International Information and Engineering Technology Association
Advancing the World of Information and Engineering

TECNICA ITALIANA-Italian Journal of Engineering Science
Vol. 65, No. 2-4, July, 2021, pp. 330-336
Journal homepage: http://iieta.org/journals/ti-ijes

CFD Modelling of Naturally Ventilated Double Skin Façades: Comparisons among 2D and 3D Models

Camilla Lops[1*], Nicola Germano[1], Sabino Matera[1], Valerio D'Alessandro[2], Sergio Montelpare[1]

[1] University "G. d'Annunzio" of Chieti-Pescara, Engineering and Geology Department (INGEO), Viale Pindaro 42, 65127 Pescara, Italy
[2] Marche Polytechnic University, Industrial Engineering and Mathematical Sciences Department (DIISM), Via Brecce Bianche 1, 60131 Ancona, Italy

Corresponding Author Email: camilla.lops@unich.it

https://doi.org/10.18280/ti-ijes.652-429

Received: 22 March 2021
Accepted: 5 May 2021

Keywords:
CFD, double skin façade, natural convection, passive cooling, turbulence model

ABSTRACT

Nowadays, Double Skin Façades (DSFs) are popular technologies adopted for both new and existing buildings. Since their introduction, new configurations and materials started to be tested to improve the DSF energy behaviour and function. Such complex technologies, able to improve comfort conditions of occupied spaces and decrease building energy requirement, are strictly related to the design phase that should be carefully evaluated.
The correct prediction of air fluxes inside the DSF cavity, in fact, is highly influenced by the adopted analysis hypothesis and settings. Moreover, the absence of multiple experimental campaigns and empirical validations in the sector represents the major concerns for scientists and researchers. Among the possible numerical approaches for studying DSFs, Computational Fluid Dynamics (CFD) analyses confirm to be the most suitable solution.
The CFD modelling activity presented in this paper intends to compare various Double Façade configurations by adopting bi- and three-dimensional domains and different turbulence models. According to the obtained results, 2D simulations can predict airflows inside and around the DSF channel with good approximation and reasonable computational effort. Moreover, the velocity profiles estimated by the turbulence formulations are in good accordance, underling only a few slight variations in proximity to the DSF layers.

1. INTRODUCTION

Since the early 1900s, Double Skin Façades have been used in buildings to improve their environmental performance. Only in the late 1970s, investigations started to be carried on to measure and predict these systems' effectiveness [1]. In general, the DSF is a pair of glass skins separated by an air corridor, also called cavity or intermediate space, ranging in width from 0.20 m to several meters [2]. The air space between the two layers works as insulation against temperature extremes and sound.

Besides the popularity of DSFs, adopted for both new and retrofitted buildings, researchers' primary concerns are related to the experienced difficulties in modelling their thermal and energy behaviour. The above-described problems are mainly due to the lack of multiple experimental campaigns and empirical validations of such interesting systems.

The global performance of Double Skin Façades is, in fact, very complex because of the involved multiple coupled physical phenomena as the air movement, heat convection, conduction, and short- and long- wave radiation [3]. Various approaches can be selected for evaluating the air fluxes and temperature distributions inside the DSF cavity. Some of them use mathematical models [4, 5], others field measurements [6-8] or finite element methods for performing fluid dynamics simulations [9, 10].

According to Papakonstantinou et al. [11], computer analyses are able to describe the natural ventilation of occupied spaces, offering predictions that are in good agreement with experimental values of air velocity, temperature and pressure. Moreover, Liddament [12] underlines how the estimations obtained by numerical methods have enabled the concepts of ventilation efficiency to be applied at the design stage, while the value of the experimental method has been restricted to the evaluation of existing structures.

Among the possible options, Computational Fluid Dynamics simulations allow reasonable estimations in the case of DSF systems. The literature review about the CFD modelling of Multi-layered façades, in fact, suggests its capability in predicting results, which are not only physically plausible but also in good agreement with available experimental campaigns. As confirmed by Dama et al. [13], the CFD approach can give a qualitative picture of the realistic flows that characterise the DSF cavity. Spiking about the quantitative scale, CFD estimations could be deeply affected by selecting wrong turbulence models or boundary conditions, which are the essential core of the simulation. For this reason, the validation and verification of the model is a fundamental step, as suggested by Chen and Srebric [14].

Xu and Ojima [6] confirm the reliability of CFD simulations applied to Double Façades and estimate a minimal and maximal error in the comparisons between measured and predicted values, respectively, 2.5% and 12%. Instead, other investigations are centred on the evaluation and definition of the best settings to be used for performing CFD analyses, as the study led by Pasut and De Carli [15]. The research focuses on defining a scientifically validated strategy for carried out CFD simulations in naturally ventilated DSF buildings. Moreover, the work intends to identify those factors which are essential in the simulation and the others that increase the model complexity without improving the prediction capacity.

