

# RECOMMENDATIONS ON THE USE OF CFD IN WIND ENGINEERING

J. Franke (franke@ift.mb.uni-siegen.de)

*University of Siegen, Dept. of Fluid- and Thermodynamics, Siegen, Germany*

C. Hirsch

*Vrije Universiteit Brussel, Brussels, Belgium*

A.G. Jensen

*FORCE Technology – DMI, Lyngby, Denmark*

H.W. Krüs

*Cyclone Fluid Dynamics BV, Waalre, The Netherlands*

M. Schatzmann

*University of Hamburg, Meteorological Institute, Hamburg, Germany*

P.S. Westbury\* & S.D. Miles

*Building Research Establishment, U.K. (\* now: University College, London, U.K.)*

J.A. Wisse

*Eindhoven University of Technology, Eindhoven, The Netherlands*

N.G. Wright

*University of Nottingham, School of Civil Engineering, Nottingham, U.K.*

ABSTRACT: Computational fluid dynamics (CFD) enjoys a widespread use in the wind engineering community. Applications are increasing despite the fact that many parameters influencing the simulation results are not yet thoroughly understood, even in the “simple” case of incompressible turbulent flow. When further physical processes are studied, like pollutant dispersion or wind driven rain or snow, the situation is even worse. Nevertheless a lot can be learned from the published simulation results, concerning best practice in the area of wind engineering. This paper summarizes the results from published simulations and tries to distil recommendations for the use of CFD in wind engineering tasks, focussing on the statistically steady simulation of pedestrian wind in built urban areas.

## ***1 INTRODUCTION***

The application of CFD in wind engineering, called computational wind engineering (CWE), has significantly increased in the last two decades. Despite its widespread use, the general appraisal of the approach for quantitative and sometimes even qualitative predictions is expressed as lack of confidence, the main objection being the availability of many physical and numerical parameters in the approach, which can be freely chosen by the user [1,2,3,4]. There are basically two types of parameters that act as sources of error in CFD results. First there are modelling errors that arise from the turbulence models used and the physical boundary conditions applied. The other errors stem from the numerical approximations. Here the grid design, the truncation error of the discretisation scheme and the error from incomplete iterative convergence influence the solution. Several comparative studies have been conducted in the last years to assess the influence of different parameters on the solution. While many lessons have been learnt from these and other studies, no generally accepted recommendations have yet been put together to increase the confidence in CWE. But there are several initiatives to establish best practice guidelines in that field. For industrial CFD in general the ERCOFTAC Best Practice Guidelines exist [5] and provide valuable information on the general topics of CFD also relevant for CWE. But special topics for CWE have been deliberately left out. There is also a draft of a guideline from the VDI on prognostic microscale windfield models for flow around buildings and obstacles [6]. It mostly concentrates on validation with few general guidelines for the setup of the numerical model. Activities on best practice in

wind engineering problems are also going on at the Thematic Area on Civil Construction and HVAC in QNET-CFD [1]. Besides these European activities there is a cooperative project for CFD prediction of wind environment in the Architectural Institute of Japan [7].

The recommendations presented in the following are mainly based on published results. They do not include pollutant dispersion and wind driven rain or snow, but restrict themselves to the prediction of mean velocities and turbulence intensities in urban areas, as they are necessary for the assessment of pedestrian comfort. Nevertheless the results can also be used in these fields, as the computed velocity and turbulence are a prerequisite for the other physical models which are necessary in these simulations. The recommendations are of course subjective and thus open for discussion. Therefore an extended version of this paper is available at the COST C14 homepage (<http://www.costc14.bham.ac.uk/>). It will be continuously updated with new results and approved criticism.

## **2 TOPICS RELATED TO THE USE OF CFD**

Computational fluid dynamics (CFD) is in general a numerical technique in which equations describing the fluid flow are solved on a computer. In case of wind engineering the flow is normally the atmospheric boundary layer (ABL) flow. This is a turbulent boundary layer for which in the context of flow in urban areas a variation of the fluid's properties can normally be neglected [1]. Turbulent flows are described by the well known continuity and momentum equations, named after Navier and Stokes.

