APPLICATION OF CFD IN BUILDING PERFORMANCE SIMULATION FOR THE OUTDOOR ENVIRONMENT

Bert Blocken[1], Ted Stathopoulos[2], Jan Carmeliet[3,4], Jan Hensen[1]
[1]Building Physics and Systems, Eindhoven University of Technology, The Netherlands
[2]Department of Building, Civil and Environmental Engineering, Concordia University, Montreal, Quebec, Canada
[3]Chair of Building Physics, Swiss Federal Institute of Technology ETHZ, Zürich, Switzerland
[4]Laboratory for Building Technologies, Empa, Dübendorf, Switzerland

ABSTRACT

This paper provides a brief, non-exhaustive overview of the status of the application of CFD in building performance simulation for the outdoor environment. It focuses on four topics: (1) pedestrian wind environment around buildings; (2) wind-driven rain on building facades; (3) convective heat and mass transfer coefficients at building surfaces; and (4) air pollutant dispersion around buildings. For each topic, some specific difficulties, advantages and disadvantages of CFD are addressed.

INTRODUCTION

In the past decades, Computational Fluid Dynamics (CFD) has been studied intensively as a tool for evaluating the indoor environment of buildings and its interaction with the building envelope, as well as for analysing the outdoor environment around buildings. While the use of CFD in engineering practice is becoming quite well established for indoor applications, this is considerably less pronounced for outdoor applications. In complex case studies, wind environmental problems such as pedestrian wind nuisance and air pollutant dispersion are still typically investigated in boundary layer wind tunnels, while wind-driven rain exposure and convective heat and mass transfer coefficients at building surfaces are generally estimated from simplified empirical or semi-empirical formulae. An important disadvantage of wind tunnel measurements however is that usually only point measurements are obtained. Techniques such as Particle Image Velocimetry (PIV) and Laser-Induced Fluorescence (LIF) in principle allow planar or even full 3D data to be obtained, but the cost is considerably higher and application for complicated geometries is hampered by laser-light shielding by the obstructions constituting the urban model. Another disadvantage is the required adherence to similarity criteria in reduced-scale testing. This can be a problem for, e.g., multiphase flow problems and flows in which density differences are an important driving force. Examples are wind-driven rain and pollutant dispersion studies. Empirical and semi-empirical formulae generally only provide a first, crude indication of the relevant parameters, often in averaged form (e.g., surface-averaged) or at a few discrete positions. The information they provide is often too simplified compared to the well-established building performance simulation tools in which this information is used. Examples are wind-driven rain intensities and convective heat and mass transfer coefficients for building envelope Heat-Air-Mass (HAM) transfer tools and Building Energy Simulation (BES) software.

Numerical modelling with CFD could be a powerful alternative because it can avoid some of these limitations. It can provide detailed information on the relevant flow variables in the whole calculation domain ("whole-flow field data"), under well-controlled conditions and without similarity constraints. However, the accuracy of CFD is an important matter of concern. Care is required in the geometrical implementation of the model, in grid generation and in selecting proper solution strategies. In addition, numerical and physical modelling errors need to be assessed by detailed verification and validation studies.

This paper provides a brief, non-exhaustive overview of the status of the application of CFD in building performance simulation for the outdoor environment. It focuses on four topics: (1) pedestrian wind environment around buildings; (2) wind-driven rain on building facades; (3) convective heat and mass transfer coefficients at building surfaces; and (4) air pollutant dispersion around buildings. For each topic, some specific difficulties, advantages and disadvantages of CFD are discussed.

PEDESTRIAN WIND ENVIRONMENT AROUND BUILDINGS

High-rise buildings can introduce high wind speed at pedestrian level, which can lead to uncomfortable or even dangerous conditions (Fig. 1). Wind discomfort and wind danger can be detrimental to the success of new buildings. Wise (1970) reports about shops that are left untenanted because of the windy environment which discouraged shoppers. Lawson and Penwarden (1975) report the death of two old ladies due to an unfortunate fall caused by high wind speed at the base of a tall building. Today, many urban authorities only grant a building permit for a new high-rise building after a wind comfort study has indicated that the negative consequences for the pedestrian wind environment remain limited. Wind comfort studies require knowledge of the mean wind velocity vector field at pedestrian height (z = 1.75 or 2 m). This information can be obtained by wind tunnel modelling or by CFD. Wind tunnel tests are generally point measurements with Laser Doppler Anemometry (LDA) or Hot Wire Anemometry (HWA). In the past, also area techniques such as sand erosion and infrared thermography have been used. They are considered less suitable to obtain accurate quantitative information. Instead, they can be used as part of a two-step approach: first an area technique is used to qualitatively indicate the most important problem locations, followed by accurate point measurements at these most important locations (Blocken and Carmeliet, 2004a).

Figure 1 Pedestrian-level wind nuisance around high-rise buildings.

