Title: **Will CFD ever Replace Wind Tunnels for Building Wind Simulations?**

Authors: Duncan Phillips, RWDI
Michael Soligo, RWDI

Subject: Architectural/Design

Keywords: CFD
Wind Engineering
Wind Tunnel Testing

Publication Date: 2019

Original Publication: International Journal of High-Rise Buildings Volume 8 Number 2

Paper Type:
1. Book chapter/Part chapter
2. **Journal paper**
3. Conference proceeding
4. Unpublished conference paper
5. Magazine article
6. Unpublished

International Journal of High-Rise Buildings
June 2019, Vol 8, No 2, 107-116
https://doi.org/10.21022/IJHRB.2019.8.2.107

International Journal of
High-Rise Buildings
www.ctbuh-korea.org/ijhrb/index.php

Will CFD ever Replace Wind Tunnels for Building Wind Simulations?

Duncan A. Phillips† and Michael J. Soligo

RWDI, 600 Southgate Dr., Guelph, Ontario, CANADA, N1G 4P6

Abstract

The use of computational fluid dynamics (CFD) is becoming an increasingly popular means to model wind flows in and around buildings. The first published application of CFD to both indoor and outdoor building airflows was in the 1970's. Since then, CFD usage has expanded to include different aspects of building design. Wind tunnel testing (WTT) on buildings for wind loads goes back as far as 1908. Gustave Eiffel built a pair of wind tunnels in 1908 and 1912. Using these he published wind loads on an aircraft hangar in 1919 as cited in Hoerner (1965 – page 74). The second of these wind tunnels is still in use today for tests including building design (Damljanović, 2012). The Empire State Building was tested in 1933 in smooth flow – see Baskaran (1993). The World Trade Center Twin Towers in New York City were wind tunnel tested in the mid-sixties for both wind loads, at Colorado State University (CSU) and the [US] National Physical Laboratory (NPL), as well as pedestrian level winds (PLW) at the University of Western Ontario (UWO) – Baskaran (1993). Since then, the understanding of the planetary boundary layer, recognition of the structures of turbulent wakes, instrumentation, methodologies and analysis have been continuously refined. There is a drive to replace WTT with computational methods, with the rationale that CFD is quicker, less expensive and gives more information and control to the architects. However, there is little information available to building owners and architects on the limitations of CFD for flows around buildings and communities. Hence building owners, developers, engineers and architects are not aware of the risks they incur by using CFD for different studies, traditionally conducted using wind tunnels. This paper will explain what needs to happen for CFD to replace wind tunnels. Ultimately, we anticipate the reader will come to the same conclusion that we have drawn: both WTT and CFD will continue to play important roles in building and infrastructure design. The most pressing challenge for the design and engineering community is to understand the strengths and limitations of each tool so that they can leverage and exploit the benefits that each offers while adhering to our moral and professional obligation to hold paramount the safety, health, and welfare of the public.

Keywords: CFD, Wind tunnel testing, Building design

1. Introduction

Computational Fluid Dynamics (CFD) is an airflow simulation tool that has become increasingly important within the building design community. The first use of CFD for indoor building airflow analysis was presented in Nielsen (1973). Blocken (2018) cites Yamada and Meroney (1971) as doing the first work related to simulation of airflow around buildings. Since then, CFD usage has expanded to include analysis of different issues for flows both within buildings and outside. This paper is concerned with flows outside buildings: for a review of CFD for indoor flows see Neilsen (2015).

Regardless of which tool is used (wind tunnel or CFD); wind flow analysis is an important part of building design. The different types of studies that are conducted include assessing pedestrian level winds (PLW), structural and cladding wind loads, near-building pollutant dispersion, snow & sand drifting, rain infiltration and natural ventilation among others. Related wind studies include assessments of construction safety and operational issues, entertainment (e.g. outdoor sporting /cultural events / art installations) and even movie shoots.

Wind tunnel testing (WTT) on buildings for wind loads was pioneered during the early 1900's (e.g., Gustave Eiffel's work on hangers) with the design of the World Trade Center in New York (1960's) being the first major tall building tested in a boundary layer wind tunnel. Since then, the sophistication of these physical experiments has increased, instrumentation and analysis have evolved and our understanding and representation of the planetary boundary layer has improved. As a result of this work, building aerodynamicists have built a body of knowledge recognizing the importance that building separation and wakes, upwind turbulence and building aerodynamics all have on wind forces acting on the building and on ground level winds speeds. These features must be captured and represented accurately for any wind flow analysis to be useful.

There is a drive to replace WTT with computational

†Corresponding author: Duncan Phillips
Tel: +1-519-823-10311
E-mail: Duncan.Phillips@RWDI.com

methods, with the rationale that CFD is quicker, less expensive and gives more information and control to the architects. However, there is little information available to building owners and architects on the limitations of CFD for external flows around buildings and within the built environment. Computational tools are available, sometimes for free, that permit the user to run a CFD simulation easily. In other cases, online tools are available that offer quick simulation, but these tools are "black boxes" and the user has no way to know whether the correct boundary layer is applied, if inlet turbulence is present, or how well the building geometry is represented within the computational grid.

Towards the end of his paper, Blocken (2018) identifies a concern that far too often CFD is being conducted by users (not practitioners) who do not understand fluid flow or simulation limitations and that the ease of access is overshadowing the need to understand. This thought is shared by Spalart and Venkatakrishnan (2016) who describe this trend as "… overconfidence and under-competence in CFD". Firms that propose using CFD alone are rarely challenged by facts on its limitations nor their usage of the tool; but they should be as this is critical to the accuracy and safety of design. Hence building owners, developers, engineers and architects are frequently unaware of the risks they incur by using CFD for different studies traditionally conducted using wind tunnels. Indeed, there are some engineers who are also caught in the trap created by the combination of compelling graphics, an ability to perform different analytical calculations quickly and a desire to revise a model and rerun the simulation.

The purpose of this paper is not to strike fear into the hearts of CFD users. Instead, this paper intends to explain what is required to happen for CFD to catch up to wind tunnels. Ultimately the authors believe that both CFD and WTT have a role in future building design. This paper addresses outdoor wind simulation only - not indoor flows. It has been written for those with a general understanding of what CFD can do but not the shortcomings. This paper does not get into the detailed mathematics but rather a description of what that mathematics is and where the limitations lie. For specific guidance on conducting CFD simulations of the urban realm see references such as Blocken (2015).