Despite the CFD's here-mentioned benefits into the design process, the user must be aware of the common downfalls and limitations specific to the analysis model. In fact, for performing correct CFD simulations, the full comprehension of the fundamental aspects that govern the fluid dynamic problem, as the conservation equations or the adopted turbulence model is crucial.

To verify the physical hypotheses adopted for the DSF modelling and compare the estimated ventilation values referred to various configurations, the present paper provides a discussion about multiple CFD simulations, both bi- and three-dimensional. Moreover, different turbulence models are adopted and tested. The paper is structured as follows: after this introduction, the selected case study is presented in Section 2. Then, 2D and 3D CFD analyses are described and shown in Section 3. Finally, Section 4 draws the main conclusions.

2. THE CASE STUDY

The analyses here-presented are applied to a simple case study, modelled on the characteristics and dimensions of the Double Skin Façade outdoor full-scale test facility realised by the Department of Civil Engineering of the Aalborg University in collaboration with the Department of Sciences and Methods for Engineering of the University of Modena and Reggio Emilia [16-18]. Figure 1 shows the test cell configuration with the schematisation of the internal zones (on the upper part) and the pictures of the DSF south façade (on the left) and the north elevation (on the right).

The mock-up dimensions are 6 m x 6 m x 6 m, and the DSF cavity depth is equal to 0.70 m. The Double Skin Façade is facing south and consists of an internal double-glazed layer and a single-glazed exterior layer. The investigated cavity is naturally ventilated. Thus, the air fluxes inside the cavity are ensured by the only natural convection.

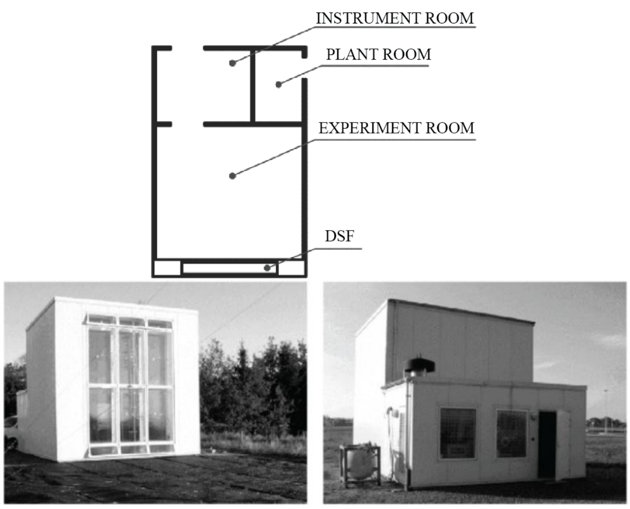

Figure 1. The plan and photos of the southern and northern façade [13]

Various CFD models are generated for comparing the obtained estimations, expressed in terms of cavity ventilation. Initially, a bi-dimensional model is elaborated, and comparisons are made among the results with the measured and CFD values obtained from the literature review. Then, investigations are carried out on a different DSF opening configuration that presents inlet and outlet sections not partial but fully open, thus avoiding pressure drops with higher velocities inside the channel. Finally, a three-dimensional model is evaluated for quantifying the impact of lateral openings on the cavity air fluxes. Figure 2 schematises the investigated options.

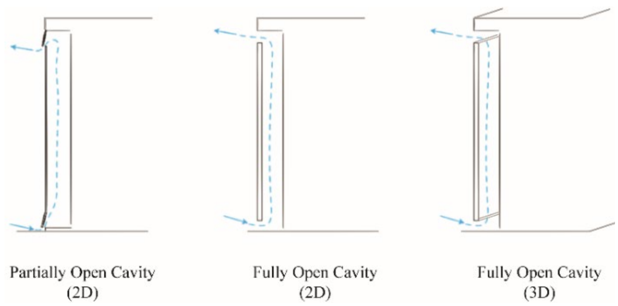

Figure 2. Schematisation of the investigated DSF configurations and models

Detailed information about the conservation equations of mass, momentum, energy and turbulence quantities can be found in Versteeg and Malalasekara [19]. Unsteady Reynolds-Averaged Navier Stokes (U-RANS) simulations are performed. Turbulence is considered by adopting various models. First, the SST k-w model is used to run preliminary analyses and validate the numerical model, considering its capability to describe natural convection in the cavity. Then, further simulations are carried on by adopting the V2F k-e and the Realisable k-e model for comparing the outputs referred to each formulation. The additional two turbulence models are selected for being extensively validated for a wide range of flows and also in the case of air fluxes inside channels and layers [15, 20, 21]. Moreover, the all y+ hybrid approach is used to determine the relationship between the first cell centre and the wall, solving the problem of mesh resolution insufficiency near the wall.