When performing a numerical simulation one has to take several aspects into account. First there is the physical model of the flow, which defines the set of equations to be solved. Then the volume in which the flow has to be computed must be defined. This is called the computational domain. This domain has to be discretised by the computational mesh which defines the spatial resolution of the numerical solution. For the discretisation of the equations on this grid appropriate numerical approximations have to be used. For the solution of the discretised equations stopping criteria have to be set to the iterative scheme, as the set of equations in fluid mechanics is non-linear. The resulting solution then has to be analysed and if considered necessary, some of the previous steps have to be repeated with adaptation to the solution. These steps will be discussed in the following with regards to wind engineering problems.

### **2.1 Defining the physical model**

Here the basic equations for the different simulation approaches are shortly reviewed. The main topic is the turbulence models for statistically steady simulations which are necessary to parametrise the unresolved scales of the flow.

#### *2.1.1 Basic equations*

The above mentioned Navier-Stokes equations are known to be valid for the description of turbulent flows. To directly solve the equations requires very fine grids to capture all the relevant scales in the flow, down to the Kolmogorov scale, and a time-dependent solution over sufficiently large times to yield stable time averages of the flow variables. This approach is called direct numerical simulation (DNS). As its computational demand is too high for the Reynolds numbers typically encountered in wind engineering it is not applicable to complex problems in this area.

The computational demands can be substantially reduced when the time-dependent equations are solved on a grid that is too coarse to capture the small scales of the flow. This approach is called large eddy simulation (LES). The small scales are formally removed from the flow variables by spatially filtering the Navier-Stokes equations. The influence of the small scales then appears at least as subfilter stresses in the momentum equation and as boundary terms. If the filter width is not constant then additional subfilter terms arise [8]. All these subfilter terms have to be modelled in terms of the computed large scale quantities. Models have normally only been used for the subfilter stresses while the other subfilter terms have been neglected. Further reference on the state of the art can be found in e.g. Sagaut [9] and Geurts [10], the latter providing some guidelines on the proper use of the method (see also Geurts and Leonard [11]).

The generally used method for the computation of turbulent flows in wind engineering is the Reynolds Averaged Navier-Stokes (RANS) approach. Within this approach the equations are averaged in time over all the turbulent scales, to directly yield the statistically steady solution of the flow variables. Like LES the averaging leads to additional terms in the momentum equation known as the Reynolds stresses which represent the effects of the turbulent fluctuations on the averaged flow, which have to be modelled. This is the task of the turbulence models which, discussed in the next section.

Another option is the use of standard turbulence models from the RANS approach in time-dependent simulations. Contrary to LES the averages are here defined as ensemble or as time averages over small time intervals, although the later definition leads to more additional terms in the momentum equation than in the case of ensemble averaging. This approach is normally called unsteady RANS (URANS). As there are very few applications to wind engineering problems it will not be discussed in the following.

### 2.1.2 Turbulence models

The most common approach in CWE is RANS. Therefore this section focuses on the turbulence models used in RANS. At the end some comments on turbulence models for LES, called subfilter or subgrid scale models, are made.

The task of a turbulence model is to prescribe the turbulent fluxes or Reynolds stresses as a function of the mean flow variables. Two different approaches are used for that in CWE. The first approach is based on the eddy viscosity assumption and models the turbulent stresses by analogy to the molecular stresses as derivatives of the mean velocity. The second approach solves additional differential equations for each unknown Reynolds stress component (differential stress models, DSM; often called Reynolds stress models, RSM). An intermediate option, known as algebraic stress models (ASM), is a non-linear extension of the eddy viscosity models, defined to take into account the anisotropy of the turbulence, through the addition of non-linear functions of the mean strain and vorticity tensors.

For the first approach with the eddy viscosity assumption, the Reynolds stresses depend linearly on the strain rate, as do the molecular stresses. For non-linear algebraic models the dependence of the Reynolds stresses on the mean velocity gradients is quadratic or even cubic [12], enabling them to represent anisotropic normal stresses as they are omnipresent in wind engineering applications. In both eddy viscosity approaches additional equations are usually solved for the turbulent kinetic energy  $k$  and the dissipation of the turbulent kinetic energy  $\varepsilon$ , or other equivalent quantities, such as  $\omega=k/\varepsilon$ . From these two quantities the turbulent or eddy viscosity is calculated.

