# Evaluation of Impact of Turbulence Model Choices in CFD for Cp-Values Used in Airflow Networks

Junji Yi[1], Andrea Frisque[2], Kendal Bushe[1]
[1]Department of Mechanical Engineering, University of British Columbia, Vancouver, BC, Canada
[2]Stantec Consulting, Vancouver, BC, Canada

## Abstract

Pressure coefficient values are incorporated into airflow network (AFN) models and are used to calculate the bulk airflow into the building. Given that Cp values are so important and at the same time are so difficult to get, accurate CFD simulations can be and are used as an easy and inexpensive way to obtain Cp values compared to wind tunnel testing. However, the accuracy of these values is not well understood. The literature on the magnitude of error to expect in Cp values is very limited. This is particularly important in building design, as the typical building design professional is not an expert in fluid dynamics, turbulence modeling, numerical mathematics and high-performance computing. It is unrealistic to expect that building professionals would become experts in this area, rather it is important to provide practitioners with clear guidance on best practices.

In this paper, we evaluate the impact of selection of turbulence model on Cp values. We compare the k-epsilon model, which is by far the most widely used model in the building design community, to the better-suited Spalart-Allmares model. We conduct a 3D RANS simulation for an isolated box-shaped building using the standard k-epsilon model and the Spalart-Allmaras model and then compare the results to data from wind tunnel experiments. Finally, a comparison is made between the Cp values obtained from the simulation and those used in airflow network models. The intent is to evaluate the appropriateness of the airflow network default values and demonstrate that CFD can be used as a tool in obtaining reliable Cp values, while also drawing attention to potential sources of errors.

## Introduction

Whole building energy simulation is widely used in the design and optimization of buildings in order to reduce building energy needs, which is a major portion of global energy use. Natural ventilation can be a cost-effective energy efficiency measure and is used in traditional as well as in modern architectures. However, predicting thermal comfort and energy consumption in natural ventilation mode can be challenging as the methods to calculate the amount of airflow entering the building are of relatively low accuracy, especially compared to other engineering disciplines. Air infiltration and natural ventilation significantly impact both the indoor environment – mainly thermal comfort and air quality – and energy demand (Foucquier et al. 2013). Therefore, careful consideration of the external wind pressure distribution on building surfaces and the wind's interaction with building components and systems is required (Meroney 2009). Most airflow network programs use the pressure coefficient (Cp) as a key parameter to characterize the flow at the opening, derive the wind pressure at this location and base on this estimate the volume of air entering the space through a small opening. Given that Cp-values so strongly affect wind-caused airflow into space (Good et al. 2008), it is important to choose it appropriately. Typically, the Cp-values incorporated in whole building simulation software are taken from the wind tunnel experiment database published by the Air Infiltration and Ventilation Centre (AIVC) (Liddament 1986). For example, CONTAMW and BSim use AIVC Cp values as the data source for low-rise buildings and IES VE and Tas implement it for high-rise buildings (Costola et al. 2009). The application of data from this source into airflow network models, however, suffers from several limitations (Costola et al. 2009):

- The method used to convert the experimental data to the database is unclear;
- The building geometry, sheltering elements, wind profile provided in this publication are lacking in detail information and are insufficient to be used in most specific real cases;
- The results, which are given by surface averaged values over the whole surface and averaged value for the "front" and the "rear" part of the roof, are deficient in describing the actual characteristics without knowing the opening size and location.

These limitations all raise uncertainty of the building simulation performance and thus it is important to understand the impact of simplifying a model and using the default inputs based on this data, or whether it might be necessary to use more reliable data as input parameters.

To practitioners this is important in a number of ways, mainly related to the reliability of designs achieving their targets, and the risk of failure, in other words actual achieved performance, as well as to the question of consistency across studies used to demonstrate appropriateness of design to Authorities Having Jurisdiction (AHJ). In the context of airflow network simulations, the most relevant application is the analysis of spaces without mechanical cooling for the risk of overheating. A variety of design decisions can be based on such studies, which can impact the cost of a building significantly, for example if shading devices are


https://doi.org/10.26868/25222708.2017.650

recommended, or if the conclusion is that mechanical cooling is needed. At the same time, if the study concludes that mechanical cooling is not necessary and the building is built accordingly, if this turns out to have been wrong, remediation is very difficult and comes at high cost. Because of this, building performance engineers regularly face the question around the topic of how reliable the results are, how big the margin of error might be, and what level of risk needs to be carried in the project budget to protect against failure.