One of the main advantages of CFD in pedestrian-level wind comfort studies is avoiding this time-consuming two-step approach by providing whole-flow field data. The use of CFD for pedestrian-level wind studies was initiated by the early, more general research efforts in which CFD was applied to study the wind-flow pattern around isolated buildings (Vasilic-Melling, 1977; Hanson et al., 1986; Paterson and Apelt, 1986; 1989; Murakami et al., 1987; Murakami and Mochida, 1988; Baskaran and Stathopoulos, 1989; Murakami, 1990a; 1990b; 1993). Together with later studies, they laid the foundations for future best practice guidelines, by focusing on the importance of the boundary conditions on the numerical results (e.g., Stathopoulos and Baskaran, 1990; Baetke et al., 1990) and by comparing the performance of various types of turbulence models in steady Reynolds-averaged Navier-Stokes (RANS) simulations (Murakami et al., 1992; Murakami, 1993; Mochida et al., 2002), and of RANS versus Large Eddy Simulation (LES) (Murakami et al., 1992; Tominaga et al., 2008a). Note that in steady RANS simulations, only the mean flow is solved, while all scales of turbulence are modelled. In LES on the other hand, the large and generally most important turbulent eddies are solved, while only the eddies smaller than a user-defined filter are modelled. In the past, especially the deficiencies of using the RANS approach with the standard k-ε model in modelling wind flow around buildings were addressed. These include the stagnation point anomaly with overprediction of turbulent kinetic energy near the frontal corner, the underestimation of separation and recirculation regions on the roof and the side faces, and the overestimation of the size of the recirculation region in the wake. Various revised k-ε models and also second-moment closure models were developed and tested, and showed improved performance for several parts of the flow field. However, the main limitation of steady RANS modelling remained, being its incapability to model inherently transient features of the flow field such as separation and recirculation downstream of windward edges and vortex shedding in the wake.

In spite of these deficiencies, steady RANS modelling with the k-ε model and other turbulence models has become quite popular for pedestrian-level wind studies. Two main categories can be distinguished: (1) fundamental studies, which are typically conducted for simple, generic building configurations to obtain insight in the flow behaviour, for parametric studies and for CFD validation; and (2) applied studies, which provide knowledge of the wind environmental conditions in specific and often much more complex case studies. Fundamental studies were performed by many authors, including Baskaran and Stathopoulos (1989), Yoshie et al. (2007), Blocken et al. (2007a, 2008a), Tominaga et al. (2008a) and Mochida and Lun (2008). Apart from these fundamental studies, also several CFD studies of pedestrian wind conditions in complex urban environments have been performed (Murakami, 1990a; Takakura et al., 1993; Stathopoulos and Baskaran, 1996; He and Song, 1999; Richards et al., 2002; Westbury et al., 2002; Blocken et al., 2004; Yoshie et al., 2007; Blocken and Carmeliet, 2008; Blocken and Persoon, 2009) (Fig. 2 and 3). Almost all studies were conducted with the steady RANS approach and a version of the k-ε model. An exception to this is the study by He and Song (1999) who used LES.

The use of CFD in pedestrian-level wind comfort studies in complex urban environments is receiving strong support from several international initiatives that specifically focus on the establishment of guidelines for such simulations (Franke et al., 2004; Franke et al., 2007; Yoshie et al., 2007; Tominaga et al., 2008b). In addition, more general CFD guidelines are available (Casey and Wintergeste, 2000), as well as more specific guidelines for modelling equilibrium atmospheric boundary layers in computational domains (Richards and Hoxey, 1993; Blocken et al., 2007a; 2007b; Hargreaves and Wright, 2007; Franke et al., 2007; Gorlé et al., 2009; Yang et al., 2009).

The increasing acceptance of CFD as a tool for pedestrian-level wind studies has recently been confirmed by the publication of the new Dutch Wind Nuisance Standard, NEN8100 (NEN, 2006; Willemsen and Wisse, 2007) that specifically allows the user to choose between wind tunnel testing and CFD. The standard also demands quality assurance, both for wind tunnel testing and for CFD. It should be noted that CFD verification and validation are the essential components of quality assurance. In case of complex urban environments, when measurements are often not available, model validation should be performed for simpler configurations, the flow features of which show resemblance with those expected in the actual complex urban configuration (Blocken et al., 2004; Franke et al., 2007; Yoshie et al., 2007; Blocken and Carmeliet, 2008; Tominaga et al., 2008b). For these simpler cases, wind tunnel measurement data are generally available in the literature. Note that steady RANS is the commonly used method, while LES is still considered out of reach for pedestrian-level wind studies in urban environments (Yoshie et al., 2007).

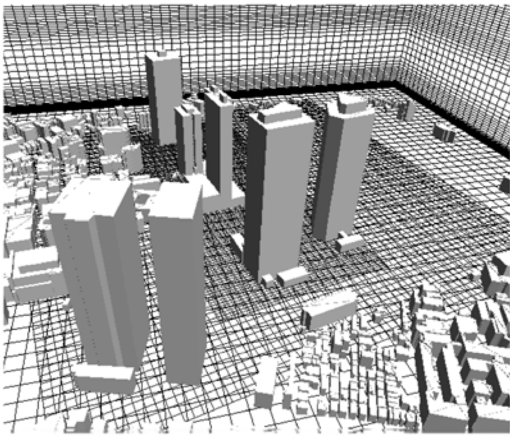

Figure 2 Part of geometry and computational grid for CFD study of pedestrian-level wind conditions in the Shinjuku Sub-central area in Tokyo, Japan (Yoshie et al., 2007).