The industry standard two equation model is the linear standard  $k$ - $\varepsilon$  model and it is widely used in CWE [13] although it is known to produce good results in wind engineering applications only “fortuitously” [1]. One of its main problems is the overproduction of turbulent kinetic energy in regions of stagnant flow (stagnation point anomaly). Several ad-hoc modifications of the model have been proposed as remedy (see Murakami [14] for a short review). These modifications in general improve the prediction of pressure coefficients in front of buildings but lead to worse predictions of the velocities, especially in the wake of obstacles [7,12]. More advanced  $k$ - $\varepsilon$  models like the renormalization group (RNG)  $k$ - $\varepsilon$  model of Yakhot et al. [15], or the realizable  $k$ - $\varepsilon$  model of Shih et al. [16] are increasingly used. These models attenuate the stagnation point anomaly without leading to worse results in the wake. Recent developments tend also to indicate that the shear stress transport (SST) version of the the  $k$ - $\omega$  model, as developed by Menter, provides a significant improvement over standard  $k$ - $\varepsilon$  models. See for instance, Menter [17] and Menter et al. [18]. However, it should be noted that the representation of roughness in such models is not entirely consistent or straightforward.

All linear eddy viscosity models suffer from the assumption of isotropy of the normal Reynolds stresses. Therefore non-linear models, which are able to deal with anisotropy, should perform better in wind engineering. But there are up to now only a few applications of these models (e.g. Krüs et al. [19]). A recent comparative study has been performed by Wright and Easom [12], who simulated the flow around a cube in the ABL. In comparison to the linear RNG  $k$ - $\varepsilon$  model and a DSM the quadratic model of Craft et al. [20] gives the best results. Another comparison of a variety of non-linear  $k$ - $\varepsilon$  models has been performed by Ehrhard et al.

[21] for the flow around a cube in a channel. They found that only the model of Lien et al. [22] showed very good results in comparison to experiments. They blame the insufficient calibration for wind engineering problems for the worse results obtained with the other models. Ishihara and Hibi [23] used the non-linear model of Shih et al. [24] for the simulation of the flow over a 3D hill and found improved results when compared with the standard  $k-\epsilon$  model.

The anisotropy of the Reynolds stresses is naturally contained in the Reynolds stress models. Due to the additional transport equations this approach needs more computational resources and gives only good iterative convergence when fine enough grids of good quality are used. The results for flows around obstacles are often better than with linear eddy viscosity models [25,26], but some of the models also have problems in stagnant flow regions, thus failing to predict experimentally observed reattachment at the top of building models [12,14].

Most of the simulations cited above have been performed for single obstacles. Simulations for more complex industrial scenarios are reported by Ferreira et al. [27], who analysed building interference effects on pedestrian level comfort and found good agreement between computation and measurement using the RNG  $k-\epsilon$  model, and Richards et al. [28], who computed pedestrian level wind speeds in Auckland with a standard  $k-\epsilon$  model and obtained results which are similar to wind tunnel erosion patterns. No parameter test was performed in both studies. A moderately complex street canyon has been analysed with five different programs, all using the standard  $k-\epsilon$  model by Ketzler et al. [29] and the results compared with wind tunnel measurements for the mean velocities. They found good agreement for the general flow field but differences in the magnitude of the velocities. As a very coarse grid has been used in that study, Theodoridis et al. [30] analysed the influence of local grid refinement and found similar results on the coarse and refined grid. But their refinement has not been systematic and substantial, as will be detailed in section 2.5.2, so the results must be regarded with caution.

As an intermediate summary it can be stated that the standard  $k-\epsilon$  model should not be used in the simulation of wind engineering problems. Preferably non-linear models or Reynolds stress models should be used, although these still require improvements. But it must be kept in mind that just a few of the cited comparisons analysed the grid dependence (see section 2.5.2) of the results. Thus the results must be regarded as a mixture of the influence of the turbulence model and the discretisation error.

A general view on numerical simulation of wind engineering problems is that a time-dependent approach can yield more accurate results than statistically steady RANS simulations [1,12,13,21]. LES is supposed to be the most general method to lead to better results in the prediction of bluff body flows. Despite the need for further research in subgrid scale modelling, which is briefly reviewed in the following, and in appropriate numerical approximations, large eddy simulations have shown to generally reproduce main turbulence properties with a higher accuracy, as compared to standard RANS type models. This is however obtained at a significant higher cost in CPU times, which will remain unrealistic at least for the foreseeable future, for engineering applications.