An example of the impact of the type of analysis accepted by AHJs is the case when eliminating mechanical cooling is only acceptable to an AHJ if an analysis is submitted that demonstrates that overheating will not be a problem. The results of such an analysis are greatly affected by the choice of Cp values, and a very diligent building simulation practitioner who uses more appropriate values as opposed to a less experienced practitioner who uses overly optimistic default values might add hundreds of thousands of dollars in requirements of for example shading devices to a residential high-rise, which would be shown as not necessary in a study with different treatment of wind. Such differences in recommendations of different practitioners for very similar projects cause significant confusion in the construction industry.

Computational Fluid Dynamics (CFD) has demonstrated an ability to provide a detailed description of flows around buildings and is widely used in the field of wind engineering, albeit with its own limitations. For conciseness and as the background literature on CFD is extensive, for more information on the theory and models referenced below we refer the reader to the text book by S.B. Pope (Pope 2000), which contains also a comprehensive list of references to the original research.

Obtaining pressure coefficient data from CFD simulation for use as input variables in an airflow network model could overcome, or at least significantly reduce, the shortcomings of the default values in airflow network models and should be able to yield higher accuracy. Integrating an airflow network model and CFD could provide an acceptable compromise between the two extreme options of either using default values or performing wind tunnel testing on an individual building design and yield a simulation approach leading to more accurate and reliable results of building performance.

While building simulation practitioners use CFD in their practice based on the above understanding, so far the applicability has not been demonstrated and there is no clear understanding of the level of accuracy and error.

The airflows around buildings are essentially always turbulent, given that they typically involve large Reynolds numbers and an interaction with the atmospheric boundary layer. CFD of these flows is thus challenging as the computational cost of solving the flow governing equations directly (that is, without providing some kind of turbulence model) is prohibitively high, primarily because of the need to provide an extremely fine spatial discretization (mesh) in order to resolve very fine-scale structures in the turbulence on the order of the Kolmogorov length-scale. An alternative would be Large Eddy Simulation (LES) - where one would solve the spatially filtered governing equations and then provide a model for the effects of the unresolved motions at sub-filter scales - however this leads to an intrinsically unsteady solution that one then needs to average in time. Furthermore, LES nevertheless still needs a rather fine mesh, which – coupled with the need to solve for the unsteady condition evolving in time and to average over several flow-through times to obtain statistically converged results – makes LES challenging as a design tool for buildings. While there are many design questions that necessitate using more advanced models such as LES, we are investigating if, for the purpose of calculating Cp values, simpler turbulence models might be sufficient.

The simpler alternative is using steady Reynolds-averaged Navier–Stokes (RANS), where the governing equations for the flow are averaged in time. The resulting equations have an unclosed term, the Reynolds stresses, which has been found to have a diffusive effect on momentum transport; this has led to a broad class of RANS turbulence models which close the Reynolds stresses by modeling them with an enhanced, "turbulent viscosity". Within that class of turbulence models, a very common approach is the k-epsilon model of Launder and Spaulding, commonly referred to as the standard k-epsilon model; this model has become nearly ubiquitous, such that commercial CFD packages often default to this model or even have only this one model available. The k-epsilon model is perhaps best viewed as an empirical fit to a large suite of experimental data: there are four tunable parameters in transport equations for the turbulent kinetic energy k and epsilon, the dissipation rate of the turbulent kinetic energy k. The turbulent viscosity is a fifth tunable parameter multiplied by the ratio of these two quantities. The four tunable parameters in the transport equations are typically set to "standard" values; the standard values have been chosen to fit many different flows, including internal and external flows. With the work being presented here, and for buildings interacting with wind in general, the Spalart-Allmaras model, which is used extensively in aerospace engineering applications, appears to possibly be more appropriate as we intend to study only the external flow around a bluff body (the building). The Spalart-Allmaras model was developed specifically for external flows and involves solving a transport equation for the turbulent viscosity directly, again with several tunable parameters, but now having been tuned only to external flows, such as boundary layers and flows around bluff bodies. Furthermore, unlike the k-epsilon model, it solves the whole field instead of adding wall functions, which do not perform well in the viscous sublayer (Ariff et al. 2009), or atmospheric boundary layer.

This paper presents 3D steady RANS modeling of the airflow around a 3-storey box-shaped building using both the standard k-epsilon and Spalart-Allmaras models. The CFD results are validated against wind tunnel experiments (Hölscher et al. 1998). The difference between the CFD results and airflow network default values are then compared.