Polyhedral and prism layer meshes are selected for discretising the model surfaces. Variable mesh sizes are inserted to improve the accuracy of predictions of specific areas (e.g., the DSF cavity) without increasing the model computational cost. The following settings are selected for the surface meshing:

- Base Size = 0.4 m
- Number of Prism Layer = 5
- Prism Layer Stretching = 1.2
- Prism Layer Thickness = 0.1 m

Uniform temperature conditions are imposed at all glazed surfaces of the façades and its ground and ceiling. The used temperature values are extrapolated by the measurement campaign carried out for the reference test cell for which the buoyancy is supported by a moderate upward wind differential pressure [13]. The recorded values are, respectively, 14.6°C for the air temperature, 29.5°C for the inner layer of the DSF and 28.6°C for the inner building surface.

Finally, some considerations are referred to the natural ventilation of the DSF cavity placed on the south elevation.

The cavity is modelled by making two main assumptions: the pressure value on domain borders is fixed at zero (pressure outlet), and the reference density inside the physics continuum is defined according to the gas for the temperature and pressure level of the domain.

The CFD simulations are carried out through the commercial software Star-CCM+ (version 13) [22], and the investigated output is the velocity profile (expressed in m/s), evaluated at different DSF heights. In detail, the investigated heights are 0.95 m, 2.50 m, and 5.15 m, selected for being placed, respectively, at the inlet, middle and outlet area. The maximum physical time is set at 20 min for all simulations.

3. CFD MODELLING OF THE CASE STUDY

3.1 Modelling of the 2D DSF for preliminary comparisons

The first model to be elaborated presents the exact configuration of the mock-up mentioned above for preliminary comparisons and validations. Figure 3 shows the surfaces used for describing both the domain and the building. The total number of mesh elements amount to 7,493, and the simulation time step is set at 0.05 s.

Figure 4 depicts the results referred to each probe line. The velocity magnitudes obtained by the performed CFD analyses, represented by the blue solid lines, are in good accordance with the estimated outputs (orange solid lines) adopted as reference for the comparisons. Moreover, both results underline a significant variation if compared to the measured values (yellow diamond markers). This phenomenon is related to the sensitivity of the velocity measurements inside the perturbated area, which could lead to a higher uncertainty of the recorded data [13]. According to the outputs, the model can be considered able to predict the correct fluid dynamic behaviour of the DSF.

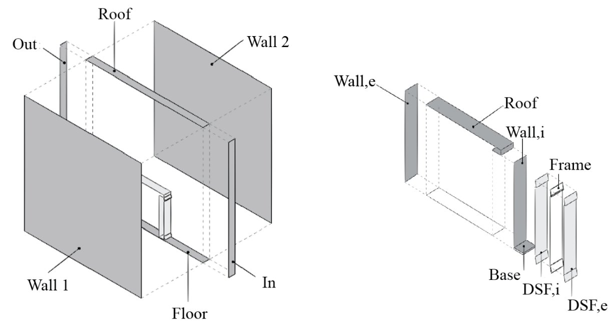

Figure 3. Representation of domain components (left) and DSF case study (right)

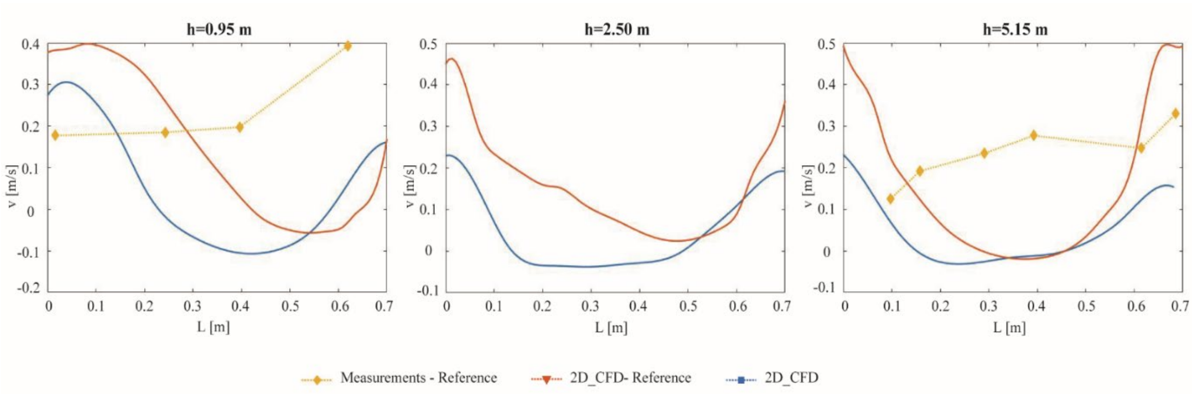

Figure 4. Velocity magnitude results for the investigated probe lines

3.2 Modelling of the fully open DSF Cavity – 2D Model

The second stage of the study is testing a different Double Façade in which the inlet and outlet areas are not partial but fully open. The simulation settings and hypotheses adopted for the previous model are also confirmed in this case. Figure 5 shows the subdivision of the domain and building into surfaces.

The total number of mesh elements amount to 71,376, and the simulation time step is set at 0.1 s.