As has been said in section 2.1.1 LES also needs models for the filtered small scales. Murakami [14] has shown that dynamic models, which use the information of the smallest resolved scales in modelling the unresolved scales, lead to improved results for the flow around a square cylinder in comparison to the constant viscosity Smagorinsky model. Another promising class of models use approximate reconstructions of the unfiltered flow variables (see Sagaut [9] and Geurts [10] and references therein). These models have given very good results in simple flows but have not yet been applied to wind engineering problems. Their principal advantage over the dynamic models is that they can also take into account the above mentioned additional subfilter terms that arise besides the subfilter stresses in wind engineering problems, where the filter width is very seldom constant. While the treatment of these additional terms has stirred the basic research of the method, another approach to LES has spread, where no explicit subfilter models are used at all. This approach is called Monotonically Integrated LES (MILES) and uses the built in dissipation of high resolution schemes for the numerical approximations of the advective terms in the Navier-Stokes equations. E.g. Grinstein and Fureby [31] have obtained very good results with this method for a backward facing step flow for a step-height Reynolds number of 22000. Applications of LES to wind engineering

problems are reported e.g. from Nozawa and Tamura [32], who analysed root mean square and peak pressures on a low rise building, from Rehm et al. [33], who studied the influence of surrounding buildings on pressure fluctuations on a building, and from He and Song [34], who evaluated pedestrian winds in an urban area. Unfortunately the last two publications contain no comparison with experiments. The standard test case for external aerodynamics is still the flow around a cube in a channel. A recent discussion on LES results for this case is provided by Rodi [35].

## 2.2 Defining the computational domain

A CAD model of the built environment to be examined has to be generated. This area is enclosed by the computational domain which cuts off the surroundings that in turn must be represented by approximate boundary conditions. Several wind directions have to be analysed. Wisse et al. [36] argued that twelve wind directions are enough for the analysis of pedestrian comfort. The following recommendations should be applied to all these directions.

### 2.2.1 Domain size

The size of the entire computational domain in vertical, lateral and flow direction depends on the area that shall be represented and on the boundary conditions that will be used. For single buildings the guidelines of Hall [37] can be applied. The inlet, the lateral and the top boundary should be  $5H$  away from the building, where  $H$  is the building height. For buildings with an extension in lateral direction much larger than the height, the blockage ratio should be below 3% [38]. The outflow boundary should be positioned at least  $15H$  behind the building to allow for flow development, as fully developed flow is normally used as a boundary condition. For the same reason this outflow length should also be applied for an urban area with many buildings, where  $H$  is to be replaced by  $H_{\max}$ , the height of the tallest building. To prevent an artificial acceleration of the flow over the tallest building, the top of the computational domain should be also at least  $5H_{\max}$  away from this building. For the blockage ratio the limit of 3% is recommended, although there are no results on whether it is better to include more of the surrounding buildings in the model and reduce the distance of the lateral boundaries from the built area.

The extent of the built area (e.g. buildings, structures or topography) that is represented in the computational domain depends on the influence of the features on the building or region of interest. In some wind tunnel simulations for example an area with a radius of 300 m around the building or place of interest is modelled [39]. Another experience from wind tunnel simulations is that a building with height  $H_n$  may have a minimal influence if its distance from the region of interest is greater than  $6-10H_n$ . Thus as a minimum requirement a building of height  $H_n$  should be represented if its distance from the region of interest is less than  $6H_n$ .

While urban areas usually do not possess any geometrical symmetry, simpler obstacles can be symmetric for certain wind directions. In these cases the symmetry can be used as boundary condition and the computational domain can be halved. But it should always be verified in advance that the flow is really symmetric by performing a simulation in the full domain, as even geometric symmetry can produce asymmetric flows as has been shown by Prevezer et al. [40].

### 2.2.2 Geometrical representation of details

Normally the overall mass-distribution of buildings has the greatest impact on wind flow patterns. Details of the facades and roofs are of secondary importance (particularly in the case of sharp-edged cubic structures). The level of detail required for individual buildings is dependent on their distance from the central building of interest. The central building at which wind effects are of main interest requires the greatest level of detail, and here features greater than about 1m should be represented. Buildings further away may normally be represented as simple blocks.