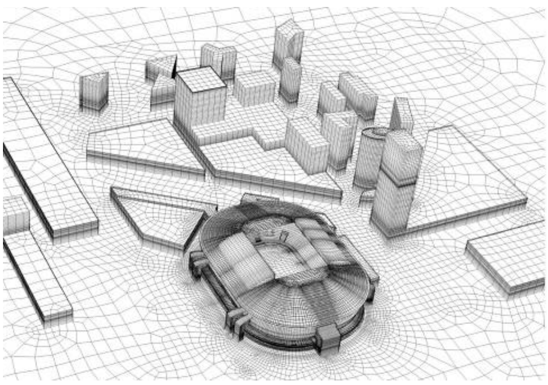

Figure 3 Part of geometry and computational grid for CFD study of pedestrian-level wind conditions around the Amsterdam "ArenA" football stadium in the Netherlands (Blocken and Persoon, 2009).

WIND-DRIVEN RAIN ON BUILDINGS

To the knowledge of the authors, the first CFD simulations of wind-driven rain (WDR) on buildings were made by Souster in 1979. For a full historical overview up to 2003, the reader is referred to (Blocken and Carmeliet, 2004b). Choi (1991; 1993; 1994a; 1994b) developed and applied a steady-state simulation technique for WDR. It allows determining the spatial distribution of WDR on building facades for given (fixed) values of the wind speed, the wind direction and the horizontal rainfall intensity. Later, Choi's simulation technique was extended into the time domain by Blocken and Carmeliet (2002; 2007a). Most of these CFD simulations were based on the RANS approach. Although validation is an essential part of RANS CFD simulations, up to now, only a few attempts have been made for CFD validation with full-scale WDR measurements (van Mook, 2002; Blocken and Carmeliet, 2002; 2006, 2007b; Tang and Davidson, 2004; Abuku et al., 2009; Briggen et al., 2009). While some authors found significant discrepancies between simulations and measurements, others indicated a fair to good agreement. A probable reason for these discrepancies is the role of turbulent dispersion of raindrops, which can be very different depending on the building geometry and the position on the building (Briggen et al., 2009). In spite of quite some research efforts, the application of CFD WDR studies in practice has remained very limited. Tang and Davidson (2004) used CFD WDR simulations to explain surface soiling patterns on the Cathedral of Learning in Pittsburg (Fig. 4). In this figure, the catch ratio is the ratio of WDR intensity to the standard rainfall intensity on the ground. Briggen et al. (2009) employed CFD simulations of WDR to explain the moisture-related damage on the monumental tower building St. Hubertus in the Netherlands (Fig. 5). Only a few authors provided guidelines for CFD WDR simulation (Choi, 1994a; 1994b; Blocken and Carmeliet, 2002; 2004b; 2006; Briggen et al., 2009). Note however that the guidelines for pedestrian-level wind studies, mentioned in the previous section, also apply for CFD WDR studies.

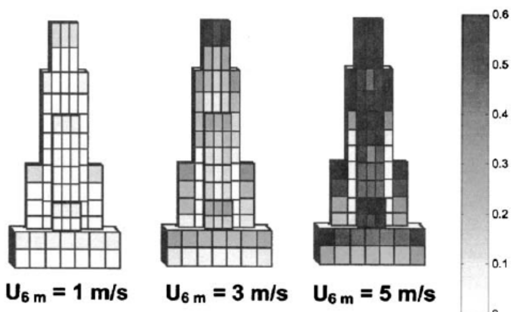

Figure 4 Illustration of the catch ratio on the Cathedral of Learning in Pittsburg (Tang and Davidson, 2004).

There are two main reasons for the limited practical use of CFD for WDR studies: (1) the excessively time-consuming character of Lagrangian particle tracking of raindrops, in which the entire building facade needs to be covered by a large number of raindrops. Lagrangian tracking implies solving the equation of motion of individual raindrops within the wind-flow field. Note that this wind-flow field is generally obtained with an Eulerian approach, i.e. not focusing on individual particles but on fixed positions in space. Lagrangian tracking needs to be performed for a large number of combinations of reference wind speed, wind direction and raindrop diameter. (2) The fact that steady RANS does not allow accurate modelling of the turbulent dispersion of raindrops, which is important for calculating WDR intensities at the lower part of high-rise building facades (Briggen et al., 2009). Accurate turbulent dispersion modelling would require transient simulations with LES, which would require even more intensive Lagrangian particle tracking efforts. To alleviate these problems, it might be necessary to abandon the traditional “Eulerian-Lagrangian” framework in CFD WDR simulations, and to resort to “Eulerian-Eulerian” modelling instead, in which not only wind-flow pattern, but also the WDR intensities are computed with an Eulerian approach.