## Method

### Computational domain and grid

The size of the computational domain was selected according to the best practice guidelines by Franke et al. (2007). For the single building of height h, the vertical and lateral extension of the domain are set to be 6h and 5h respectively. In the flow direction, the distance between the inflow boundary and the building is set to be 5h. The region behind the building is selected to be 15h away from the outflow boundary to allow re-development of the flow. We choose a cube with edge length 10m as the building, as this matches available experimental data. The final dimensions of the domain are $W \times D \times H = 110\,m \times 210\,m \times 70\,m$, with resulting the blockage ratio being 0.06%.

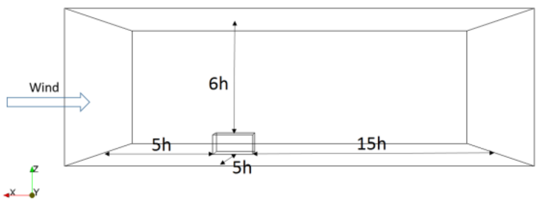



*Figure 1: Computational domain setup.*

The grid used in this simulation has 418,207 tetrahedral cells. Figure 2 shows the grid convergence study, demonstrating that the grid used is sufficiently fine to appropriately numerically solve the governing set of partial differential equations. Plotted in the figure is the pressure coefficient value along the 1-2-3-4 trajectory on the building surface, which will be explained in detail in the results section. The figure is shown here to demonstrate appropriateness of the selected grid and therefore validity of the results shown in detail below. Figure 2 shows results for three different grids: the reference grid (Grid C) with 418,207 cells, and two other coarser grids with 23,947 cells (Grid A) and 144,605 cells (Grid B) respectively. The result shows that the reference grid we selected is suitable for this simulation.

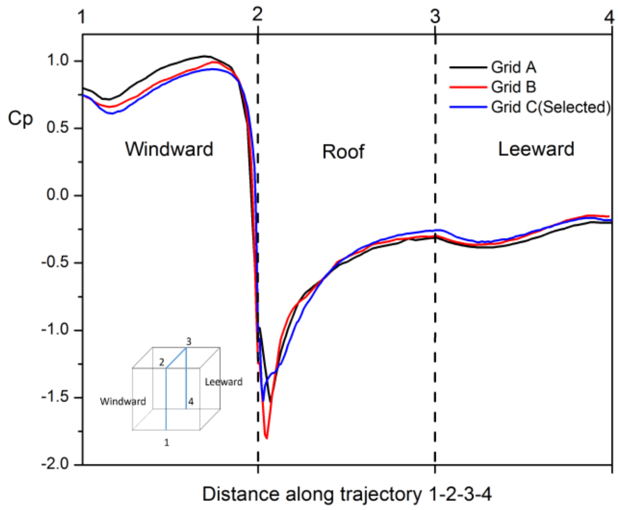



*Figure 2: Grid convergence analysis.*

### Inflow boundary conditions

The inflow boundary conditions are set to be a fully developed vertical profile of wind speed and turbulent viscosity, which is specified using a logarithmic profile for wind velocity and a profile for the turbulent viscosity calculated from turbulent kinetic energy k and dissipation rate of turbulent kinetic energy epsilon (Hargreaves and Wright 2006):

$$U = \frac{u_*}{\kappa} \ln\left(\frac{z+z_0}{z_0}\right). \tag{1}$$

$$k = \frac{u_*^2}{\sqrt{C_\mu}}. \tag{2}$$

$$\varepsilon = \frac{u_*^3}{\kappa(z+z_0)}. \tag{3}$$

$$\nu_t = C_\mu \frac{\sqrt{k}}{\varepsilon}. \tag{4}$$

$$u_* = \kappa \frac{U_{ref}}{\ln\left(\frac{z+z_0}{z_0}\right)}. \tag{5}$$

Eq.(1) gives the vertical profile of the inlet velocity, where $u_*$ is the friction velocity, $\kappa$ is the von Karman constant, $z$ the height coordinate, $z_0$ the surface roughness and $U_{ref}$ the reference velocity at reference height (Eqs. 2-5). The turbulent viscosity is a function of k and epsilon, where $C_\mu$ is the turbulent viscosity coefficient.

In this case, the inflow Reynolds number at reference height (10 m) is $2 \times 10^5$.

## Results and Discussions

In order to evaluate the appropriateness of Cp values from CFD calculations, experimental data to compare to is needed. Hölscher et al. (1998) published data from wind tunnel experiments of a 10m x 10m x 10m cube. They report Cp values along the mid-span, starting in the middle of the bottom edge of the windward face. This bottom edge location is labeled 1, the middle of the top edge is location 2, the middle of the top edge of the leeward face is labeled 3, and lastly the middle of the bottom edge of the leeward face is labeled 4. Along the trajectory 1-2-3-4 five measurements are taken on each surface, windward face, top/roof, and leeward face, resulting in 15 measurement points in total.