The level of detail may depend on the application. For example, if surface pressures on the roof of a particular building are of interest, the need to represent details on the roof are more critical than if the pedestrian-level wind speeds are required. Again, the level of detail may be limited by the computational mesh required to resolve these details. In some cases, it may be

possible to judge whether the omission of various details are likely to make the results more or less conservative.

The need to represent local landscaping (for example, vegetation) depends on the application of the results. Pedestrian comfort can be improved by vegetation. However, there appears to be no documentation on the effect of modelled vegetation in relation to real vegetation. If the pressures on ventilation openings at roof level, say, are of interest, the need to represent landscaping is likely to be less critical.

### 2.2.3 *Boundary conditions*

The boundary conditions represent the influence of the surroundings that have been cut off by the computational domain. At the top boundary usually symmetry is prescribed, which enforces a parallel flow. The same is applied at the lateral boundaries. Therefore the blockage ratio should obey the recommendation given in section 2.2.1 to prevent a too strong artificial acceleration of the flow. However, another option is to handle the top and side boundaries as outflow boundaries, allowing a normal velocity component at these boundaries, which is zero in the case of symmetry boundary conditions. Thus the natural outflow, which is due to the increasing displacement of the fluid even in a boundary layer flow without obstacles, is taken into account. Of course no re-entry of the flow across these boundaries is allowed. Therefore the blockage ratio should be also below 3%, when using an outflow boundary condition at the top and side boundaries.

At the boundary behind the obstacles, where all or most of the fluid leaves the computational domain, an outflow boundary is used. At the outflow boundary the derivatives of all flow variables are forced to vanish, corresponding to a fully developed flow. Therefore this boundary should be far away from the built area, as already stated in section 2.2.1.

At the inflow an equilibrium boundary layer is usually prescribed, at a distance of at least  $5H$ , see section 2.2.1. The mean velocity is obtained from the logarithmic profile corresponding to the upwind terrain via the roughness length  $z_0$ . Available information from nearby meteorological stations or the profiles of the wind tunnel simulations are used in determining the wind speed at the reference height. For two-equation turbulence models the boundary for the turbulent kinetic energy and its dissipation are also obtained from the assumption of an equilibrium boundary layer. The relevant formulas are described by Richards and Hoxey [41]. The same coefficients that are used in the turbulence model should be used in the analytical formulation of the boundary conditions. Before simulating the flow over obstacles it should be analysed whether the chosen grid and boundary conditions are consistent and there is no substantial change in the specified boundary profiles. Whether this requirement is fulfilled depends crucially on the roughness of the bottom wall. In most commercial programs the roughness of a wall is implemented for sand roughened surfaces with a corresponding roughness height  $k_s$ . For a fully rough surface the roughness length  $z_0$  and the roughness height  $k_s$  are related via  $k_s = z_0 \exp(\kappa B)$ , where  $\kappa$  is the von Karman constant ( $\kappa \approx 0.4$ ) and  $B \approx 8.5$  is the constant in the logarithmic velocity profile for rough surfaces (see e.g. Durbin and Petterson Reif [42]). For those values the relation is  $k_s \approx 30z_0$ , showing that the roughness height is one order larger than the roughness length. This leads to very large computational cells at the rough wall since the first calculation node off the wall should be placed at least  $k_s$  away from the wall. This leads to a very bad resolution of the flow close to the wall. For wind comfort analysis this is not appropriate and one should rather use a smooth wall in the region of interest with corresponding smaller cells to place the height at which pedestrian wind speeds are calculated (between 1.5 and 2m) in the third or fourth cell away from the wall. Positions where a solution is looked for should in general not be placed in the immediate neighbourhood of a wall, due to the usual wall function modelling of the flow at the wall. This modelling is known to be invalid in regions of flow separation. The effect of wall functions on the solution away from the wall is however small [1]. The VDI Richtlinie [6] also recommends placing at least two nodes between the wall and the position of interest.

Concerning LES the modelling at walls is also still an open question, even for smooth walls (see e.g. Piomelli and Balaras [43]). Another problem is the specification of time-dependent inflow conditions corresponding to the approaching turbulent boundary layer. These are either generated from random functions that give a intended spectrum and take the

spatial correlation into account [44], or from a separate calculation of a rough boundary layer flow with periodic boundary conditions, as proposed by Nozawa and Tamura [32].