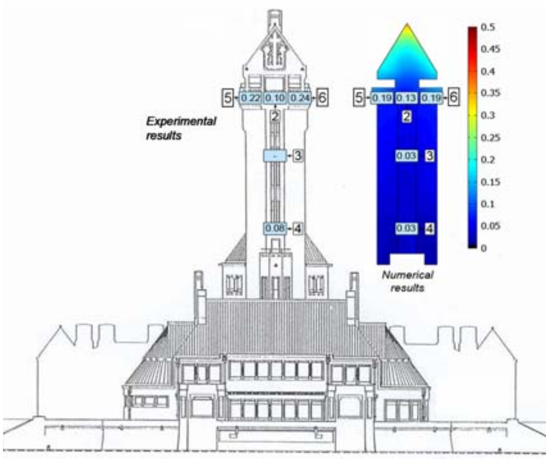

Figure 5 Comparison of measured and simulated catch ratio on the south-west facade of the tower of the monumental building St. Hubertus in the Netherlands (Briggen et al., 2009).

CONVECTIVE HEAT AND MASS TRANSFER COEFFICIENTS AT EXTERIOR BUILDING SURFACES

In the past, convective heat and mass transfer coefficients (CHTC and CMTC) for exterior building surfaces have been investigated using wind tunnel measurements (e.g., Kelnhofer and Thomas, 1976), full-scale measurements (e.g., Sharples et al., 1984; Loveday and Taki, 1996; Liu and Harris, 2007) and CFD simulations (Emmel et al., 2007; Defraeye et al., 2009, Blocken et al., 2009). A recent overview of wind convection coefficient correlations has been provided by Palyvos (2008). While a large number of valuable experimental investigations have been conducted, the number of CFD analyses for exterior CHTC and CMTC for buildings is very small. This might seem strange given the very large number of such CFD studies that have been conducted in other disciplines, such as mechanical and electronic engineering. The main reason for this are the extreme requirements for these simulations for building applications. As opposed to mechanical and electronic engineering applications, the Reynolds numbers in civil and building engineering are several orders of magnitude larger (10^5-10^7). The higher the Reynolds number, the lower the thickness of the viscous sublayer and buffer layer that determine the convective surface resistance. For building applications, the thickness of the viscous sublayer can go down to 10 mm – 100 μm (Blocken et al. 2009). Accurate CFD modelling of convective heat and mass transfer requires accurate and detailed modelling of each part of the boundary layer. Unfortunately, previous simulations (Emmel et al. 2007) were conducted using wall functions, in which the detailed influence of the boundary layer is strongly simplified. It has been shown that using wall functions can easily yield overestimations of the CHTC by up to 60% (Blocken et al., 2009). Accurate modelling of convective heat and mass transfer requires at least a few cells in the viscous sublayer. As a result, very high-resolution grids are required, with large differences between the largest (up to 1 km) and smallest (down to 10 μm) length scales. Such high grid resolution gradients and very small cells slow down convergence, can inhibit convergence to be obtained with less-diffusive turbulence models such as second-moment closure models, and can even cause computer round-off errors to become important. To the knowledge of the authors, such simulations have up to now only been performed by Defraeye et al. (2009) and Blocken et al. (2009), but these authors did not go further than considering a simple cubic building model (Fig. 6).

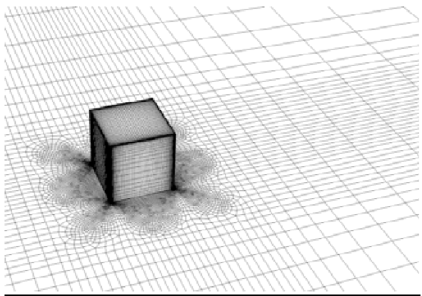

Figure 6 High-resolution grid for CFD simulation of convective heat transfer coefficients around an isolated cubic building model, with first grid cell point at 160 μm from the facade (Blocken et al., 2009).

This type of high-resolution CFD simulations is not practical for actual cases of building simulation, and will probably remain so until specially-adapted wall functions are developed that can accurately take into account surface heat and mass transfer. However, note that also in this case, still grids with a relatively high resolution will be needed. Therefore, accurate exterior building heat and mass transfer with CFD will remain very demanding and might remain out of range for many practical building simulation efforts for a considerable time to come.