The corresponding metric in the CFD simulations is the resolution of the computational grid along the 1-2-3-4 trajectory, which leads to two orders of magnitude more individual data points along the trajectory and thus a much finer resolution of the pressure coefficient.

Figure 3 compares pressure coefficient values along the mid-span, or 1-2-3-4 trajectory, for a wind direction of 0°. The wind tunnel experiment data (Hölscher et al. 1998) is represented by black squares, and for easier comparison connected by black lines. The CFD calculations are represented by the blue and red lines, showing the results for the standard k-epsilon model and the Spalart-Allmaras model respectively. In addition, the pink dashed line shows the typical airflow network model default values from the AIVC database (Liddament 1986).

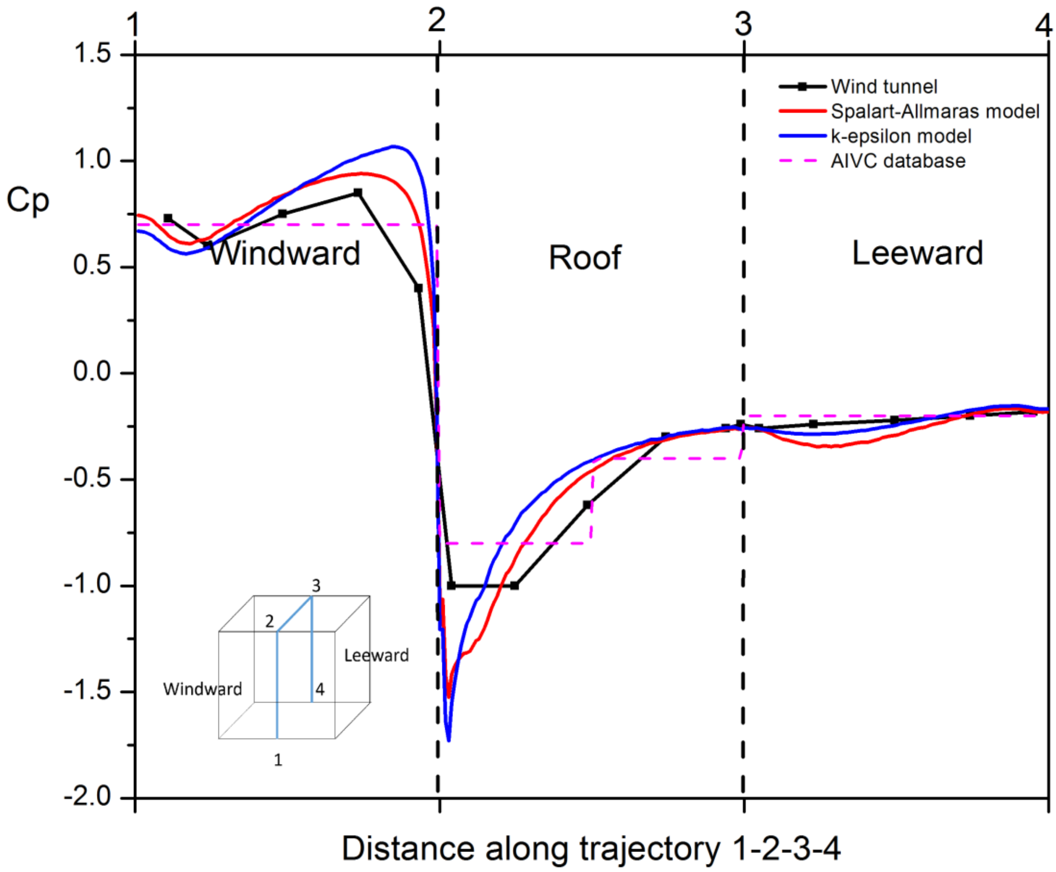



*Figure 3: Result comparison between wind tunnel experiment and CFD simulation.*

.

Overall, the CFD results match the wind tunnel experiments fairly well; the Cp values from the AIVC database, which are derived from a different set of wind tunnel measurements and then surface averaged, are confirmed to be an appropriate surface average for the present wind tunnel measurement as well. Not much detail is given in Liddament (1986) about the details of the wind tunnel measurements, for example the Reynolds number is not given, so it is difficult to judge how universal this match is; here, however, the AIVC database values happen to match well. The reference values used in most of the airflow network models that are taken from the AIVC database is given in Table 1.

