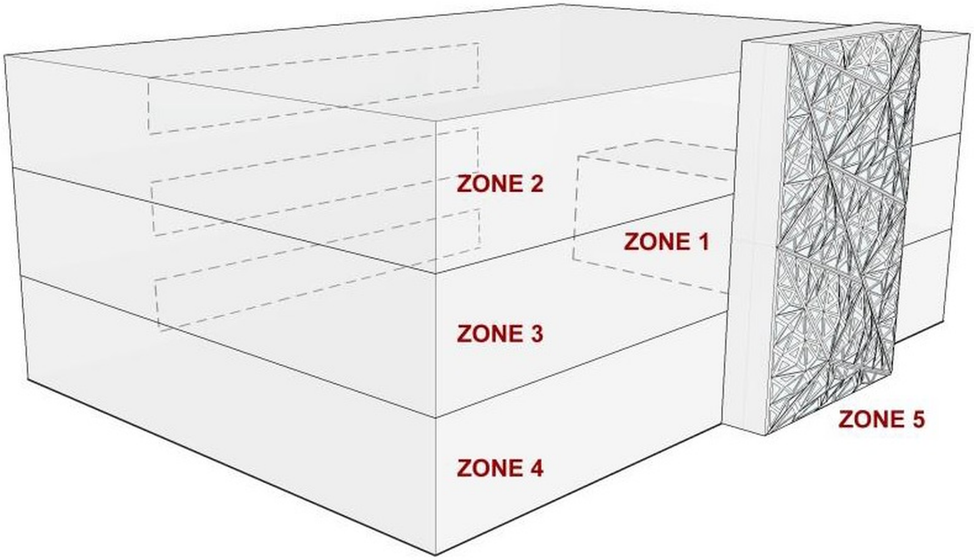


Figure 1
Diagram illustrating the zones of the thermal model of the folded DSF that was simulated using EnergyPlus. The DSF covers just part of the building since only one office room was studied.

MAIN THERMAL & AIRFLOW SIMULATION METHODS USED FOR DSFS

Based on the level of resolution of building simulations, they can be categorised into either macroscopic or microscopic. The macroscopic level deals with whole building systems, interior and exterior conditions over periods of time. The microscopic level focuses on smaller spatial and time scales. Accordingly, thermal and airflow modelling in buildings also can be divided generally to two main approaches; Airflow Network (AFN) models for the macroscopic level and CFD models for the microscopic level (Djunaedy et al., cited in Poizaris, 2006).

AFN models treat every building component and relevant HVAC fluid flow systems as a network of nodes that represents rooms and parts of rooms. The concept of mass conservation for inlet and outlet flows leads to non-linear equations which are integrated over time to characterise the flows. Because of its abilities, it can be used with thermal models, in such cases it is integrated with a thermal network which solves the heat balance at each node. Each thermal zone has just one node at its centroid and is assumed to be 'well-stirred' (De Gracia, et al. 2013; Poirazis, 2006). A main advantage is that, compared to other airflow models, it is the main method used to predict the overall ventilation performance of a building (Chen, 2009). The main limitations of the AFN are that it is not appropriate in cases when the temperature distribution in a zone is significant, and that it only provides information about bulk flows of air (EnergyPlus, 2014).

On the other hand, CFD simulations calculate the desired flow quantities (such as velocity, temperature, etc.) at a large number of points that are connected and distributed throughout the physical domain at hand, forming what is called a mesh or a grid. In CFD models the geometry under investigation is surrounded by a two or three-dimensional grid of nodes and for each node the conservation equation for mass, momentum and thermal energy is solved. Limitations of CFD simulations in practice are mentioned by numerous authors such as De Gracia, et al., (2013), Hensen, et al., van Dijk and Oversloot, Ding, et al., Jaroš, et al. and Chen (all cited in Poizaris, 2006), most importantly; they are too detailed and sophisticated for the early design stage, need high computer power and time, and are not user friendly as they require advanced knowledge to be used.

ENERGYPLUS MODEL

The DSF was assumed to be an addition to an existing office building in Cairo. The folds were intended to reduce the amount of incident solar radiation falling on the façade through self-shading. Additionally, the main structural elements of the facade contain 5x5 cm perforations that represent a ventilation network spreading across the façade area, to reduce overheating of the cavity. The DSF type is multi-story (no vertical or horizontal partitioning in the cavity space) that is 9 m wide, 13.3 m high (starting form first floor) extending 1 m after the ceiling of the last floor to include openings for ventilation. It is difficult to state all the settings of the model in this paper, however the most important material settings are mentioned as follows:

- Inner glazing: double pane, blue tinted, light transmittance= 0.37, solar transmittance= 0.43
- Outer glazing: double pane, clear, Low E, light transmittance= 0.74, solar transmittance= 0.5
- Insulated Aluminum cladding panels: 14 mm in width, with Thermal resistance: 0.0172 m^2K/W, Thermal conductivity: 0.35 W/mK, and Heat transfer coefficient: 5.34 $\frac{W}{m^2}K$