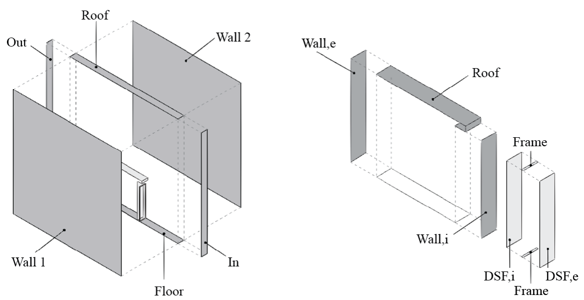

Figure 5. Representation of domain components (left) and DSF case study (right)

Comparisons in terms of velocity magnitude among the two bi-dimensional models are plotted in Figure 7. The obtained results underline that more aerodynamic profiles, as it happens in the DSF improved version, ensure higher cavity velocities. This phenomenon is due to the absence of elements that cause velocity losses.

3.3 Modelling of the fully open DSF Cavity – 3D Model

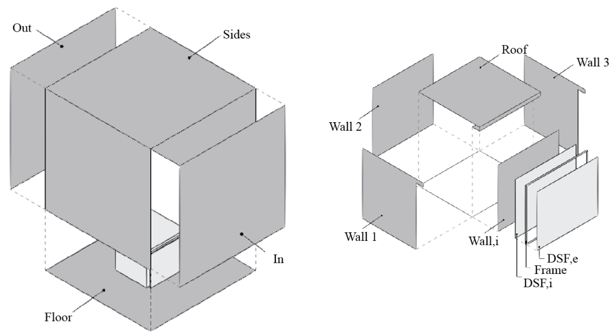

Figure 6. Representation of domain components (left) and DSF case study (right)

After testing the effectiveness of the improved version of the DSF case study, a 3D model is elaborated to quantify the impact of lateral openings on the cavity air fluxes. The previously adopted assumptions are also confirmed in this case, and the scale factor between building and domain is set to 10%. The total model cells are 657,736.

Figure 6 depicts the components of the domain and building and the generated surface and volume meshes. The time step is fixed at 0.01 s for avoiding model convergence problems.

Figure 8 shows the obtained results and comparisons with the 2D configuration. According to the outputs, there are no significant variations between the estimations coming from the 2D (blue solid lines) and 3D (orange solid lines) models except for the lower probe line, placed at 0.95 m. In fact, the bi-dimensional model is able to describe the air fluxes inside the DSF cavity with good accuracy and less computational cost than the three-dimensional simulation, which, instead, is much more complex and with higher solving times.

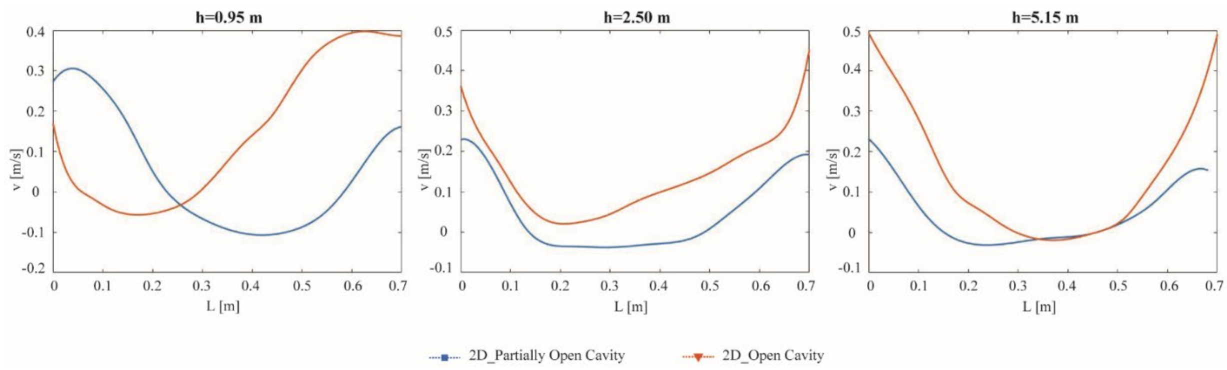

Figure 7. Velocity magnitudes for the selected probe lines referred to the 2D partially and fully open cavity