### 2.3 Defining the computational grid

The computational results depend crucially on the grid that is used to discretise the computational domain. The grid has to be designed in such a manner that it does not introduce errors that are too large. This means that the resolution of the grid should be fine enough to capture the important physical phenomena like shear layers and vortices with sufficient resolution. Also the quality of the grid should be high. Therefore grid stretching/compression should be small in regions of high gradients, to keep the truncation error small. The expansion ratio between two consecutive cells should be below 1.3 in these regions. For the widely used Finite Volume methods another criterion for grid quality is the angle between the normal vector of a cell surface and the line connecting the midpoints of the neighbouring cells [45]. Ideally these should be parallel.

With regard to the shape of the computational cells, hexahedra are to be preferred over tetrahedra, as the former are known to introduce smaller truncation errors and display better iterative convergence [46]. On walls the grid lines should be perpendicular to the wall [5]. Prismatic cells should therefore be used together with tetrahedral cells away from the wall, if a tetrahedral grid is used. E.g. Fothergill et al. [26] found improved results for a prismatic/tetrahedral grid as compared to a purely tetrahedral grid.

For the necessary resolution it is impossible to make recommendations in advance as this is very problem dependent. If simulations employ the logarithmic wall model, the position of the first computational node should be of course placed in the logarithmic region, corresponding to a non-dimensional wall distance of at least 30 [5]. The wall distance must also comply with the prescribed wall roughness, as detailed in section 2.2.3. For the resolution of the built area at least 10 cells per cube root of the building volume should be used and 10 cells per building separation. This must be understood as initial minimum grid resolution. The necessary resolution then will have to be analysed by using grid refinement which is discussed separately in section 2.5.2.

### 2.4 Defining the numerical approximations

To render the basic equations described in section 2.1 solvable on the computer, they have to be discretised and transformed into algebraic equations. The most important numerical approximation is the one used for the non-linear advective terms in the basic equations (see e.g. Cowan et al. [47]). First order methods like the upwind scheme must not be used. They can and should be used for the initial iterations, but then higher order approximations must be used for the final solution. It should be noted that the ASME Journal of Fluid Engineering has a policy of not publishing results from first order approximations [48].

For time dependent problems second order methods are also recommended for the approximation of time derivatives.

### 2.5 Defining the solution

The resulting system of algebraic equations is finally solved on a computer. Although in the RANS approach a steady solution is presumed it should be verified by a time-dependent calculation that the solution reaches a steady state.

One should also be aware that most programs limit some flow variables to reasonable values. These limits should be checked in advance of the simulation. Some programs continuously print information about limitation of variables (e.g. turbulent kinetic energy, turbulent dissipation or temperature).

#### 2.5.1 Iterative convergence

Most of the computer programs use iterative methods to solve the algebraic system of equations. Starting from an initial guess the flow variables are recalculated in every iteration until each equation is solved up to an user specified error. The termination criterion is usually based on the residuals of the corresponding equations. These residuals should tend towards

zero. Scaling of the residuals is usually done with the residuals after the first iteration. The scaled residual then shows how much the initial error has dropped. In industrial applications typically a termination criterion of 0.001 is used, which is in general too high to have a converged solution. A reduction of the residuals of at least five orders of magnitude is recommended. For validation purposes of turbulence or other physical models much lower criteria should be used. If the residual is driven down to machine accuracy ( $10^{-12}$  for double precision), there is no more iterative error present in the solution. In addition to the residuals, monitoring positions should be defined in the region of interest and local flow variables should be recorded. If these variables are constant, then the solution in the region of interest can be regarded as converged.

If the solution shows bad convergence or no convergence at all one can decrease the under-relaxation factors. But one should then check at the end of the calculation whether switching back to the default values alters the solution [5].

### 2.5.2 *Grid dependence of the solution*

Every numerical solution depends on the grid that is used. As has been said in section 2.3, the grid must have a good quality, especially in the region of interest. To quantify the influence of the grid resolution on the solution a grid convergence study should be ideally made. For this at least three systematically and substantially refined grids should be used. The ratio of cells for two consecutive grids should be at least 3.4 [45]. With the results on the three grids the error in the solution can be estimated with Richardson extrapolation [49,50], if the three grids are fine enough to yield results in the asymptotic range. This should be done for local or average values in the area of interest. If computing capacity does not allow grid sizes in the asymptotic range, then this result also shows that the grid is not yet fine enough. Instead of refining the entire grid, local grid refinement based on some refinement criterion (usually derivatives of flow variables) can be used to estimate the grid independent solution.