Simulations of the DSF facing the South-East orientation took place on a day that represented a typical summer day in Cairo (2nd of July) in which average ambient temperature is 32°C, average site wind speed is 4.9 m/s with an average direction of 287 degrees (North West). Furthermore, a specific timestep was selected which was at 16:00. This facilitates the comparison with OpenFOAM results as simulations can be calculated for a specific point in time. Figure

1 shows the thermal model used. It is composed of four zones representing a part of the office building and one office room which was studied, in addition to a fifth zone representing the proposed DSF itself.

Two values were selected from the simulation results for comparison; the cavity temperature (Zone Operative Temperature) which was 37.6˚C, and the total Volume Flow Rate at the outlet openings which amounted to 3.74 $\frac{m^3}{s}$. This value corresponded to 77 air changes per hour for the cavity that had a volume of 175 m^3. These airflow results were expected to be overestimated.

OPENFOAM MODEL

An appropriate solver must be chosen depending on the physical phenomena that we wanted to simulate, which included heat transfer by free convection (buoyancy), forced convection (wind), radiation and conduction. After a lot of experimenting with various solvers, and their combinations, the task proved to be very difficult, as there was no solver capable of simulating all these phenomena together. The challenge was even more complicated due to the relatively complex geometry of the façade. This lead to the need of certain simplifications, which are briefly presented in Table 1.

The solver used in this simulation process is called rhoSimpleFoam. It is a steady-state solver used for simulating turbulent RANS (Reynolds Average Navier-Stokes equation) flow of compressible fluids. To simplify and proceed with the simulations, fluid flow due to buoyancy was neglected as it is considered weak when compared with the flow that is induced by wind, and also due to the fact that cavity was partially shaded from direct sunlight which reduced cavity heating. The standard k-ε turbulence model was used in this case.

A Stereo-lithography (stl) file format of the model was exported from Rhino 3d modeller. Different faces or patches were exported separately (Figure 2) to enable the specification of different boundary conditions for each of them.

There were numerous boundary conditions assigned to each patch, most importantly were those obtained from the EnergyPlus simulation results. They are the surface temperatures of the patches that ranged from 48.9˚C for example for the Top patch, to 37.2˚C for the Office Windows patch. The temperature of the inlet airflow itself was 35.6˚C. The Volume Flow Rate at the inlets was used to calculate the inlet flow velocity instead of the wind speed as it was the only output generated from EnergyPlus that gives information about the inlet airflow. Since all inlets have the same surface area, the total volume flow rate of all inlet openings was divided by their total surface area to calculate the velocity of air:

Total volume flow rate at the inlets ($\frac{m^3}{s}$) = total area of inlets x air velocity

3.74 $\frac{m^3}{s}$ = 7.3 m^2 x air velocity

Therefore, the air velocity at inlets = 3.74/7.3= **0.51 m/s**. The rest of the boundary conditions were given values chosen from the OpenFOAM settings.

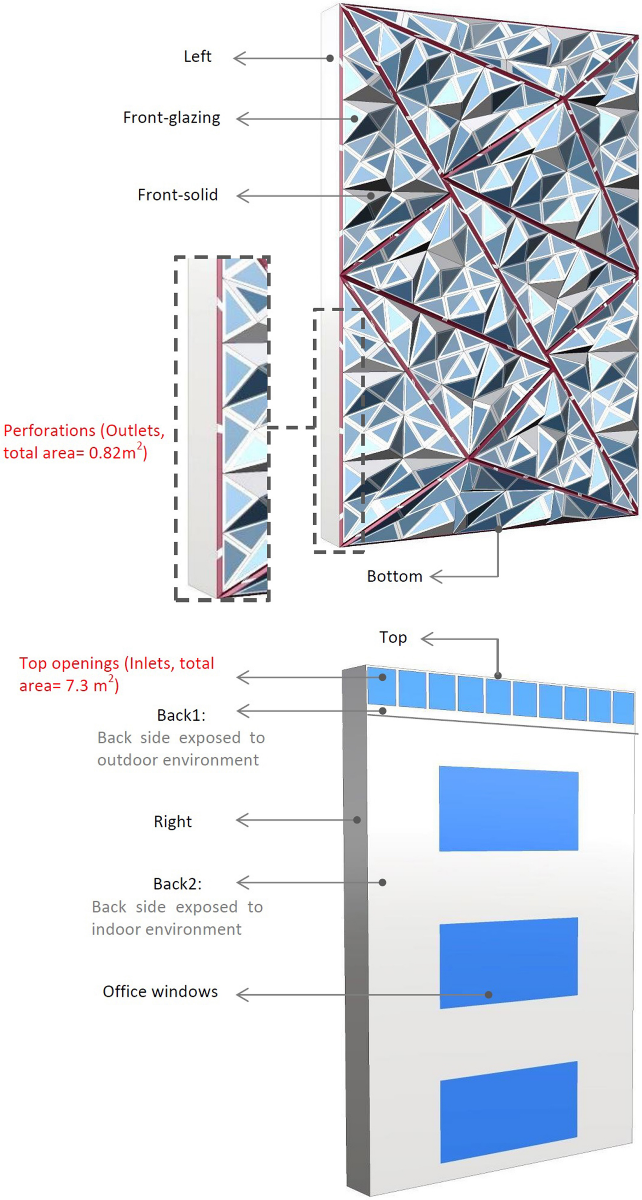
In OpenFOAM the simulated temperature is calculated at every single point of the mesh. Therefore, in order to be able to compare it with the temperature output of EnergyPlus, an average value is calculated for all the points in the mesh, and amounted to 309.6˚K (36.46˚C). The velocity at the perforations ranged from 0 to 5 m/s, with an average of 4.36 m/s. The average velocity should be multiplied by their total surface area to calculate the total volume flow rate.