*Table 1: Reference data from AIVC (0° wind angle, 1:1 length to width ratio, exposed, low-rise buildings)*

| | Windward | Leeward | Roof | |
|---|---|---|---|---|
| | | | Front | Rear |
| Surface Averaged Cp-value | 0.7 | -0.2 | -0.8 | -0.4 |

On the windward side the CFD calculations seem to overestimate the pressure coefficient value somewhat, with the Spalart-Allmaras model being much closer to the wind tunnel measurements than the k-epsilon model. While the difference between the two turbulence models might not look too significant at first sight, a prediction for the airflow into a 3rd story window could be overpredicted by as much as 30% – 40%, which could for example drastically change the result of an analysis for summer overheating. The overestimation of the airflow by the k-epsilon model could lead to an overprediction of the heat that can be removed by natural ventilation, be it from solar loads or from internal loads, and thus prevent an actual overheating problem from being identified. The Spalart-Allmaras model still overpredicts the pressure coefficient somewhat, which is important to know when relying on Cp values from CFD in design calculations.

More information on the pressure distribution on the front face is shown in Figure 4 (a) for the standard k-epsilon model and in Figure 5 (a) for the Spalart-Allmaras model through pressure coefficient contours on the surface. The pressure varies significantly not only along the mid-span but also across the surface. This further demonstrates the limitation of using area averaged Cp values, as, depending on the location of an opening on the surface, the Cp value could be as low as 0.4 or above 0.9, compared to a surface averaged value of 0.7 (Table 1).

The windward face mostly experiences pressure from the wind impact. Because of the profile of the incoming wind condition in the atmospheric boundary layer, the highest pressure is in the upper part of the building, before the impact of the edge becomes noticeable. In the lower part of the face, because of the constraint, the ground is placing on the airflow, there is a recirculation zone, or eddy, leading to an area of lower pressure in the bottom third.

As the flow separates from the building at the top edge of the windward face, the pressure coefficient changes dramatically, as can be seen both in Figure 3 and in Figures 4 (b) and 5 (b). Here the wind tunnel experiment is limited by the resolution achieved with the five measurement points on the roof. Both CFD models predict a very sharp drop in pressure at the leading edge of the flow over the roof. This is in full agreement with a vast range of experimental and computational data about flow separation, and about the low pressure in the recirculation zone below the separated flow. Further along the roof the flow re-attaches and the pressure steeply increases again. As can be seen in Figures 4 and 5 (b), different from the windward face, on the roof the pattern is fairly consistent across the face. The dominant effect here is detachment and re-attachment of the parallel atmospheric boundary layer flow.

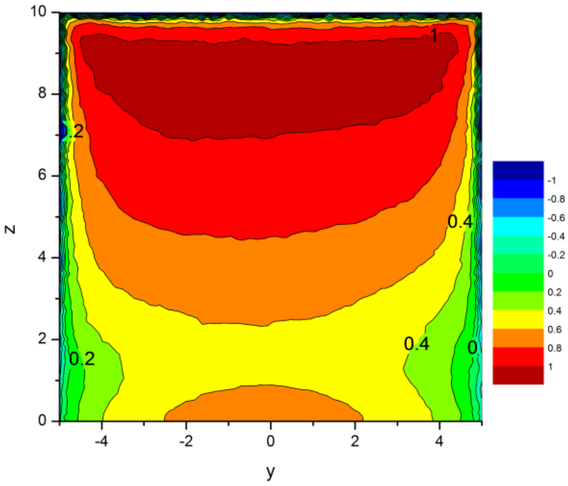
While the CFD simulations resolve this effect much better, it is difficult to know how accurate the exact values in the pressure drop are. As the Spalart-Allmaras model was specifically developed to address such phenomena, as they are most important in aerospace scenarios, it is expected to provide the better estimate.

Recognizing the dramatic difference in pressures on the roof, the surface averaged Cp values are split in a value for the first half of the roof and a value for the second half of the roof. However, opening location on the roof will strongly affect airflow, much beyond whether it is located in the first half or second half.

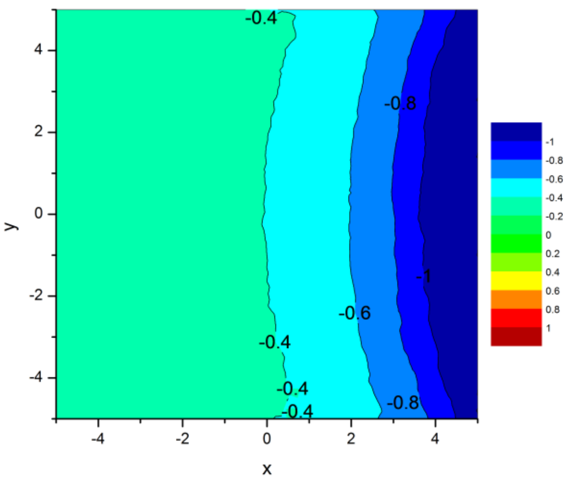
Another important implication of the flow pattern along the roof is that contaminants and odours can be caught in an eddy in a detached recirculation zone and brought into an opening even if the pressure is low and the overall flow direction is out of the opening, because of turbulent diffusion. Also, used air, leaving the opening due to low pressure (on average), could nonetheless be re-entrained.