AIR POLLUTANT CONCENTRATION DISTRIBUTIONS AROUND BUILDINGS

In the past two decades, a large number of so-called "microscale" CFD simulations to investigate air pollutant concentration distributions around buildings have been conducted. Micro-scale refers to simulations at horizontal length scales smaller than – typically – 5 km. Due to the complexity of microscale pollutant dispersion around buildings, much of the research in wind tunnels and with CFD has focused on two generic basic situations: the urban street canyon (e.g. Leitl et al., 1997; Kastner-Klein and Plate, 1999) and the isolated building (e.g. Li and Meroney, 1983; Leitl et al., 1997; Selvam, 1997; Tominaga et al., 1997; Li and Stathopoulos, 1997; Meroney et al., 1999; Tominaga and Stathopoulos, 2007a; 2007b; 2008; Blocken et al., 2008b). These generic studies have proven to be very suitable for validation, verification and sensitivity analysis. The reason is that, even although both situations are strong simplifications of reality, the flow and dispersion processes involved are very complex and contain most of the salient features that are also present in the complex urban environment. Apart from these generic studies, several authors have performed studies in complex urban environments (e.g., Hanna et al., 2006; Patnaik et al., 2007; Neofytou et al., 2008; Huang et al., 2008; Löhner et al., 2008), although extensive validation studies for such cases have not yet been reported.

A general conclusion from the generic studies is that steady RANS simulations in combination with typical turbulent Schmidt numbers (i.e. the ratio of turbulent viscosity to turbulent mass diffusivity) of 0.7-0.9 provide too low lateral turbulent diffusion compared to wind tunnel testing. In the past, several authors have attributed this to the fact that steady RANS modelling cannot incorporate the inherently transient behaviour of separation and recirculation downstream of windward edges, and of von Karman vortex shedding in the wake, which are particulary important for pollutant dispersion (Leitl et al., 1997; Meroney et al., 1999; Blocken et al., 2008b). In fact, Tominaga and Stathopoulos (2007a; 2008) showed that LES modelling, which takes ino account these transient features because it actually solves the large eddies in the flow, can strongly improve pollutant concentration predictions. This seems to indicate that LES modelling is a requirement for pollutant dispersion modelling, also in complex urban environments. However, the computational demands associated with this are very large. Although the statement by Yoshie et al. (2007) that LES is still out of reach for practical pedestrian-level wind studies may equally apply to pollutant dispersion modelling, valuable attempts with LES modelling in complex urban environments have been performed, supported by efficient grid generation techniques and parallel computing facilities (Hanna et al., 2006; Patnaik et al., 2007; Löhner et al., 2008).

CONCLUSIONS

A brief, non-exhaustive overview of the status of application of CFD in building performance simulation for the outdoor environment has been provided for four topics. While CFD offers some considerable advantages compared to wind tunnel testing and simplified empirical or semi-empirical equations, its practical application in at least wind-driven rain, convective heat and mass transfer and pollutant dispersion studies is currently hampered by several important limitations. Some main limitations are the deficiencies of steady RANS modelling, the time-consuming and significantly more complex character of LES, the need for high-resolution grids and the constant requirement of CFD validation and verification. Research efforts will continue to focus on alleviating these limitations, but at least equally important is avoiding user errors by increased high-quality education in CFD and its application for outdoor building performance simulation.

ACKNOWLEDGEMENTS

The authors thank the anonymous reviewers of the abstract who provided valuable suggestions for this paper.

REFERENCES

Abuku, M., Blocken, B., Nore, K., Thue, J.V., Carmeliet, J., Roels, S. 2009. On the validity of numerical wind-driven rain simulation on a rectangular low-rise building under various oblique winds. Build. Environ. 44(3): 621– 632.

Baetke, F., Werner, H., Wengle, H. 1990. Numerical simulation of turbulent flow over surface-mounted obstacles with sharp edges and corners. J. Wind Eng. Ind. Aerodyn.. 35(1-3): 129-147.

Baskaran, A., Stathopoulos, T. 1989. Computational evaluation of wind effects on buildings. Build. Environ. 24: 325–333.

Blocken, B., Carmeliet, J. 2002. Spatial and temporal distribution of driving rain on a low-rise building. Wind Struct. 5(5): 441-462.

Blocken B, Carmeliet J. 2004a. Pedestrian wind environment around buildings: Literature review and practical examples. Journal of Thermal Envelope and Building Science 28(2): 107-159.

Blocken, B., Carmeliet, J. 2004a. Pedestrian wind environment around buildings: Literature review and practical examples. J. Thermal Env. Build. Sci. 28(2): 107-159.

Blocken, B., Carmeliet, J. 2004b. A review of wind-driven rain research in building science. J. Wind Eng. Ind. Aerodyn. 92(13): 1079-1130.

Blocken, B., Roels, S., Carmeliet, J. 2004. Modification of pedestrian wind comfort in the Silvertop Tower passages by an automatic control system. J. Wind Eng. Ind. Aerodyn. 92(10): 849-873.

Blocken, B., Carmeliet, J. 2006. The influence of the wind-blocking effect by a building on its wind-driven rain exposure. J. Wind Eng. Ind. Aerodyn. 94(2): 101-127

Blocken, B., Carmeliet, J., Stathopoulos, T. 2007a. CFD evaluation of wind speed conditions in passages between parallel buildings—effect of wall-function roughness modifications for the atmospheric boundary layer flow. J. Wind Eng. Ind. Aerodyn. 95(9-11) 941-962.

Blocken, B., Stathopoulos, T., Carmeliet, J. 2007b. CFD simulation of the atmospheric boundary layer: wall function problems. Atmos. Environ. 41(2) 238-252.