2. Background on CFD Modeling

2.1. Navier Stokes, Reynolds and k-ε

It is instructive to recognize the foundation of CFD modeling. Most CFD software products solve a set of equations known as the Navier Stokes (NS) equations. These equations are based on two conservation laws: the conservation of mass and the conservation of momentum (Newton's 2nd Law) and were developed by Navier and Stokes in the 1880's - see Reynolds (1894). The resulting four equations are time varying, non-linear and highly inter-connected. Regardless of how these equations are solved, you end up with a field of time varying velocities and pressures. Unfortunately, there are only a few analytical solutions to these equations and these are for flows that are quasi-steady state, effectively confined (e.g., constrained by boundaries) and essentially in the non-turbulent flow regime.

To extend the applicability outside the analytical solutions, there are effectively two means of solving these equations: direct numerical simulation (DNS) or simplifying the non-linearity and interconnectedness. At the time of writing, DNS is only practical for fundamental research investigations of simple problems. A third option of considering an alternate form of fluid flow analysis is addressed below.

A solution to deal with the transient nature of fluid flow was proposed by Osborne Reynolds in the 1890's (Reynolds, 1895). This was to break (decompose) the time varying velocities into a mean and fluctuating component, insert them into the Navier-Stokes equations, and average the equations. This is how the Reynolds Average Navier Stokes (RANS) equations came into being. One of the consequences of this mathematical formulation is that a set of stress tensors are created as second derivatives. These stress tensors need to be modeled in order to solve the RANS equations and this is how the term turbulence closure came into being.

It is the closure to these stress tensors that lead to the turbulence models that CFD users are most familiar with. Fundamental to these models is that the impact of the stress tensors can be treated as a form of viscosity often referred to as the eddy viscosity. The consequence of this assumption is that turbulence is isotropic (e.g., uniform in all directions) and that the models are "tuned" based on the turbulent flows they were compared to. This tuning is important as the constants that exist in the turbulence closures are not necessarily universally applicable. The isotropic assumption is also a flaw because most flows generate turbulence in one direction and that is transferred to the others through flow stresses. This means that turbulent fluctuations can be stronger in one direction than another.

Most CFD practitioners use a two-equation model that is a variant of the k-ε (pronounced "k-epsilon") model. This model adds two equations to the Navier Stokes equations to account for the transport of turbulence kinetic energy (k) and dissipation (ε). There are other variants to this (e.g., RNG, realizable k-ε) and different authors have cited the benefits of one another - e.g., Thet Mon Soe and San Yu Khaing (2017).

While the two equation models such as k-ε and its derivatives are intuitively easy to understand and are relatively stable mathematically, these turbulence closures cannot address complex flows such as those that include flow separation, swirling or recirculation zones nor sharp changes in geometry - sometimes referred to as "strong streamline curvature". The reader might have noticed that these features are all flow phenomena present when wind passes around buildings. There are a number of reasons for the failure of two-equation closures to reasonably simulate these features. These limitations prevent RANS based simulation models from accurately predicting flow around buildings. In fact, there is no RANS closure model with a single set of parameters that can accurately model multiple types of flow regimes. On the other hand, these limitations do not exist in wind tunnel tests since the flows are physically modeled using the same fluid properties hence the wind tunnel methodology very accurately represents these flow conditions. At the time of writing, most readily available CFD software uses these sorts of turbulence closures and this should be of concern to users. Hence a key question implicitly posed by a CFD user is "How accurate does it have to be?" but they do not follow-up with reflection on that question.

2.2. Large Eddy Simulation (LES)

In fluid flows the generation of turbulence occurs through shear caused by separation, recirculation and other rapid changes in flow velocity (speed and direction). As the large eddies break up, they create smaller ones. Conservation of momentum requires that the energy in the smaller eddies must equal that from which they came and, as the size of the eddies becomes smaller, they reach a size where the viscosity in the fluid dissipates this kinetic energy into internal energy (e.g., heat). In reality, viscosity operates at all scales, but it is a small component of energy loss until the eddies reach small sizes. As the eddies break down, the original directionality to the turbulence is also broken down and the turbulence becomes isotropic. This transition from non-homogeneous to isotropic is important. Just add to the complexities, small eddies can coalesce, reversing the energy cascade in a phenomenon called "backscatter" - see Piomelli et al. (1991).

In Kolmogorov (1941) he noted that large eddies of a flow are dependent on the geometry, while the small ones are not - there is a transition from homogeneous turbulence to isotropic. This means that at the small scales an isotropic assumption may be reasonable. Hence if one could "filter" the flow, so that large eddies are simulated directly and the effect of the small ones are modeled, a closer representation to real turbulent flow could be achieved. In 1963 Smagorinsky proposed this as a means to simulate atmospheric flows (Smagorinsky, 1963) and later in 1970 it was implemented in a channel flow (Deardorff, 1970). This is where the term "large eddy simulation" (LES) comes from although Deardorff did not coin the phrase. There are a few methods of selecting the filter and in some software implementations the "filter" is set by the size of the grid. This is a handy approach as it is computationally more costly to resolve the smaller eddies but there is some evidence to suggest they have a less important role in building flow simulations.

This is one reason why LES seems to be performing better for flows around buildings than RANS: an LES formulation directly solves the flow for the larger eddies and simplifies the treatment of small ones. By small, we mean eddies on the order of 10's of centimeters / inches for flows around buildings. If the application of interest is forces on a shading device, eddies of those sizes may play a role hence one needs to be careful about generalizations. However, this approach still permits one to reduce the computational effort.

2.3. Alternate CFD Types – Lattice Boltzman and Smoothed Particles

LES and RANS are two approaches to solving the Navier Stokes Equations. There are other approaches to simulating fluid flow that do not involve these equations. Instead the fluid is simulated through the use of particles. These particles contain information regarding flow speed, direction, energy, etc. and are tracked through the simulation domain. Hosain and Bel Fdhila (2015) offer a review of different meshed, meshless and hybrid approaches. Two are briefly described here.