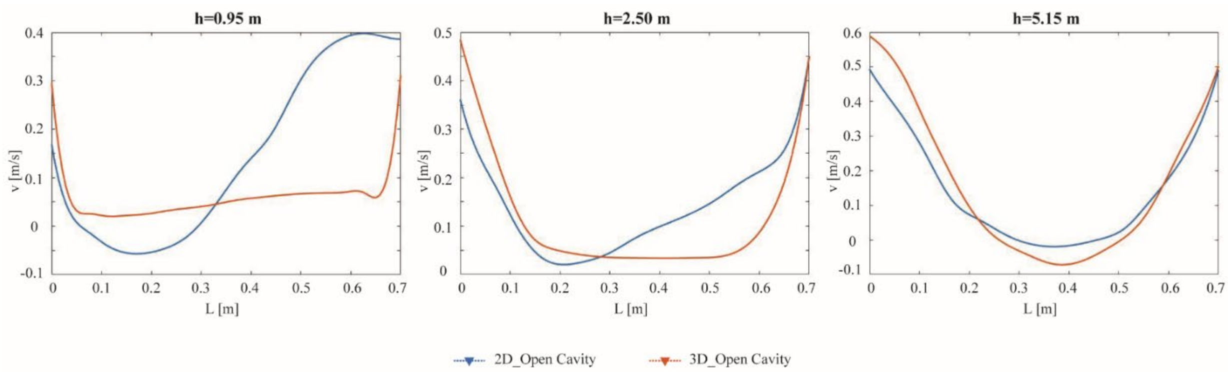

Figure 8. Velocity magnitudes for the selected probe lines referred to the 2D and 3D fully open cavity

3.4 Comparisons among Different Turbulence Models

Once the performance associated with different DSF configurations and simulation domains has been analysed, various turbulence models are implemented. The results obtained by adopting the SST k-w formulation are compared with those referred to the V2F and Realisable k-e turbulence models. Comparisons are made in terms of velocity profiles.

Figure 9 to Figure 11 show the convective plume generated by the Double Skin Façade that moves the air till reaching the top of the domain. All the formulations estimate a similar velocity trend inside and around the DSF. Moreover, better accordance exists among the two k-e model.

The velocity profiles, subdivided for probe line and DSF configuration, estimated, instead, inside the DSF cavity are depicted in Figure 12 to Figure 14. The investigated formulations underline good accordance in evaluating the outputs for the selected configurations.

In the case of the partially open cavity (Figure 12), the SST k-w model tends to overestimate velocities in proximity to the channel's borders. In contrast, it underestimates the values inside the internal area. The V2F and Realisable k-e model, instead, show very similar predictions, both qualitatively and quantitatively.

Looking at the DSF fully open cavity model (Figure 13), more appreciable differences can be seen among the investigated formulations. The Realisable k-e model (blue solid lines in the charts) records the most intense ventilation above the inner layer of the cavity, especially in correspondence to the probe line placed at 2.5 m. The SST k-w (yellow solid lines in the graphs) shows the lowest values in comparisons to the others, underestimating the velocity profile inside the intermediate areas of the channel. The last model, the V2F k-e, estimates intermediate values if compared to the other two formulations as it is emphasised at the inlet and outlet section of the DSF.

Similar trends are also estimated for the 3D model (Figure 14). At the inlet area (see probe line 0.95 m), all turbulence formulations describe a very similar trend in predicting the expected velocity profile inside the cavity. More significant variations are, instead, shown at 2.50 and 5.15 m. In both cases, the most evident deviation is expected inside the intermediate area of the channel, especially for the probe line placed at the bottom of the DSF. Moreover, in the specific case, the SST k-w model exhibits greater values in correspondence to the inner and outer layer of the cavity than the other two formulations, which, instead, underline good agreement among each other.