## 3 **VALIDATION REQUIREMENTS**

The recommendations given above are based on knowledge about CFD in general and on published results that mainly deal with the flow around single obstacles. To establish secured guidelines on CWE for the prediction of pedestrian wind in urban areas by steady RANS simulations further validation is certainly necessary. In the validation strategy the systematic differences between field experiments, laboratory experiments and RANS simulations, as detailed by Schatzmann and Leitl [2], must be kept in mind. While the results from RANS simulations correspond to time averages over a theoretically infinite time interval, the data provided by field experiments represent averages over 10 min or 30 min time intervals. Longer intervals are not feasible, because meteorological conditions already change within 30 min. Due to the resulting poor repeatability of field experiments the computational results should be validated with data from wind tunnel experiments with steady-state boundary conditions, like they are available in the CEDVAL data base for dispersion modelling in idealized obstacle arrays [51]. Detailed measurements of the velocities and the turbulence in the approach flow and in the urban area are necessary, not only at pedestrian level. If time-dependent simulations are to be included in the validation, also peak gust wind speeds should be measured. These are also useful for assessing the possibility of deriving peak gust wind speeds from steady state calculations. The complete data sets should be available for at least twelve wind directions, as discussed by Wisse et al. [36].

The numerical simulations should then be performed taking the above recommendations into account. If different programs are to be compared it must be ensured that the same grid(s), boundary conditions and iterative convergence criteria are used [52]. If different turbulence or other physical models are to be compared within one program, iterative convergence down to machine accuracy is recommended to eliminate the iterative convergence error. On all accounts it must be ensured that the numerical solutions at the measurements points are constant or at least fluctuate around a constant value. Also the discretisation error must be estimated with a grid refinement study, thus leaving only the error of the turbulence model in the solution. A more theoretical and comprehensive discussion of verification and validation for CFD can be found in the book by Roache [53].



## 4 CONCLUSIONS

As the greatest concern about the accuracy of CFD in wind engineering is attributed to the lack of clear guidelines on the physical and numerical parameters that have to be provided by the user to the computer programs, this work summarised the results for mean velocities and turbulence in the built environment from statistically steady RANS simulations available in the literature to deduce recommendations on the proper use of CWE for that purpose. These subjective guidelines can be recapitulated as follows:

- Avoid using the standard  $k$ - $\epsilon$  model. Either use more advanced linear eddy viscosity models like the RNG  $k$ - $\epsilon$  or the realizable  $k$ - $\epsilon$  model. Ideally use non-linear eddy viscosity models or Reynolds stress models to account for the anisotropy of the Reynolds stresses.
- The blockage of the flow by the built area should be below 3 % and the outflow boundary far enough away from the built area, in a region of developed flow.
- Verify the assumption of an equilibrium boundary layer corresponding to the prescribed approach flow by e.g. performing a simulation in an empty domain with the same grid and boundary conditions.
- Do not use first order schemes for numerical approximations.
- Judge iterative convergence of the solution by monitoring key values in the region of interest in addition to the residuals. If performing turbulence or other physical model validation the solution should be converged to machine accuracy.
- The minimal grid resolution should be 10 cells per cube root of a building volume and 10 cells per building separation. Hexahedra or at least prisms should be used at walls. Pedestrian wind speeds should not be analysed in the first cell on the ground. Use local grid refinement in the region of interest to check for grid dependence of the results. Ideally perform a systematic grid convergence study which should be always used in validation efforts.
- The documentation of the simulation must include all the parameters mentioned above.

It is also concluded that further validation is necessary to secure the recommendations with the aid of complete data sets from wind tunnel measurements of pedestrian wind in complex urban areas. Wind tunnel measurements have to be used for the validation, as they use steady-state boundary conditions which comply with the underlying definition of the statistically steady RANS results.