Smoothed-particle hydrodynamics (SPH) is one approach that is meshless. In this approach the properties of the fluid are represented by particles and the variation of these properties are smoothed from each particle to those within a certain distance of it. The particles move within the simulation domain transporting these properties driven by representations of the fundamental forces (e.g., Newton's Laws). This approach does not require a grid as the particles can be anywhere. The simulations are by definition transient. The flow information is inferred from the density of the particles within the simulation and their properties. SPH has been used to great effect in free surface flows (e.g., spills or dam breaks). Challenges using SPH for building analysis include implementation of a turbulent inlet atmospheric boundary condition as well as calculating pressures on buildings from the particles. However, the mathematics of SPH are ideal for the use of graphical processor units (GPU's) which are more cost effective than traditional CPUs. This means that the approach can be readily scaled up which is anticipated to increase simulation accuracy. Efforts are underway to apply the approach to buildings although it is still a way off from practical implementation in building design.

Simply put, the **Lattice Boltzman Method (LBM)** solves the Boltzman equation on a lattice. Practically, this means that particles are arranged at the nodes of a lattice and each particle has a probability density function associated with it moving in a specific direction. The bulk fluid properties are the summation of the statistics of the individual particles. This approach is hybrid in that the methodology needs a lattice (mesh) but that lattice is easy to apply. It does however have a number of fundamental challenges including how to assign inlet turbulence to the particles entering the domain. This method too is ideal for implementation on GPU's.

2.4. Which CFD Method is Best for Outdoor Flows around Buildings?

Asking this question indicates an opportunity to learn more: they are all best for something. They all have advantages and limitations - some of which are discussed in the next section. In his review, Blocken (2018) refers to multiple publications from the literature and concludes that for PLW studies, the use of RANS CFD is still supported. This was confirmed by work done for the City of London to develop guidelines for PLW studies - see Shilston et al. (2018). The City of London permits RANS to be used but prohibits the use of lower-order turbulence closures. This work also documents a need to run at least 36 wind directions (at time of writing which may increase when the final document is released) wind directions and to use a standard wind climate model to eliminate errors introduced by users unfamiliar with interpretation and analysis of weather data. Conversely, Blocken (2018) notes that LES outperforms RANS for simulations of contaminant dispersion near a building. Finally, in an early comparison for the building design community, Murakami et al. (1996) noted that only LES came close to predicting the pressure distribution around a building when they compared four different turbulence approaches including k-ε. The k-ε model has not changed since that time, only the implementation of the inlet wind boundary conditions - see below.

3. Three Challenges to Using CFD in the Built Environment

It is the authors' opinion that three main challenges must be overcome before CFD modeling becomes as reliable and accurate as wind tunnel testing. These are described below.

3.1. First: Inlet Boundary Condition for Velocity and Turbulence

It is well understood that as one goes up in the atmosphere, wind speed changes as a function of height, usually increasing, although this is not always true as publications on the wind engineering for Burj Khalifa showed how a Shamal influences design - see Phillips et al. (2012). Both a logarithmic and power law relationship have been applied to represent the mean wind speed profiles for synoptic winds. There are differences of opinion which of these is better with some consensus suggesting that while the logarithmic law is more physical close to the ground, the power law is better in the urban canopy but both can work - see Lateb et al. (2016).

There is also a relationship between turbulence intensity and altitude. Richards and Hoxey (1993) are acknowledged as having developed the implementation for both turbulence intensity and mean velocity profiles into a RANS CFD simulation.

With the atmospheric boundary layer (ABL) reasonably well understood, it should be relatively easy to implement an equation to introduce these parameters into a flow field. Earlier in this paper, the differences between RANS and LES were described. Implementing the mean and turbulence characteristics of an urban boundary layer in RANS is relatively easy: one needs both the mean velocity profile and turbulence intensity as a function of height. However, as noted, RANS does not do a good job on flows within the urban realm. Therefore developing a boundary layer specification within an LES model is necessary. It turns out that this is actually quite difficult.

There are three primary methods typically cited by which an ABL can be implemented into LES:

- The **Precursor Method**, equivalent to Lund's (Lund et al., 1998) Recycling approach, uses an inlet zone setup where roughness blocks are implemented at ground level within the simulation in order to create the appropriate amount of shear at the ground. Shear creates turbulence and that is transported up into the boundary layer. The term recirculation is applied because at a plane sufficiently downstream, the resultant flow is fed back in to the inlet - thus creating an effective extension of the simulation domain. This approach has yielded satisfactory results for the turbulence profile and statistics - see Vasaturo, R. et al. (2018). Criticisms of this approach are that it burdens any LES simulation with additional computational effort and storage requirements. The authors' experience is that it can also be unstable and lock itself into a self-reinforcing unnatural flow pattern if one is not careful.
- The **Synthetic Eddy Method (SEM)** which is related somewhat to the Vortex Method, uses an approach where a velocity distribution is created through the superposition of many eddies, of different scales and locations, on the flow entering the domain - see Jarrin et al. (2006).
- The **Random Flow Generation** approach has many forms but ultimately these rely on using a prescribed spectrum (e.g., turbulence characteristics) for each velocity component and then creating a fluctuating inlet flow field based on these. While the turbulence statistics are usually appropriate, the velocity field does not have coherence – meaning the turbulent structures or eddies lack the proper physical structure, i.e., they do not look like eddies, and quickly break down as they enter the flow domain. Aboshosha et al. (2015) proposed a modification to this approach to improve the realism of the flow field and has met with some success.