The detached flow behind the building leads to a more straightforwardly predictable pressure, therefore the wind tunnel measurements, the CFD simulations with both turbulence models, and the surface averaged Cp value show a good agreement on the leeward surface. The detailed flow pattern can be expected to be an eddy behind the building, and vortex shedding off the leeward edge of the roof, which is the top edge of the leeward face. The vortex shedding can be expected to result in minor periodic pressure fluctuations. Again, recirculations zones carry the risk of contaminants or odours being caught in the eddy and re-entrained into the building, which is not shown by the Cp values of any of the shown approaches.

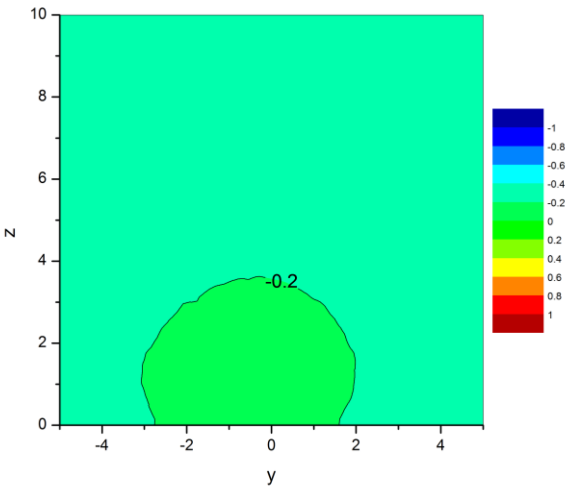
Overall, as expected, the Spalart-Allmaras model produces a more accurate result when comparing with wind tunnel data in the case shown here.




*(a)*

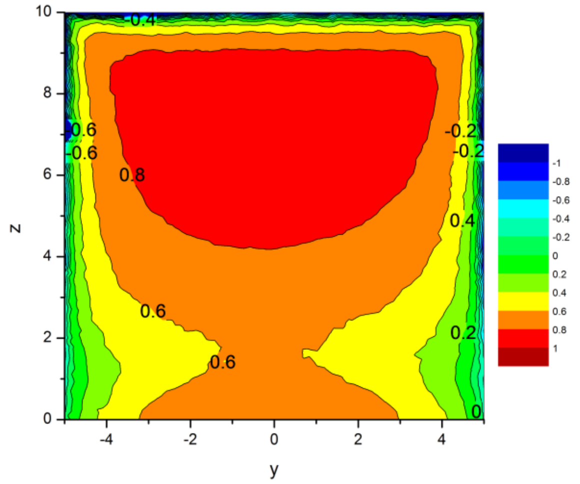



*(b)*

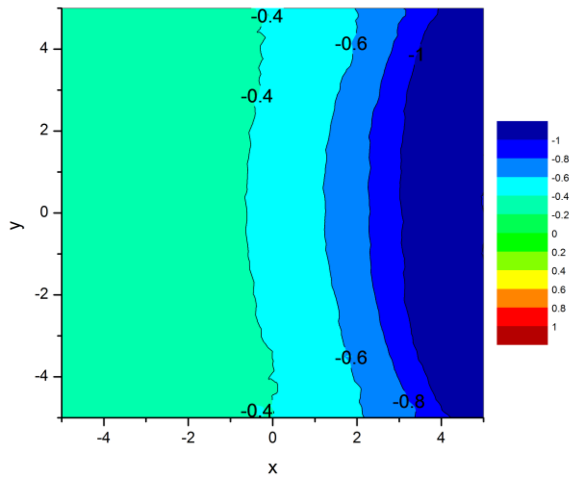



*(c)*

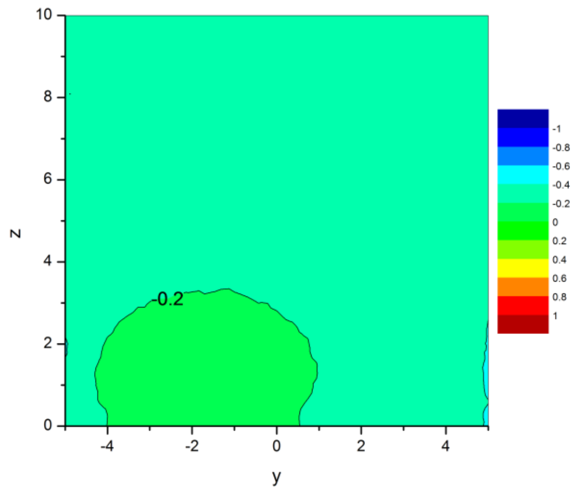
*Figure 4: Pressure coefficient contour plot for k-epsilon model: (a) windward-side; (b) roof; (c)leeward-side.*