Blocken B, Carmeliet J. 2007a. On the errors associated with the use of hourly data in wind-driven rain calculations on building facades. Atmos. Environ. 41(11): 2335-2343

Blocken B, Carmeliet J. 2007b. Validation of CFD simulations of wind-driven rain on a low-rise building facade. Build. Environ. 42(7): 2530–2548.

Blocken B, Moonen P, Stathopoulos T, Carmeliet J. 2008a. A numerical study on the existence of the Venturi-effect in passages between perpendicular buildings. Journal of Engineering Mechanics – ASCE 134(12): 1021-1028.

Blocken B, Stathopoulos T, Saathoff P, Wang X. 2008b. Numerical evaluation of pollutant dispersion in the built environment: comparisons between models and experiments. J. Wind Eng. Ind. Aerodyn. 96(10-11): 1817-1831.

Blocken B, Carmeliet J. 2008. Pedestrian wind conditions at outdoor platforms in a high-rise apartment building: generic sub-configuration validation, wind comfort assessment and uncertainty issues. Wind Struct. 11(1): 51-70

Blocken, B., Persoon, J., 2009. Pedestrian wind comfort around a large football stadium in an urban environment: CFD simulation, validation and application of the new Dutch wind nuisance standard. Submitted.

Blocken, B., Defraeye, T., Derome, D., Carmeliet, J. 2009. High-resolution CFD simulations for convective heat transfer coefficients at the facade of a low-rise building. Build. Environ. In press. (doi:10.1016/j.buildenv.2009.04.004)

Briggen, P.M., Blocken, B., Schellen, H.L. 2009. Wind-driven rain on the facade of a monumental tower: numerical simulation, full-scale validation and sensitivity analysis. Build. Environ. 44(8): 1675-1690.

Casey, M., Wintergerste, T. 2000. Best Practice Guidelines, ERCOFTAC Special Interest Group on Quality and Trust in Industrial CFD, ERCOFTAC, Brussels.

Choi, E.C.C. 1991. Numerical simulation of wind-driven rain falling onto a 2-D building, in: Asia Pacific Conf. on Computational Mechanics, Hong Kong, 1721-1728.

Choi, E.C.C. 1993. Simulation of wind-driven rain around a building. J. Wind Eng. Ind. Aerodyn. 46&47: 721-729.

Choi, E.C.C. 1994a. Determination of wind-driven rain intensity on building faces. J. Wind Eng. Ind. Aerodyn. 51: 55-69.

Choi, E.C.C. 1994b. Parameters affecting the intensity of wind-driven rain on the front face of a building. J. Wind Eng. Ind. Aerodyn. 53: 1-17.

Defraeye, T., Blocken, B., Carmeliet, J. 2009. CFD analysis of convective heat transfer at the surfaces of a cube in a deep turbulent boundary layer. Submitted.

Emmel, M.G., Abadie, M.O., Mendes, N. 2007. New external convective heat transfer coefficient correlations for isolated low-rise buildings. Energ. Buildings 39(3): 335-342.

Franke, J., Hirsch, C., Jensen, A.G., Krüs, H.W., Schatzmann, M., Westbury, P.S., Miles, S.D., Wisse, J.A., Wright, N.G., 2004. Recommendations on the use of CFD in wind engineering. In: van Beeck, J.P.A.J. (Ed.), Proceedings of the International Conference on Urban Wind Engineering and Building Aerodynamics. COST Action C14, Impact of Wind and Storm on City Life Built Environment. Von Karman Institute, Sint-Genesius-Rode, Belgium, 5–7 May 2004.

Franke, J., Hellsten, A., Schlünzen, H., Carissimo, B. (Eds.), 2007. Best practice guideline for the CFD simulation of flows in the urban environment. COST Office Brussels, ISBN 3-00-018312-4.

Gorlé, C., van Beeck, J., Rambaud, P., Van Tendeloo, G., 2009. CFD modelling of small particle dispersion: the influence of the turbulence kinetic energy in the atmospheric boundary layer. Atmos. Environ. 43(3): 673-681.

Hanna, S.R., Brown, M.J., Camell, F.E., Chan, S.T., Coirier, W.J., Hansen, O.R., Huber, A.H., Kim, S., Reynolds, R.M. 2006. Detailed simulations of atmospheric flow and dispersion in downtown Manhattan: An application of five computational fluid dynamics models. Bulletin of the American Meteorological Society 87(12): 1713.

Hanson, T. Summers D.M., Wilson, C.B. 1986. Validation of a computer simulation of wind flow over a building model. Build. Environ. 21: 97–111.

Hargreaves, D.M., Wright, N.G. 2007. On the use of the k–ε model in commercial CFD software to model the neutral atmospheric boundary layer. J. Wind Eng. Ind. Aerodyn. 95(5), 355-369.

He, J., Song, C.C.S., 1999. Evaluation of pedestrian winds in urban area by numerical approach. J. Wind Eng. Ind. Aerodyn. 81: 295–309.