The image below presents a snapshot of the turbulence created by all three of these approaches plus incremental improvements (e.g., MSSEM and CDFRG). Observations one can draw from this image is that the Lund recycling

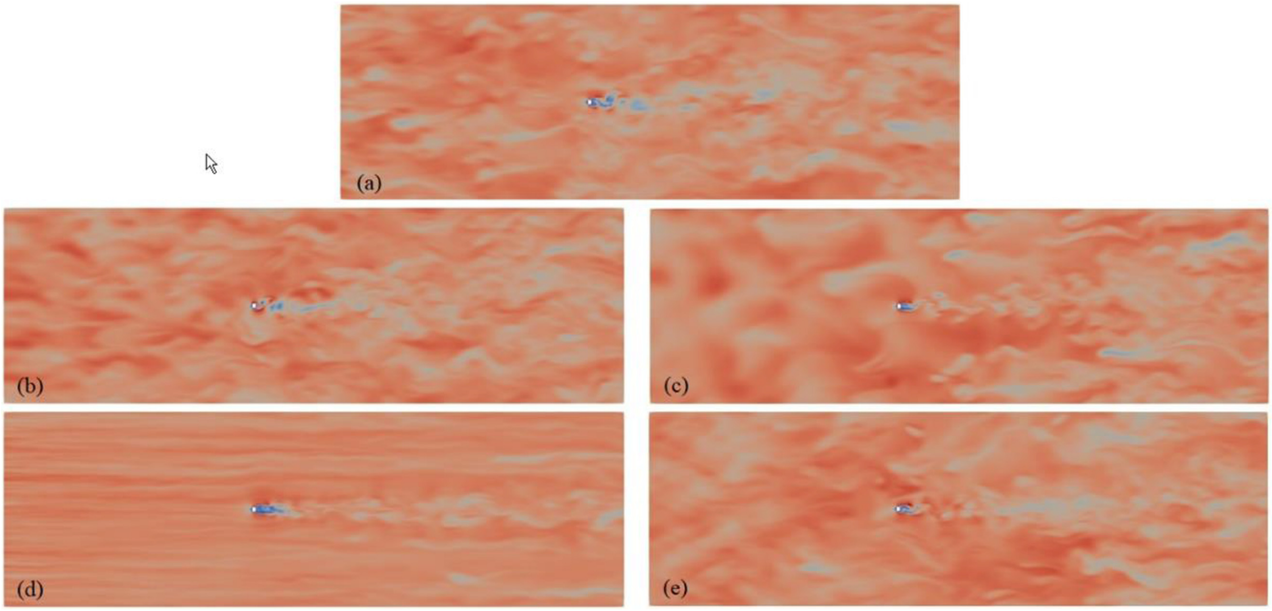

Figure 1. Instantaneous Velocity (e.g. a snap shop in time) in a Horizontal plane Using Different ABL Approaches: a) Lund's recycling approach, b) MSSEM = multi-scale synthetic eddy method, c) SEM = synthetic eddy method, d) DSRFG = discretizing and synthesizing random flow generation, e) CDRFG = consistent discrete random flow generation e.g. Aboshosha et al. (2015) – Image from Luo et al. (2018).

method (a) generates a waviness coupled with sudden changes in flow velocity; the SEM method (c) captures the waviness but there are not sudden changes in flow speed except behind the building; the DSRFG (d) method is more streaky than eddy-like upwind of the building; while both the MSSEM (b) and CDRFG (e) appear to catch both the waviness and flow velocity changes with the scales of the changes in the MSSEM slightly finer.

3.2. Second: Maintaining the Velocity and Turbulence Profiles into the Domain

Once the flow enters the simulation domain, the properties carefully set at the inlet need to be maintained to the buildings of interest. These buildings cannot be too close to the inlet otherwise the close proximity of the inlet forces an incorrect flow around them. References (e.g., Franke 2011) exist that suggest the inlet and outlet should be between 6 and 10 reference building heights from the building of interest. Hence, the CFD solver needs to transport the inlet conditions to the test buildings. One of the consequences of the mathematics used to convert the non-linear Navier-Stokes equation into something solvable is that an artificial diffusion is introduced. Simply put, unless energy is added into the flow, and the shear at the ground level is maintained, the wind flow will slump. In the image below, the higher speed air from above increasingly intrudes into the zone closer to the ground as the flow moves along the simulation domain. The streamlines in the image to the right also show this. These results are from the trials of our colleagues but the behavior is common as shown in the next image - see also Aliabadi et al. (2018).

There are methods which can be implemented to improve the persistence of the ABL profiles both for velocity and turbulence. One such method is to add physical roughness elements along the ground surface like one does in

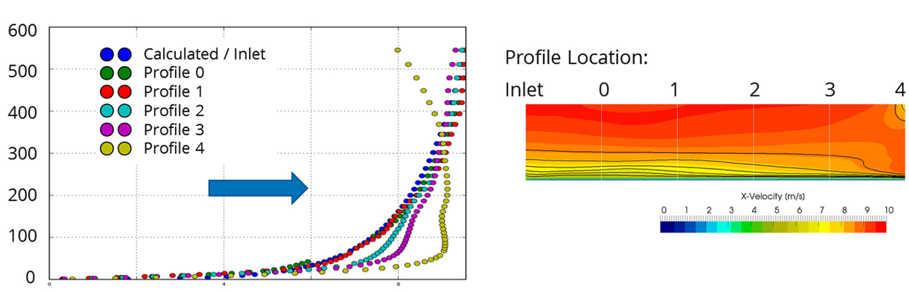

Figure 2. Mean Wind Speed Profiles (left) and Streamlines (right) for an LES Simulation of Channel Flow.

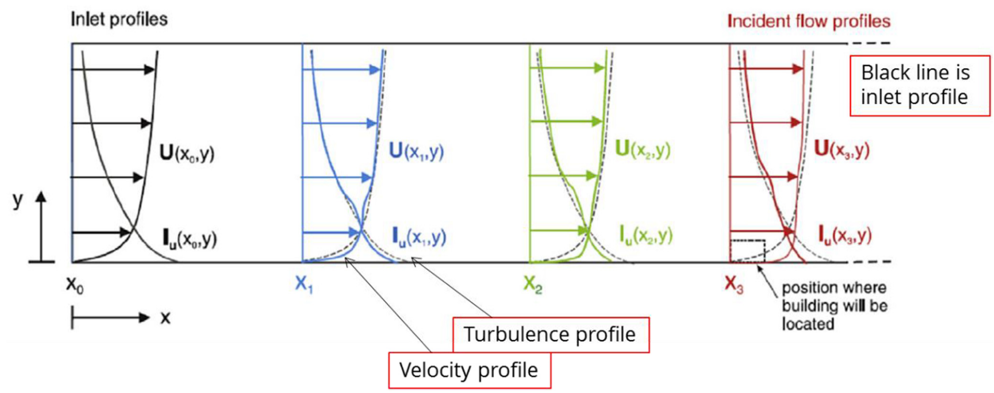

Figure 3. Sketch of How Mean Wind Speed and Turbulence Profile Decays: from Blocken et al. (2007).

a physical wind tunnel. This is a computationally expensive approach since each roughness element needs to be resolved by the computational mesh and can introduce local biases in the flow. Another method is to numerically impose shear and turbulence generation at the ground plane via a wall function but this requires close control of the nearby mesh and does not create turbulent structures, but rather sets the overall shear conditions - hence the mean velocity profile in the flow direction is adjusted but turbulence is not. There are also more exotic methods such as Aboshosha et al.'s (2015) fractal roughness method.