Total volume flow rate = total area of perforations (outlets) x average velocity at perforations= 0.82 m^2 x 4.36 m/s = 3.62 $\frac{m^3}{s}$.

OBSERVATIONS AND RESULTS

The temperature results in Figure 3 show the dissipation of heat from the hot DSF surfaces to the inside of the cavity either by forced convection or by diffusion. In certain parts of the cavity the effect of convection is stronger when the velocity is relatively high, while in others diffusion has a greater effect when flow ve-

Figure 2
Diagram illustrating the different patches of the OpenFOAM model in the front view (top) and back view (bottom), the inlets and outlets are written in red. The overall dimensions are 13.3 m in height, 9 m in width, and 1.25 m in depth. The cavity volume of the DSF is 175 m³.



Model aspect	Assumption/simplification
Physics simulated	Airflow due to wind and heat transfer by convection and conduction only
Geometry	Small perforations grouped into bigger ones with the same total surface area
Duration to simulate	Specific point in time: 2nd of July at 16:00

Table 1
Summary of assumptions and/or simplifications for the model prepared in OpenFOAM.

locities are low. Forced convection occurs due to the incoming air at the cavity inlets which is entering at a lower temperature than the surfaces and is equal to the ambient temperature of 308˚K (35.6˚C) which was assigned as a boundary condition. Other results showed that the incoming air pushes warmer air to the front folded faces of the DSF where it is trapped in some parts unless there is a perforation to act as an outlet and allow the warmer air to escape.

By comparing the visualised results of the temperature and velocity, it was observed that, in general, the cavity temperature is often lower in areas with higher airflow velocity and therefore have more heat loss by convection. It was also observed that the temperature in the middle of the cavity is higher than the bottom of the cavity despite having relatively higher airflow velocity. This is due to the thinner cross-section at the area so the heat transferred by diffusion has a strong effect and the flow velocity is not strong enough to ventilate it.

The comparison between the results of EnergyPlus and OpenFOAM showed an unexpected similarity between the results. It was expected that there would be inaccuracies in EnergyPlus, especially in the results regarding the airflow. However, EnergyPlus only slightly overestimated both temperature and airflow values by only 3.1 % and 3.3 % respectively. However, it cannot give us a detailed insight of the behaviour of the airflow and temperature distribution inside the cavity which is important for the evaluation of the proposed design solution. So even if EnergyPlus can predict the average velocity of the air, it is not capable of demonstrating, for example, that this air is not flowing evenly throughout the cavity and that only the upper half is considered well ventilated. Furthermore, since in this paper only one model was simulated using OpenFOAM, the results can give only a credible indication of the reliability of EnergyPlus. This reliability would be reinforced by repeating the comparison with different models having varying performances.