Huang, H., Ooka, R., Chen, H., Kato, S., Takahashi, T., Watanabe, T., 2008. CFD analysis on traffic-induced air pollutant dispersion under non-isothermal condition in a complex urban area in winter. J. Wind Eng. Ind. Aerodyn. 96(10-11): 1774-1788.

Kastner-Klein, P., Plate, E.J. 1999. Wind-tunnel study of concentration fields in street canyons. Atmos. Environ. 33(24-25): 3973-3979.

Kelnhofer, W.J., Thomas C.J. 1976. External convection heat transfer coefficients on a building model. ASME Paper 76-WA/FE-30, ASME Winter Annual Meeting, New York.

Lawson, T.V., Penwarden, A.D. 1975. The effects of wind on people in the vicinity of buildings. 4th Int. Conf. Wind Effects on Buildings and Structures, Heathrow.

Leitl, B.M., Kastner-Klein, P., Rau, M., Meroney, R.N. 1997. Concentration and flow distributions in the vicinity of U-shaped buildings: wind-tunnel and computational data. J. Wind Eng. Ind. Aerodyn. 67&68: 745-755.

Li, W., Meroney, R.N. 1983. Gas dispersion near a cubical model building. Part I. Mean concentration measurements. J. Wind Eng. Ind. Aerodyn. 12: 15-33.

Li, Y., Stathopoulos, T. 1997. Numerical evaluation of wind-induced dispersion of pollutants around a building. J.Wind Eng. Ind. Aerodyn. 67&68: 757-766.

Liu Y, Harris DJ. 2007. Full-scale measurements of convective coefficient on external surface of a low-rise building in sheltered conditions. Build. Environ. 42(7): 2718-2736.

Löhner, R., Cebral, J.R., Camelli, F.E., Appanaboyina, S., Baum, J.D., Mestreau, E.L., Soto, O.A., 2008. Adaptive embedded and immersed unstructured grid techniques. Comput. Meth. Appl. Mech. Eng. 197(25-28): 2173-2197.

Loveday DL, Taki AH. 1996. Convective heat transfer coefficients at a plane surface on a full-scale building facade. Int. J. Heat Mass Transf. 39(8): 1729-1742.

Meroney, R.N., Leitl, B.M., Rafailidis, S. and Schatzmann, M. 1999. Wind-tunnel and numerical modeling of flow and dispersion about several building shapes. J. Wind Eng. Ind. Aerodyn. 81: 333-345.

Mochida, A.., Tominaga, Y., Murakami, S., Yoshie, R., Ishihara, T., Ooka, R. 2002. Comparison of various k–ε models and DSM applied to flow around a high-rise building—report on AIJ cooperative project for CFD prediction of wind environment. Wind Struct. 5(2–4): 227–244.

Mochida, A., Lun, I.Y.F. 2008. Prediction of wind environment and thermal comfort at pedestrian level in urban area. J. Wind Eng. Ind. Aerodyn. 96(10-11): 1498-1527.

Murakami, S., Mochida; A., Hibi, K. 1987. Three-dimensional numerical simulation of airflow around a cubic model by means of large eddy simulation. J. Wind Eng. Ind. Aerodyn. 25: 291–305.

Murakami, S., Mochida, A. 1988. 3-D numerical simulation of airflow around a cubic model by means of the k-ε model. J. Wind Eng. Ind. Aerodyn. 31: 283-303.

Murakami, S. 1990a. Computational wind engineering. J. Wind Eng. Ind. Aerodyn. 35(1-3): ix-xi.

Murakami, S. 1990b. Numerical simulation of turbulent flowfield around cubic model: current status and applications of k-e model and LES. J. Wind Eng. Ind. Aerodyn. 33: 139-152.

Murakami, S., Mochida, A., Hayashi, Y., Sakamoto, S. 1992. Numerical study on velocity-pressure field and wind forces for bluff bodies by k-ε, ASM and LES. J. Wind Eng. Ind. Aerodyn. 44(1-3): 2841-2852

Murakami, S. 1993. Comparison of various turbulence models applied to a bluff body. J. Wind Eng. Ind. Aerodyn. 46 & 47: 21-36.

NEN. 2006. Wind comfort and wind danger in the built environment, NEN 8100 (in Dutch) Dutch Standard.

Neofytou, P., Haakana, M., Venetsanos, A., Kousa, A., Bartzis, J., Kukkonen, J., 2008. Computational fluid dynamics modelling of the pollution dispersion and comparison with measurements in a street canyon in Helsinki. Environ. Monit. Assess. 13(3): 439-448.

Palyvos, J.A. 2008. A survey of wind convection coefficient correlations for building envelope energy systems' modelling. Appl. Therm. Eng. 28(8-9): 801-808.

Paterson D.A., Apelt C.J. 1986. Computation of wind flows over three-dimensional buildings. J. Wind Eng. Ind. Aerodyn. 24: 192–213.

Paterson, D.A., Apelt, C.J. 1989. Simulation of wind flow around three-dimensional buildings. Build. Environ. 24: 39–50.