3.3. Third: Prediction of Aerodynamic Flow Around Bluff Bodies (Buildings)

The first two challenges are associated with generating the correct ABL and getting that wind information transported to the building(s) of interest within the CFD domain. The last challenge is to correctly predict the wind flow around the building itself. There is a difference between getting a flow field to look right going around a building versus predicting values with which one can design. The images below of instantaneous velocity are from three different simulations of flow around a cylinder. The first is for a version without inlet turbulence; the second includes inlet turbulence; and the third has the same inlet turbulence with a doubling of the cells. The results clearly show that the prediction is markedly different. This means that the CFD practitioner needs to not only understand what turbulence is, but also ensure that the fluid properties at the building of interest are correct in order to conduct an accurate simulation.

The second example shown below is of flow around a building for ground level pedestrian winds. The simulation setup, grid, geometry and all other features are the same with the only difference being the choice of turbulence model. The three models presented here are standard k-ε closures except for the one on the right where the simulation team adjusted a parameter ($C\mu$) in the RNG model which increases the level of diffusion in locations of shear. This approach is sometimes used in fire simulations to improve the prediction of entrainment into the smoke plume; used here reduces the length of the tail of the high speed jet (red streaks). The revised $C\mu$ value selected was 0.18 whereas the standard is 0.09. Recall that

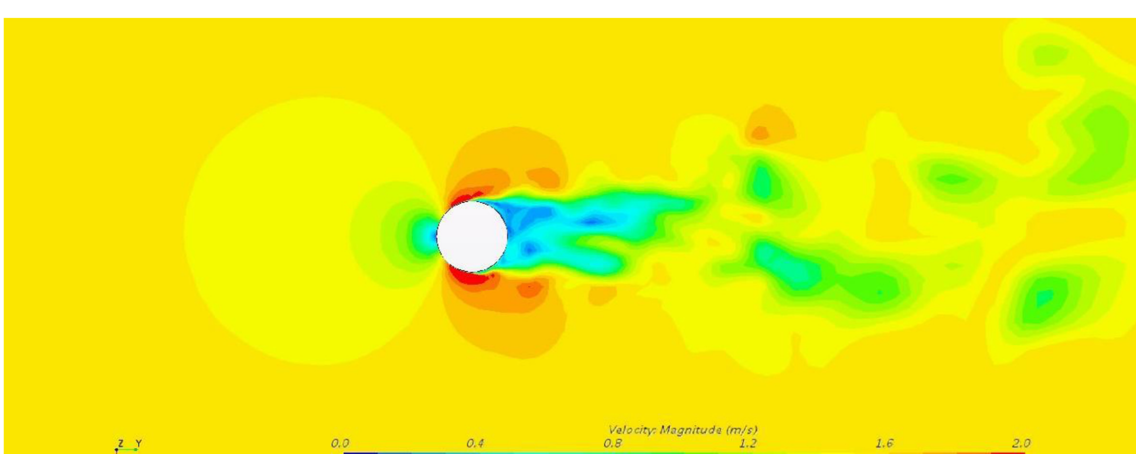

Figure 4. Simulation of Flow Around a Cylinder Without Inlet Turbulence.

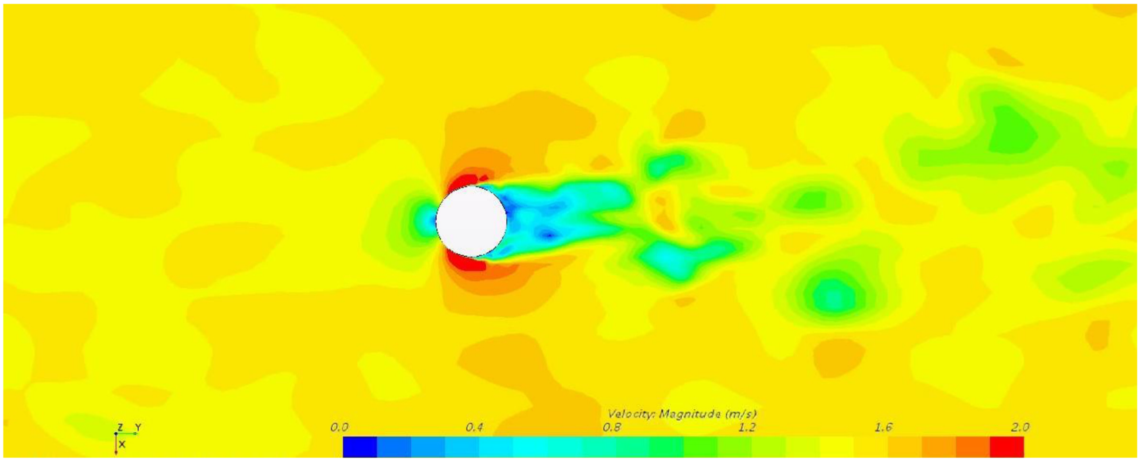

Figure 5. Simulation of Flow Around a Cylinder with Inlet Turbulence.