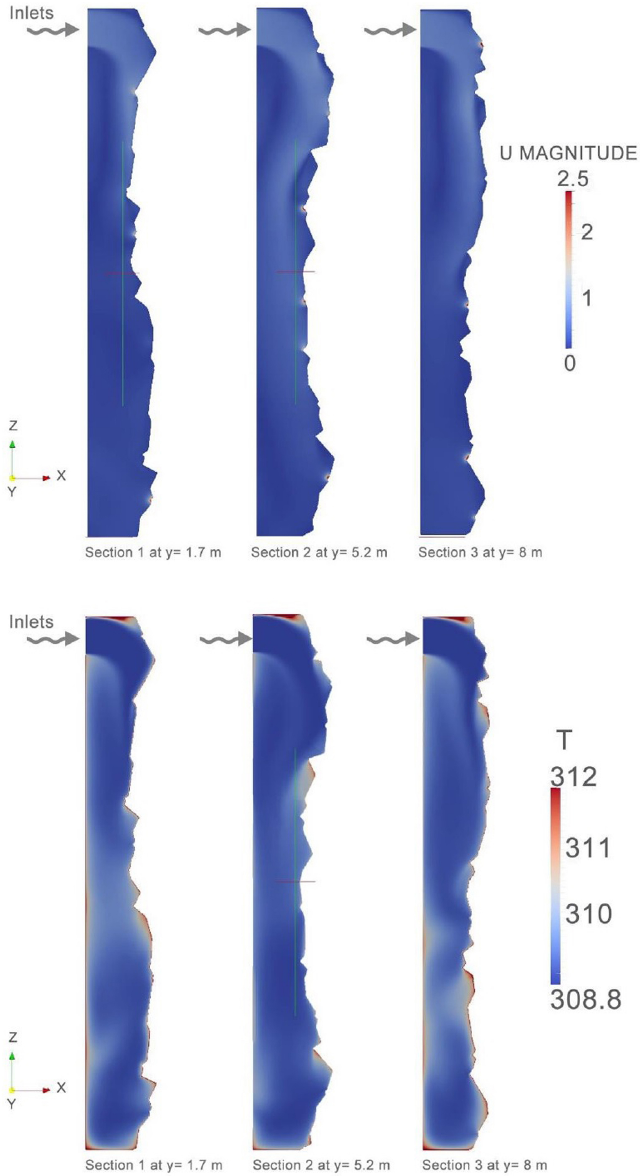
It must be mentioned again that simplifications were made to the OpenFOAM model, hence inaccuracies might be present in the results. Nonetheless, it is assumed that these results are more reliable than the ones obtained by EnergyPlus. What can be certain from the OpenFOAM simulations are the qualitative results, such as those regarding the flow pattern and behaviour, and temperature distribution. The quantitative results; the exact numerical values of the temperature and velocity, are subject to criticism as with any other software.

CONCLUSION

This investigation demonstrated that EnergyPlus can give a general idea of the DSF performance, even if it is considered geometrically complex, and is most suitable in early design phases. However, the airflow pattern and temperature distribution results by OpenFOAM are important in understanding the behaviour and improving the design accordingly. CFD simulations need expert knowledge and much more computing time, and are important in later, more detailed design phases when final decisions must be taken.

The simulations of DSFs remains a challenge even with CFD software, and any simulation would be subject to a certain degree of error. A real veri-

Figure 3
Vertical sections 1, 2 and 3 from left to right illustrating the magnitude (top) of the airflow velocity (in m/s) and the temperature (bottom) throughout the DSF cavity at the last simulated time step at which a converged state of the flow is reached in the CFD simulations.



fication, either for EnergyPlus or OpenFOAM results, would require physical experimentation which falls within the scope of future work.

ACKNOWLEDGEMENT

I would like to thank Dr. Francesco Battista from the Department of Mechanics at Sapienza University, who was my tutor and guided me through the development of the CFD model. His assistance was critical for the achievement of this work.

REFERENCES

El Ahmar, S. and Fioravanti, A. 2015 'Simulating the thermal and daylight performances of a folded porous double façade for an office building in Cairo', *Proceedings of the 49th International Conference of the Architectural Science Association (ASA)*, Melbourne

Chen, Q. 2009, 'Ventilation performance prediction for buildings: A method overview and recent applications', *Building and Environment*, 44(4), pp. 848-858

EnergyPlus, - 2014, *Input Output Reference: The Encyclopedic Reference to EnergyPlus Input and Output*, US Department of Energy, Illinois

De Gracia, A., Castell, A., Navarro, L., Oro, E. and Cabeza, L.F. 2013, 'Numerical modelling of ventilated facades: A review', *Renewable and Sustainable Energy Reviews*, 22, p. 539–549

Kim, D. and Park, C. S. 2011, 'Difficulties and limitations in performance simulation of a double skin facade', *Energy and Buildings*, 43, p. 3635–3645

Poizaris, H. 2006, *Double Skin Facades: A literature review*, International Energy Agency (IEA), Solar Heating and Cooling (SHC) Task 34, Lund

Sabooni, M. A., Vaseti, H. M., Maerefat, M. and Azimi, A. 2012 'Development of the capability of Energyplus software to simulation of building double-skin facade', *Proceedings of the International Symposium on Sustainable Energy in Buildings and Urban Areas (ICHMT)*, Kusadasi

You, W., Qin, M. and Ding, W. 2013, 'Improving building facade design using integrated simulation of daylighting, thermal performance and natural ventilation.', *Building Simulation*, 6, p. 269–282

Zhang, R., Lam, K., Yao, S. C. and Zhang, Y. 2013, 'Coupled Energyplus and computational fluid dynamics natural ventilation simulation.', *Building and Environment*, 68, pp. 100-113