*(a)*




*(b)*




*(c)*

*Figure 5: Pressure coefficient contour plot for Spalart-Allmaras model: (a) windward-side; (b) roof; (c)leeward-side.*

The present work stays within the standard assumption made in airflow network modeling that the openings are so small relative to the surface they are in that the opening does not influence the airflow around the building. However, at least locally at the opening, this may not be the case. As the airflow through the opening depends on the local flow characteristics at the opening, there could be a significant impact on the airflow into a space. Future work could be done in more complex conditions, including providing different sizes and locations of building openings.

Another potential extension of the present study could be to evaluate more turbulence models. While steady RANS is the most commonly used turbulence model in building airflow simulations (Blocken 2014), because of the advantage of lower computational effort, it is nonetheless deficient in resolving coherent flow patterns like large eddies. Large Eddy Simulation (LES) and hybrid RANS/LES techniques like Detached Eddy Simulation (DES) are capable of predicting unsteady effects in a more consistent way (Hargreaves and Wright 2006). DES for example could resolve the vortex shedding at the leeward edge of the roof.

## Discussion

The main contribution of the present paper is, as a start to developing appropriate CFD procedures for wind pressure coefficients for practitioners in the building design community. This reduces the need for specialty expertise while at the same time reducing the amount of error in current design decisions based on potentially flawed CFD analyses and interpretations.

One example of a challenge present in interpreting CFD results, which is particularly present in CFD in the built environment as a result of the inevitable relatively coarse meshes as compared to other disciplines, is the over-interpretation of details and structures in the results. Figure 6 shows the same data as Figure 5, only this time plotted with colour contours set to the data range of every figure individually, as many software tools will provide by default. Especially Figure 6 (c) creates the impression of a lot of relevant detail, and the question of why the result is not symmetrical, as would be expected. We've taken at face value without questioning the meaning of the data variations, this could lead to a design decision around opening placements based on those patterns in the result. Comparing Figure 6 (c) to Figure 5 (c), with the only difference being that the colour contours in Figures 4 and 5 are set equal for all building faces, representing the whole range of variation, a lot less detail is visible. This is because the colour coding approach used in Figure 6 and by default in many software products will emphasize differences, even if they are very small to negligible. In the present simulations, these variations represent the small variations across the simulation domain/ across the mesh that will remain as a result of the numerical calculations. While these variations decrease with better mesh resolution and better convergence of results and eventually become very small, they will not become zero.