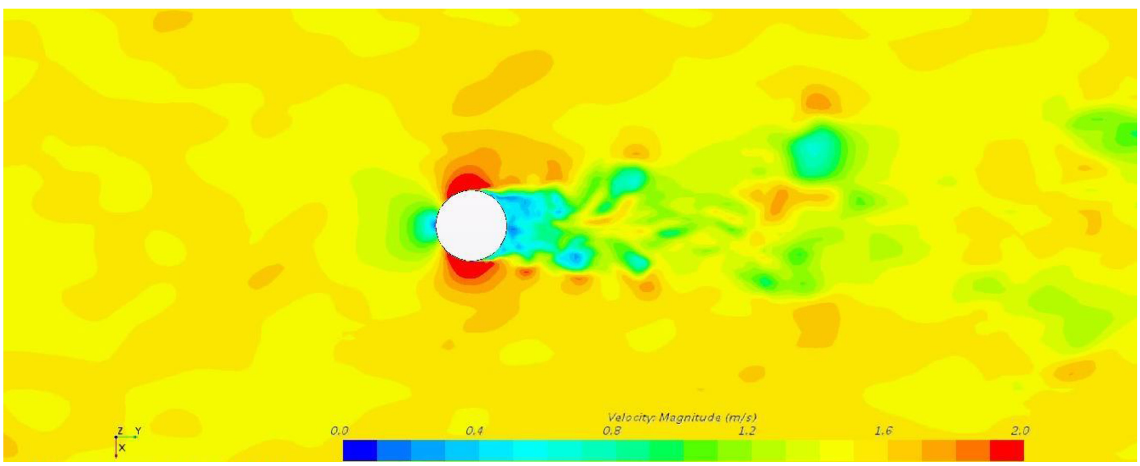

Figure 6. Simulation of Flow Around a Cylinder with Inlet Turbulence and A Doubling of Cells.

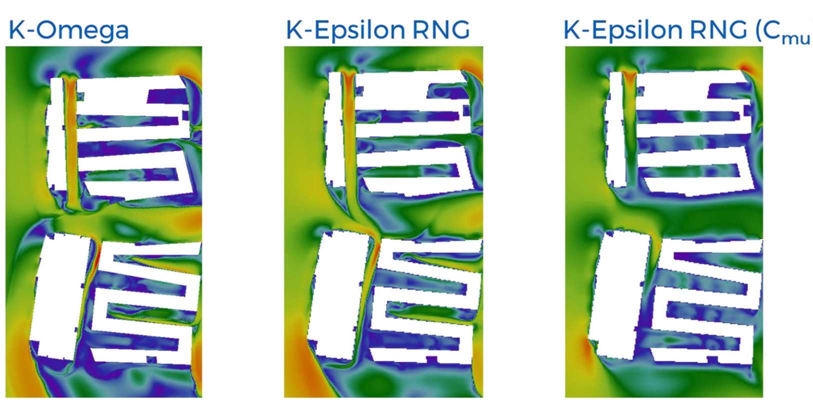

Figure 7. PLW Wind Simulation Results Using RANS for Three Different Turbulence Models.

the discussion earlier noted how the constants in RANS turbulence models were generated by comparing CFD predictions to measured flows. There is at least one commercial software product on the market for building design that has taken this approach of manipulating turbulence models to get a result that "looks better". The authors here do not advocate that one start playing with the constants unless you understand what they do. In this example, selection of the appropriate turbulence model is important.

A final example is one from recent literature. In this example, Capra et al. (2018) compared predictions of wind forces using high resolution LES to those from a wind tunnel. The example used both some standard LES app-

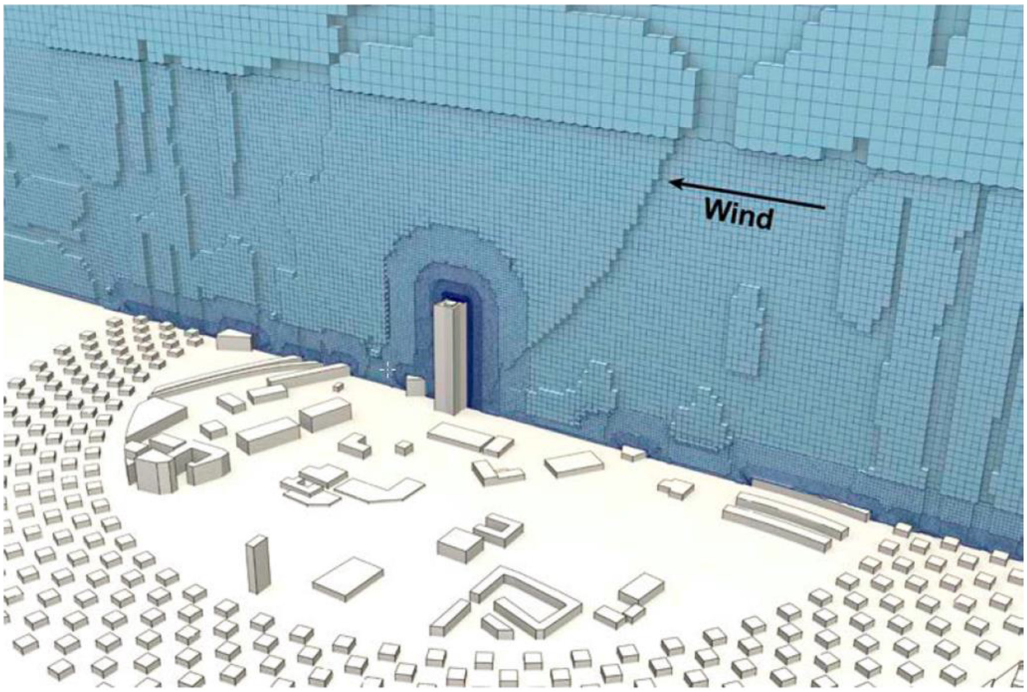

Figure 8. Geometry and Grid for LES Simulation of Wind Loads - from Capra et al. (2018).