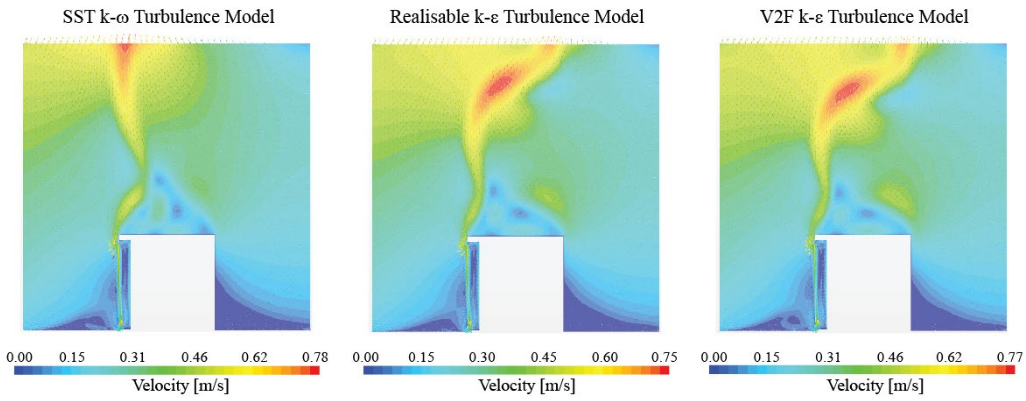

Figure 9. Velocity profile for the 2D partially open cavity

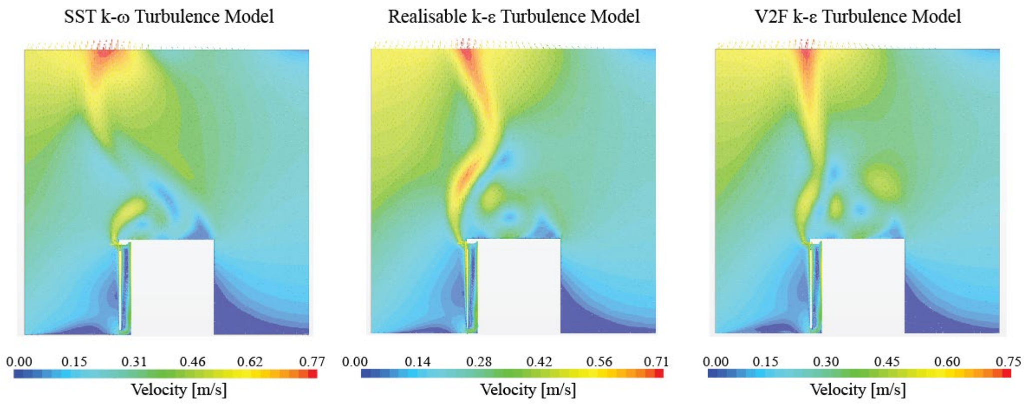

Figure 10. Velocity profile for the 2D fully open cavity

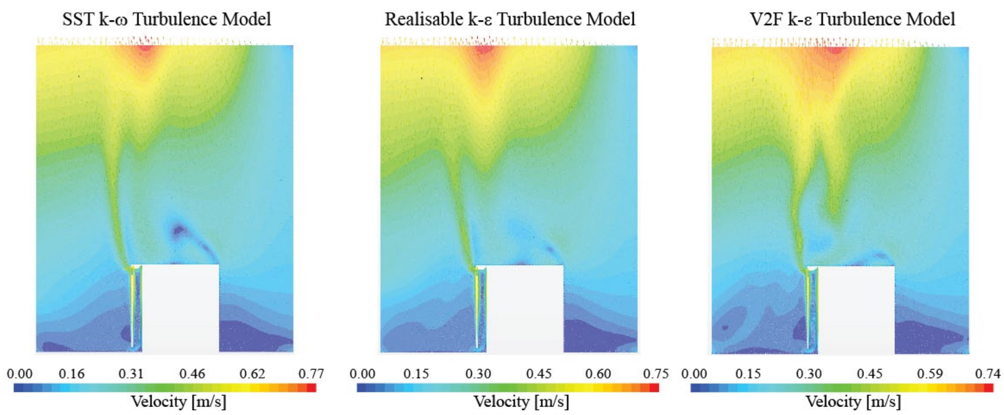

Figure 11. Velocity profile for the 3D fully open cavity

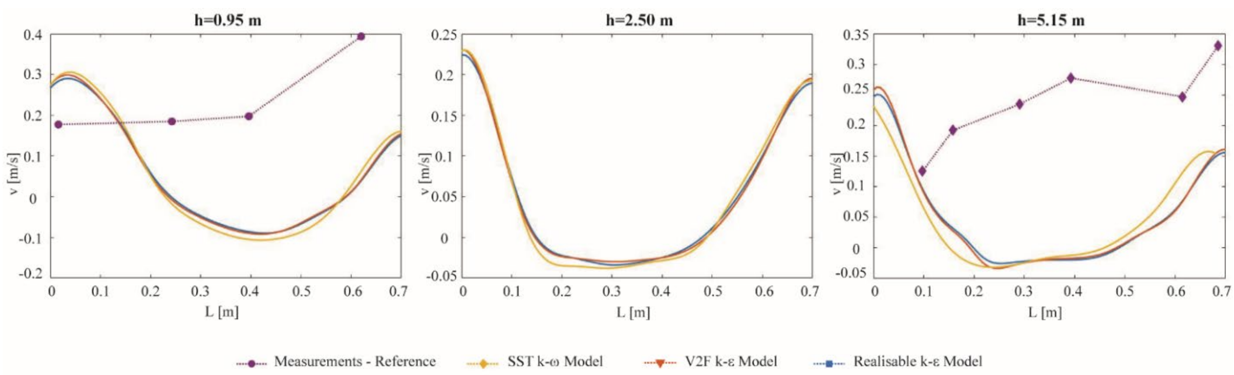

Figure 12. Velocity profiles referred to the 2D partially open cavity model for the selected probe lines