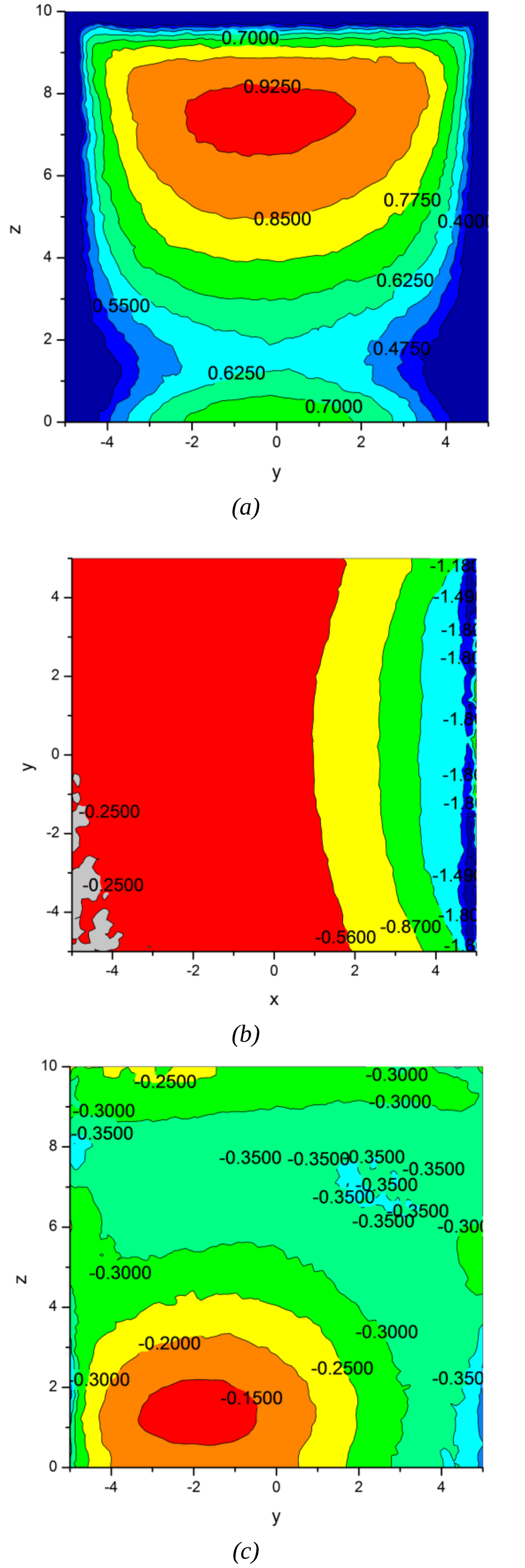



*Figure 6: Pressure coefficient contour plot for Spalart-Allmaras model, varied colour scheme: (a) windward-side; (b) roof; (c)leeward-side.*

## Summary and Conclusions

In this paper we conduct a 3D RANS simulation for wind flowing around a cubic building for two different turbulence models, the standard k-epsilon model and the Spalart-Allmaras model. The calculations are validated using data from wind tunnel experiments. In general both turbulence models fairly closely match the experiments, however, particularly close to the top edge of the windward face, both at the top of that face as well as on the other side of this edge, the front of the roof, the Spalart-Allmaras model shows its superiority in representing the detaching and re-attaching of outside flows. This is to be expected, as the Spalart-Allmaras model was developed in aerospace engineering specifically for this type of flow scenario.

A comparison is made between the Cp values obtained from the simulation and those typically used in airflow network models, showing the great loss of detail when being reduced to just surface averaged values.

The simulation results show that CFD is capable of producing useful pressure coefficient data. Moreover, the simplification in the default Cp value in most airflow network models over-simplifies the information and it is, therefore, inappropriate to be used if seeking higher accuracy. Particularly the surface averaged Cp values lack the ability of informing on the impact of opening location on a building surface.

## Acknowledgement

Special thanks go to Institute for Computing, Information and Cognitive Systems (ICICS) for providing support through ICICS travel grant, which enables me to attend the BS2017 conference.

## References

Foucquier, A., Robert, S., Suard, F., Stéphan, L., & Jay, A. (2013). State of the art in building modeling and energy performances prediction: A review. *Renewable and Sustainable Energy Reviews*, 23, 272-288.

Meroney, R. N. (2009). CFD Prediction of Airflow in Buildings for Natural Ventilation. *Wind Engineering*.

Good, J., Frisque, A., & Phillips, D. (2008). The role of wind in natural ventilation simulations using airflow network models. *Proceedings of SimBuild, 30*, 693-708.

Liddament, M. W. (1986). Air infiltration calculation techniques: an applications guide. Air infiltration and ventilation centre.

Costola, D., Blocken, B., & Hensen, J. L. M. (2009). Overview of pressure coefficient data in building energy simulation and airflow network programs. *Building and Environment, 44(10)*, 2027-2036.

Pope, S.B. (2000). Turbulent Flows, Cambridge University Press

Franke, J. (Ed.). (2007). Best practice guideline for the CFD simulation of flows in the urban environment. Meteorological Inst..

Hargreaves, D. M., & Wright, N. G. (2006). The use of commercial CFD software to model the atmospheric boundary layer. *In Proceedings of the Fourth International Symposium on Computational Wind Engineering*, 797-800.

Ramponi, R., & Blocken, B. (2012). CFD simulation of cross-ventilation for a generic isolated building: impact of computational parameters. *Building and Environment*, 53, 34-48.

Ariff, M., Salim, S. M., & Cheah, S. C. (2009). Wall y+ approach for dealing with turbulent flow over a surface mounted cube: part 1—low Reynolds number. *In Proceedings of the 7th International Conference on CFD in the Minerals and Process Industries*, Melbourne, Australia (pp. 9-11).

Hölscher, N., & Niemann, H. J. (1998). Towards quality assurance for wind tunnel tests: A comparative testing program of the Windtechnologische Gesellschaft. *Journal of Wind Engineering and Industrial Aerodynamics*, 74, 599-608.

Blocken, B. (2014). 50 years of computational wind engineering: past, present and future. *Journal of Wind Engineering and Industrial Aerodynamics*, 129, 69-102.