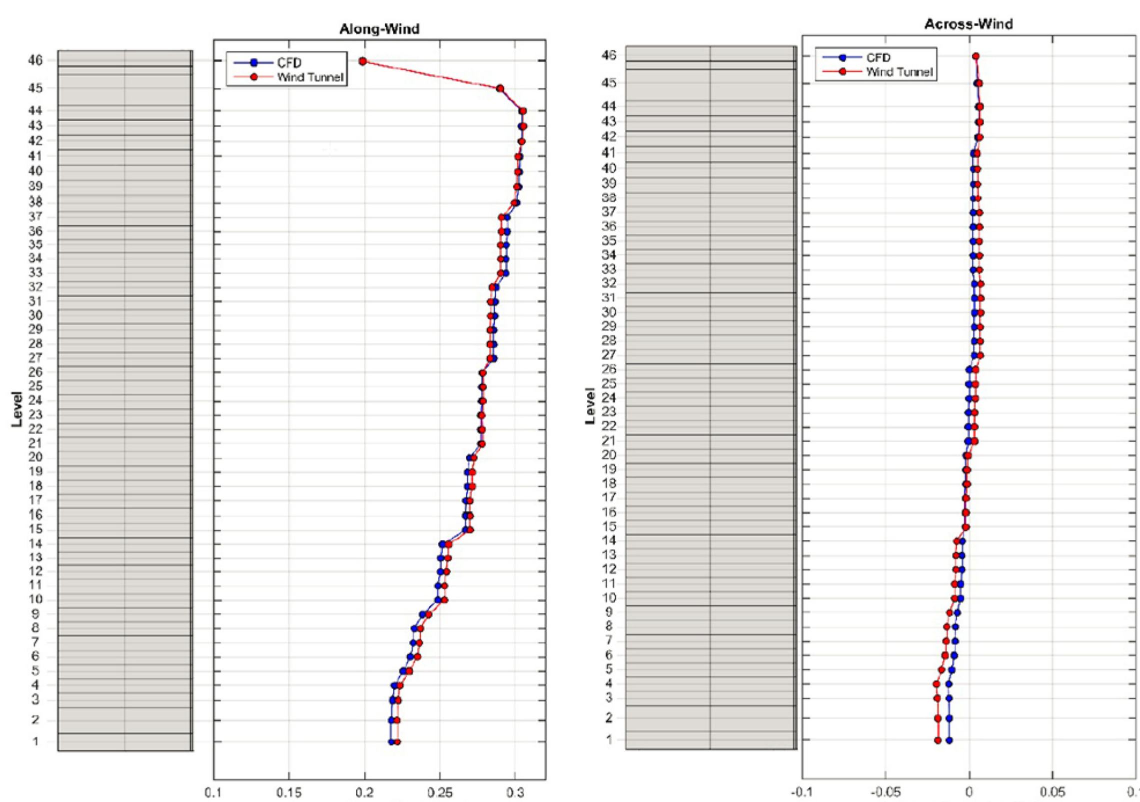

Figure 9. Results Comparing Wind Load Predictions from Wind Tunnel and CFD Simulations - from Capra et al. (2018).

roaches as well as heroic simulations to capture the flow features. Discussion with Capra noted that the cost for a single wind direction ranged from $1,400 to $8,600 USD depending on the grid density. For a cladding or structural wind loading study, one needs to simulate 36 wind directions. Clearly the results are encouraging but the associated costs are significant. Crucial to any comparative study is that CFD simulations be conducted blind (e.g., the CFD user does not know the answer in advance). Only in this way can CFD methodologies be demonstrated to be as robust as wind tunnel testing.

4. Recommendations and Observations

To aid the reader further, we suggest the following:

1. Always select an analysis team with years of experience in wind engineering and fluid dynamics to conduct the studies since wind tunnels and CFD are only tools.
2. Compelling pictures from a computer do not ensure accuracy. Follow-up any CFD simulation or WTT with a pragmatic "is this reasonable" assessment and employ an understanding of building aerodynamics to that assessment.
3. Avoid rationalization of poor results.
4. While the consensus is that well-conducted RANS simulations work adequately for PLW studies, it is not appropriate for wind loading and near field pollutant dispersion studies at this time.
5. The LES methodology currently shows the most promise for wind loading predictions, with LBM and SPH also developing quickly. It should be noted that none of these have demonstrated reliable and repeated accuracy and thus the reason most building codes do not accept CFD results for wind loads.
6. Engineers experienced in both CFD and wind tunnel testing will guide you to the best tool for your project.
7. Understand exactly what is being provided as a product or service. Your simulation or physical test should have some measure of guarantee for accuracy and conformity to building design codes.
8. At some point, international bodies will have to develop standards for CFD simulations as they have for wind tunnel testing.

5. Conclusions

Hopefully the reader has come to the same conclusion as the authors – both wind tunnel testing and CFD have an important role to play in present and future building and infrastructure design. This perspective is shared by people in other industries – e.g., Spalart and Venkatakrishnan (2016), Stump (2018). The challenge for the design community is to understand the strengths and limitations of each and maximize the benefits that each offers for the next project. The other challenge for the design community is to understand the risks of poor simulations. Whether this is because of poor CFD gridding; the wrong choice of turbulence model; a lack of inlet turbulence or lack of expert fluid dynamic experience (remember, CFD is only the tool). This will require that we manage the unbridled enthusiasm to conduct simulations rapidly and ensure that simulations, whether physical in the wind tunnel or virtual using CFD, are conducted properly.

Acknowledgements

The authors would like to acknowledge the contributions of the team at RWDI. We are fortunate to work with a dedicated group of scientists, engineers and technicians. The work of many people is present in this paper and we would miss someone if we tried to name them all. We would like to also acknowledge the thoughtful remarks on our draft manuscript from the reviewer at CTBUH as well as those from Goncalo Pedro and Wayne Boulton at RWDI.

References

Aboshosha, H., Elshaer, A., Bitsuamlak, G.T., and El Damatty, A. (2015). Consistent Inflow Turbulence Generator for LES Evaluation of Wind-Induced Responses for Tall Buildings. *Journal of Wind Engineering and Industrial Aerodynamics*, 142, 198-216.

Aboshosha, H., Bitsuamlak, G., and El Damatty, A. (2015). LES of ABL Flow in the Built-Environment Using Roughness Modeled by Fractal Surfaces. *Sustainable Cities and Society*, 19, 46-60.

Aliabadi, Amir A.,Veriotes, Nikolaos, and Pedro, Gonçalo. (2018). A Very Large-Eddy Simulation (VLES) Model for the Investigation of the Neutral Atmospheric Boundary Layer. *Journal of Wind Engineering & Industrial Aerodynamics*, 183, 152-171.

Baskaran, B. A. (1993). Wind Engineering Studies on Tall Buildings: Transitions in Research. *Buildings and Environment*, 28, 1-19.