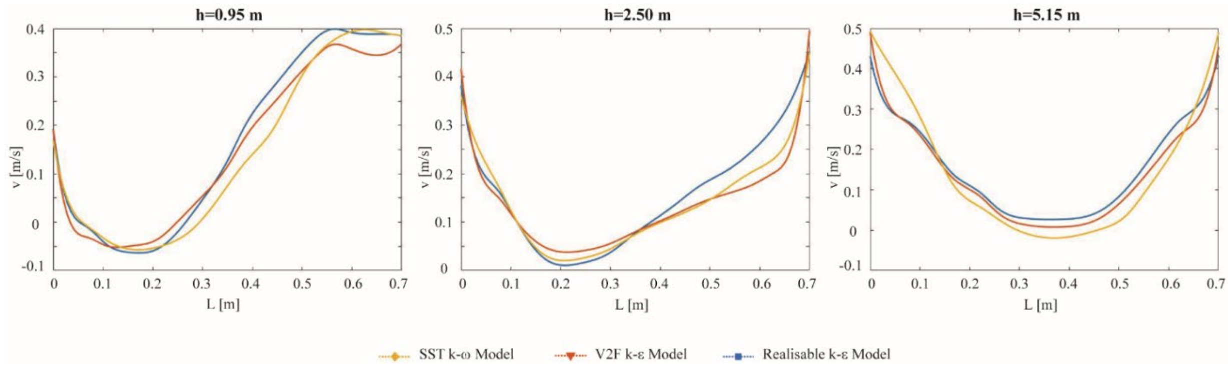

Figure 13. Velocity profiles referred to the 2D fully open cavity model for the selected probe lines

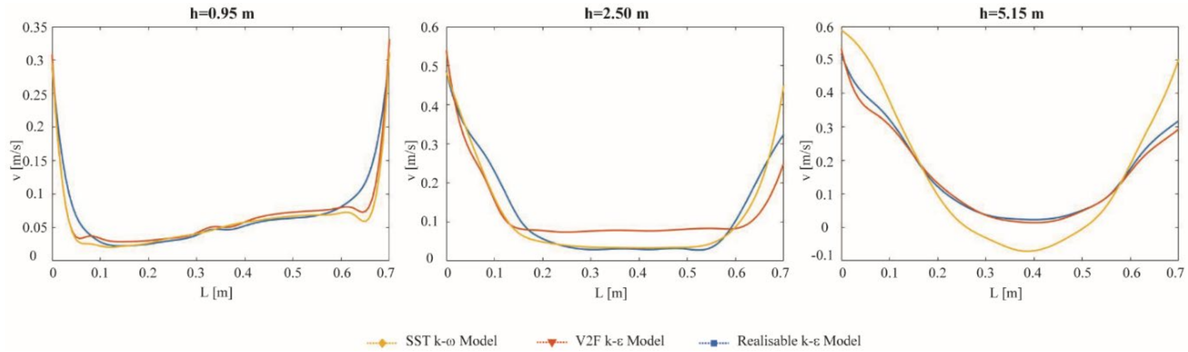

Figure 14. Velocity profiles referred to the 3D fully open cavity model for the selected probe lines

4. CONCLUSIONS

The research work aimed to estimate the velocity profile inside a naturally ventilated Double Skin Façade cavity by adopting various configurations, turbulence formulations, and both bi- and three-dimensional models. The first stage of the analysis concerned intercomparisons among the Computational Fluid Dynamics simulations here performed and the outputs underlined by the literature review and obtained from the experimental campaign and numerical modelling. Then, an improved DSF version was tested through 2D and 3D modelling, and various turbulence formulations are evaluated.

The main conclusions of the investigation are summarised as follows:

- Adopting a fully open inlet and outlet section ensures better performances and more intense natural ventilation inside the DSF cavity. The fully open channel, in fact, is characterised by higher velocities, expected for the whole elevation of the Double Façade, than those estimated for the partially open cavity. This phenomenon is due to the presence of not aerodynamically profiled sections.
- Bi-dimensional analyses are able to sufficiently describe the velocity profile inside and around the DSF cavity. 2D simulations predict values that are in accordance with the trends and entities underlined by the 3D analyses but with a much lower computational effort.
- The SST k-w and the two k-e (the V2F and Realisable) turbulence models ensure good accuracy in the estimations. They all predict a similar trend for the velocity profiles, expected inside and outside the DSF cavity. Reasonable variations are also underlined in proximity to the inner layer of the DSF cavity for almost all the investigated probe lines. In contrast, more significant differences are emphasised in the middle of the cavity around the outlet zone.

REFERENCES

[1] Ji, Y., Cook, M.J., Hanby, V., Infield, D.G., Loveday, D.L., Mei, L. (2008). CFD modelling of naturally ventilated double-skin facades with Venetian blinds. Journal of Building Performance Simulation, 1(3): 185-196. https://doi.org/10.1080/19401490802478303

[2] Uuttu, S. (2001). Study of current structures in double-skin facades (Master's thesis). M.S. thesis, Helsinki University of Technology, Helsinki, Finland.

[3] Safer, N., Woloszyn, M., Roux, J.J. (2005). Three-dimensional simulation with a CFD tool of the airflow phenomena in single floor double-skin facade equipped with a venetian blind. Solar Energy, 79(2): 193-203. https://doi.org/10.1016/j.solener.2004.09.016

[4] Paassen, D., van der Voorden, M. (2000). Development of simplified tools for evaluation energy performance of double façades. In International Building Physics Conference. Tools for Design and Engineering of Buildings. Proceedings, pp. 347-355.