Patnaik, G., Boris, J.P., Young, T.R., Grinstein, F.F., 2007. Large scale urban contaminant transport simulations with MILES. Journal Fluids Eng. 129(12) 1524-1532.

Richards, P.J., Hoxey, R.P. 1993. Appropriate boundary conditions for computational wind engineering models using the k-ε turbulence model. J. Wind Eng. Ind. Aerodyn. 46&47: 145-153.

Richards, P.J., Mallison, G.D., McMillan, D., Li, Y.F. 2002. Pedestrian level wind speeds in downtown Auckland. Wind Struct. 5(2-4): 151-164.

Selvam, R.P. 1997. Numerical simulation of pollutant dispersion around a building using FEM. J. Wind Eng. Ind. Aerodyn. 67&68: 805-814.

Sharples S. 1984. Full scale measurements of convective energy losses from exterior building surfaces. Build. Environ.19: 31-39.

Souster, C. 1979. A theoretical approach to predicting snow loads and driving rain deposition on buildings, Ph.D. Thesis, University of Sheffield, UK, 1979.

Stathopoulos, T., Baskaran, A. 1990. Boundary treatment for the computation of 3D turbulent conditions around buildings. J. Wind Engng Ind. Aerodyn. 35: 177–200.

Stathopoulos, T., Baskaran, A. 1996. Computer simulation of wind environmental conditions around buildings. Eng. Struct. 18 (11): 876–885.

Takakura, S., Suyama, Y., Aoyama, M. 1993. Numerical simulation of flowfield around buildings in an urban area. J. Wind Eng. Ind. Aerodyn. 46-47: 765-771.

Tang, W., Davidson, C.I. 2004. Erosion of limestone building surfaces caused by wind-driven rain. 2. Numerical modelling. Atmos. Environ. 38(33): 5601-5609.

Tominaga, Y., Stathopoulos, T. 2007a. Numerical simulation of dispersion around an isolated cubic building - influence of turbulence models and turbulent Schmidt number. Proc. 12th Int. Conf. Wind Engineering, Cairns, Australia.

Tominaga, Y., Stathopoulos, T. 2007b. Turbulent Schmidt numbers for CFD analysis with various types of flowfield. Atmos. Environ. 41(37): 8091-8099.

Tominaga, Y., Murakami, S., Mochida, A. 1997. CFD prediction of gaseous diffusion around a cubic model using a dynamic mixed SGS model based on composite grid technique. J. Wind Eng. Ind. Aerodyn. 67-68: 827-841.

Tominaga, Y., Mochida, A., Murakami, SH, Sawaki, S. 2008a. Comparison of various revised k–ε models and LES applied to flow around a high-rise building model with 1:1:2 shape placed within the surface boundary layer. J. Wind Eng. Ind. Aerodyn. 96(4): 389-411.

Tominaga, Y., Mochida, A., Yoshie, R., Kataoka, H., Nozu, T., Yoshikawa, M., Shirasawa, T. 2008b. AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings. J. Wind Eng. Ind. Aerodyn. 96(10-11) 1749-1761.

Tominaga, Y., Stathopoulos, T. 2008. Numerical simulation of plume dispersion around an isolated cubic buildings: comparisons between RANS and LES computations. BBAA VI International Colloquium on: Bluff Bodies Aerodynamics & Applications, Milano, Italy, July, 20-24, 2008.

van Mook, F.J.R., 2002. Driving rain on building envelopes, Ph.D. thesis, Building Physics and Systems, Technische Universiteit Eindhoven, Eindhoven University Press, Eindhoven, The Netherlands, 198 p.

Vasilic-Melling, D., 1977. Three Dimensional Turbulent Flow Past Rectangular Bluff Bodies (3rd edition), Ph.D. Thesis, Imperial College of Science and Technology, London.

Westbury, P.S., Miles, S.D., Stathopoulos, T. 2002. CFD application on the evaluation of pedestrian-level winds. Workshop on Impact of Wind and Storm on City Life and Built Environment, Cost Action C14, CSTB, June 3–4, Nantes, France.

Willemsen, E., Wisse, J.A., 2007. Design for wind comfort in The Netherlands: Procedures, criteria and open research issues. J. Wind Eng. Ind. Aerodyn., 95(9-11): 1541-1550.

Wise, A.F.E. 1970. Wind effects due to groups of buildings. Royal Society Symposium Architectural Aerodynamics, London.

Yang, Y., Gu, M., Chen, S., Jin, X., 2009. New inflow boundary conditions for modelling the neutral equilibrium atmospheric boundary layer in computational wind engineering. J. Wind Eng. Ind. Aerodyn. 97(2): 88-95.

Yoshie, R., Mochida, A., Tominaga, Y., Kataoka, H., Harimoto, K., Nozu, T., Shirasawa, T. 2007. Cooperative project for CFD prediction of pedestrian wind environment in the Architectural Institute of Japan. J. Wind Eng. Ind. Aerodyn. 95 (9-11): 1551-1578.