Blocken, Bert. (2015). Computational Fluid Dynamics for Urban Physics: Importance, Scales, Possibilities, Limitations and Ten Tips and Tricks Towards Accurate and Reliable Simulations. *Building and Environment*, 91, 219-245.

Blocken, Bert. (2018). LES over RANS in Building Simulation for Outdoor and Indoor Applications: A Foregone Conclusion?, *Building Simulation*, 11, 821-870.

Blocken, B., Carmeliet, J., and Stathopoulos, T. (2007). CFD Evaluation of Wind Speed Conditions in Passages between Parallel Buildings Effect of Wall-Function Roughness Modifications for the Atmospheric Boundary Layer Flow. *Journal of Wind Engineering and Industrial Aerodynamics*, 95, 941-962.

Capra, S., Cammelli, S., Roeder, D., and Knir, J. (2018). *Numerically Simulated Wind Loading on a High-Rise Structure and its Correlation with Experimental Wind Tunnel Testing*, The 7th International Symposium on Computational Wind Engineering 2018, Seoul: July 18-22, 2018.

Damljanović, D. (2012). Gustave Eiffel and the Wind: A Pioneer in Experimental Aerodynamics. *Scientific Technical Review*, 62, 3-13.

Deardorff, J. W. (1970). A Numerical Study of Three-Dimensional Turbulent Channel Flow at Large Reynolds Numbers. *Journal of Fluid Mechanics*, 41, 453-480.

Franke, Jörg, Hellsten, Antti, Schlünzen, Heinke and Carissimo, Bertrand. (2011). The COST 732 Best Practice Guideline for CFD Simulation of Flows in the Urban Environment: A Summary. *International Journal of Environment and Pollution*, 44, 419-427.

Hoerner, S.F. (1965), *Fluid Dynamic Drag*, Published by the Author.

Hosain, Md. Lokman. and Bel Fdhila, Rebei. (2015). *Literature Review of Accelerated CFD Simulation Methods Towards Online Application*, Presented at the 7th International Conference on Applied Energy – ICAE2015, published in: Energy Procedia, 75, 3307-3314.

Jarrin, N., Benhamadouche, S., Laurence, D., and Prosser, R. (2006). A Synthetic-Eddy-Method for Generating Inflow Conditions for Large-Eddy Simulations. *International Journal of Heat and Fluid Flow*, 27, 585-593.

Lateb, M., Meroney, R.N., Yataghene, M., Fellouah H., Saleh, F., and Boufadel, M.C. (2016). On the Use of Numerical Modelling for Near-Field Pollutant Dispersion in Urban Environments - A Review. *Environmental Pollution*, 208, 271-283.

Lund, T. S., Wu, X., and Squires, K. D. (1998). Generation of Turbulent Inflow Data for Spatially-Developing Boundary Layer Simulations. *Journal of Computational Physics*, 140, 233-258.

Luo, Y. et al. (2018), *Comparison of Inflow Generation Methods for LES Simulation of Wind Flow Around a High-Rise Building*, The 7th International Symposium on Computational Wind Engineering 2018, Seoul: July 18-22, 2018.

Murakami, S., Mochoda, A., Ooka, R., Kata, S. and Iizuka, S. (1996). Numerical Predictions of Flow around a Building with Various Turbulence Models: Comparison of k-ε EVM, ASM, DSM and LES with Wind Tunnel Tests. *ASHRAE Transactions*, 102, 741-753.

Nielsen, P. (1973). Berechnung der Luftbewegung in Einem Zwangsbelüfteten Raum. *Gesundheits-Ingenieur*, 94, 299-302.

Neilsen, P. (2015). Fifty years of CFD for Room Air Distribution. *Building and Environment*, 91, 78-90.

Phillips, D., Irwin, P., and Xie, J. (2012). *Design of Sustainable Asian Supertall Buildings for Wind*, Presented at the CTBUH 2012 9th World Congress, Shanghai: September 19-21.

Piomelii, U., Cabot, W. H., Moin, P., and Lee, S. (1991). Subgrid-Scale Backscatter in Turbulent and Transitional Flows. *Physics of Fluids A*, 3.

Reynolds, O. (1895). On the Dynamical Theory of Incompressible Viscous Fluids and the Determination of the Criterion. *Philosophical Transactions of the Royal Society*, 186, 123-164.

Richards, P. J. and Hoxey, R. (1993). Appropriate Boundary Conditions for Computational Wind Engineering Models Using the k-epsilon Turbulence Model. *Journal of Wind Engineering and Industrial Aerodynamics*, 46-47, 145-153.

Shilston, R., Ozkan, E., and Hackett, D. (2018). *Development of the City of London Wind Assessment Guidelines*. The 7th International Symposium on Computational Wind Engineering 2018, Seoul: July 18-22, 2018.

Smagorinsky, J. (1963). General Circulation Experiments with the Primitive Equation I the Basic Experiment. *Monthly Weather Review*, 91, 99-164.

Spalart, P. R. and Venkatakrishnan, V. (2016). On the Role and Challenges of CFD in the Aerospace Industry. *The Aeronautical Journal*, 120, 209-232.

Stump, J. (2018). *Symbiosis: Why CFD and Wind Tunnels Need Each Other*, Published Online – **Aerospace America**, Accessed: 2019/02/24, https://aerospaceamerica.aiaa.org/features/symbiosis-why-cfd-and-wind-tunnels-need-each-other/

Thet Mon Soe and San Yu Khaing. (2017). Comparison of Turbulence Models for Computational Fluid Dynamics Simulation of Wind Flow on Cluster of Buildings in Mandalay. *International Journal of Scientific and Research Publications*, 7, 337-350.

Vasaturo, R. et al. (2018). Large Eddy Simulation of the Neutral Atmospheric Boundary Layer: Performance Evaluation of Three Inflow Methods for Terrains With Different Roughness. *Journal of Wind Engineering & Industrial Aerodynamics*, 173, 241-261.

Yamada, T. and Meroney, R. N. (1971). *Numerical and Wind Tunnel Simulation of Airflow Over an Obstacle*, In Proceedings of the National Conference on Atmospheric Waves, Salt Lake City: 1971.