[5] Hanby, V.I., Cook, M.J., Infield, D.G., Ji, Y., Loveday, D.L., Mei, L., Holmes, M.J. (2008). Nodal network and CFD simulation of airflow and heat transfer in double skin facades with blinds. Building Services Engineering Research and Technology, 29(1): 45-59. https://doi.org/10.1177/01436244080290010401

[6] Xu, L., Ojima, T. (2007). Field experiments on natural energy utilization in a residential house with a double skin façade system. Building and Environment, 42(5): 2014-2023.

https://doi.org/10.1016/j.buildenv.2005.07.026
[7] Zöllner, A., Winter, E.R.F., Viskanta, R. (2002). Experimental studies of combined heat transfer in turbulent mixed convection fluid flows in double-skin-facades. International Journal of Heat and Mass Transfer, 45(22): 4401-4408. https://doi.org/10.1016/S0017-9310(02)00160-6
[8] Ding, W., Hasemi, Y., Yamada, T. (2005). Natural ventilation performance of a double-skin façade with a solar chimney. Energy and Buildings, 37(4): 411-418. https://doi.org/10.1016/j.enbuild.2004.08.002
[9] Naylor, D., Collins, M.R. (2004). Evaluation of an approximate method for predicting the U-value of a window with a between-panes louvered shade. In CHT-04-Advances in Computational Heat Transfer III. Proceedings of the Third International Symposium. Begel House Inc., pp. 1-15. https://doi.org/10.1615/ICHMT.2004.CHT-04.710
[10] Marjanovic, L., Cook, M.J., Hanby, V.I., Rees, S. (2005). CFD modelling of convective heat transfer from a window with adjacent venetian blinds. In the Ninth International IBPSA Conference, pp. 709-716.
[11] Papakonstantinou, K.A., Kiranoudis, C.T., Markatos, N.C. (2000). Numerical simulation of air flow field in single-sided ventilated buildings. Energy and Buildings, 33(1): 41-48. https://doi.org/10.1016/S0378-7788(00)00063-3
[12] Liddament, M.W. (1992). The role and application of ventilation effectiveness in design. In Proceeding of International Symposium on Room Air Convection and Ventilation Effectiveness, University of Tokyo, pp. 59-75.
[13] Dama, A., Angeli, D., Larsen, O.K. (2017). Naturally ventilated double-skin façade in modeling and experiments. Energy and Buildings, 144: 17-29. https://doi.org/10.1016/j.enbuild.2017.03.038
[14] Chen, Q., Srebric, J. (2002). A procedure for verification, validation, and reporting of indoor environment CFD analyses. Hvac&R Research, 8(2): 201-216.
[15] Pasut, W., De Carli, M. (2012). Evaluation of various CFD modelling strategies in predicting airflow and temperature in a naturally ventilated double skin façade. Applied Thermal Engineering, 37: 267-274. https://doi.org/10.1016/j.applthermaleng.2011.11.028
[16] Kalyanova, O. (2008). Double-skin facade: modelling and experimental investigations of thermal performance. Thesis, Aalborg University, Aalborg, Denmark.
[17] Larsen, O.K., Heiselberg, P., Jensen, R.L. (2014). Experimental data and boundary conditions for a Double-Skin Facade building in external air curtain mode. 1-24.
[18] Kalyanova, O., Heiselberg, P., Felsmann, C., Poirazis, H., Strachan, P., Wijsman, A. (2009). An empirical validation of building simulation software for modelling of double-skin facade (DSF). In Proceedings of the 11th IBPSA Conference (International Building Performance Simulation Association), pp. 27-30.
[19] Versteeg, H.K., Malalasekera, W. (2007). An introduction to computational fluid dynamics: The finite volume method. Pearson education. Harlow, Essex, England, 1995, 1-520.
[20] Coussirat, M., Guardo, A., Jou, E., Egusquiza, E., Cuerva, E., Alavedra, P. (2008). Performance and influence of numerical sub-models on the CFD simulation of free and forced convection in double-glazed ventilated façades. Energy and Buildings, 40(10): 1781-1789. https://doi.org/10.1016/j.enbuild.2008.03.009
[21] Janssens, B., Bosschaerts, W. (2016). Study of the airflow window. Energy Procedia, 85: 303-310. https://doi.org/10.1016/j.egypro.2015.12.256
[22] *STAR CCM+ User Guide*, 4.02 Version, CD-ADAPCO,. 2008, 1-406.

NOMENCLATURE

DSF	Double Skin Façade
CFD	Computational Fluid Dynamics
2D	Bi-dimensional
3D	Three-dimensional
m	Meters
U-RANS	Unsteady Reynolds-Averaged Navier Stokes
SST	Menter's Shear Stress Transport
k	Turbulent kinetic energy, $m^2 s^{-2}$
V2F	Velocity scale for the eddy viscosity
°C	Degree Celsius
min	Minutes
v	Velocity, $m s^{-1}$
L	Length of the DSF cavity, m
s	Seconds
h	Height of the probe lines
in	Inlet
out	Outlet

Greek symbols

w, e	Dissipation rate of the turbulent kinetic energy, $m^2 s^{-3}$

Subscripts

i	Internal
e	External