## 5 REFERENCES

- 1 Castro, I.P., “CFD for external aerodynamics in the built environment”, The QNET-CFD Network Newsletter, Vol. 2, No. 2 (2003), 4-7. See also <http://www.qnet-cfd.net>.
- 2 Schatzmann, M. and Leitl, B., “Validation and application of obstacle-resolving urban dispersion models”, Atmospheric Environment, Vol. 36, No. 30 (2002), 4811-4821.
- 3 Westbury, P.S., Miles, S. and Stathopoulos, T., “CFD application on the evaluation of pedestrian-level winds”, in G. Augusti, C. Borri, and C. Sacré, editors, *Impact of Wind and Storm on City life and Built Environment*, pages 172-181, CSTB, Nantes, 2002.
- 4 Stathopoulos, T., “The numerical wind tunnel for industrial aerodynamics: Real or virtual in the new millenium?”, Wind & Structures, Vol. 5, No. 2-4 (2002), 193-208.
- 5 Casey, M. and Wintergerste, T., editors, *ERCOFTAC SIG "Quality and Trust in Industrial CFD": Best Practice Guidelines*. ERCOFTAC, 2000.
- 6 VDI-Richtlinie 3783, Blatt 9 (Entwurf), *Umweltmeteorologie. Prognostische mikroskalige Windfeldmodelle. Evaluierung für Gebäude- und Hindernisumströmung*, (in German), 2003.
- 7 Mochida, A., Tominaga, Y., Murakami, S., Yoshie, R., Ishihara, T. and R. Ooka, “Comparison of various  $k$ - $\epsilon$  models and DSM to flow around a high rise building - report of AIJ cooperative project for CFD prediction of wind environment”, Wind & Structures, Vol. 5, No. 2-4 (2002), 227-244.
- 8 Ghosal, S. and Moin, P., “The basic equations for large eddy simulation of turbulent flows in complex geometries”, Journal of Computational Physics, Vol. 118 (1995), 24-37.
- 9 Sagaut, P., *Large Eddy Simulation for Incompressible Flows*, Springer Verlag, Berlin Heidelberg New York, 2001.

- 10 Geurts, B.J., *Elements of direct and large-eddy simulation*, Edwards, Philadelphia, 2004.
- 11 Geurts, B.J. and Leonard, A., "Is LES Ready for Complex Flows?", in B. E. Launder and N. Sandham, editors, *Closure Strategies for Turbulent and Transitional Flows*, pages 720-739, Cambridge University Press, Cambridge, 2002.
- 12 Wright, N.G. and Easom, G.J., "Non-linear k- $\epsilon$  turbulence model results for flow over a building at full scale", *Applied Mathematical Modelling*, Vol. 27 (2003), 1013-1033.
- 13 Murakami, S., "Setting the scene: CFD and symposium overview", *Wind & Structures*, Vol. 5, No. 2-4 (2002), 83-88.
- 14 Murakami, S., "Overview of turbulence models applied in CWE-1997", *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 74-76 (1998), 1-24.
- 15 Yakhot, V., Orszag, S.A., Thangam, S., Gatski, T.B. and Speziale, C.G., "Development of turbulence models for shear flows by a double expansion technique", *Phys. Fluids A*, Vol. 4 (1992), 1510-1520.
- 16 Shih, T.-H., Liou, W.W., Shabbir, A., Yang, Z. and Zhu, J., "A New k- $\epsilon$  Eddy-Viscosity Model for High Reynolds Number Turbulent Flows - Model Development and Validation", *Computers Fluids*, Vol. 24, No. 3 (1995), 227-238.
- 17 Menter, F., "Eddy viscosity transport equations and their relation to the k- $\epsilon$  model", *Journal of Fluids Engineering*, Vol. 119 (1997), 876-884.
- 18 Menter, F. R., Kuntz, M. and Langtry, R., "Ten Years of Industrial Experience with the SST Turbulence Model", in K. Hanjalic, Y. Nagano, M. Tummers, editors, *Turbulence, Heat and Mass Transfer 4*, Begell House Inc., 2003.
- 19 Krüs, H.W., Haanstra, J.O., van der Ham, R. and Wichers Scheur, B., "Numerical simulations of wind measurements at Amsterdam airport Schipohl", in J. Wisse, K. Kleinman, C. Geurts and de Wit, M., editors, *Proceedings of the 3<sup>rd</sup> European & African Conference on Wind Engineering*, pages 201-208, FAGO and CO, Eindhoven, 2000.
- 20 Craft, T.J., Launder, B.E. and Suga, K., "Development and application of a cubic eddy-viscosity model of turbulence", *Int. J. Heat and Fluid Flow*, Vol. 17 (1996), 108-115.
- 21 Ehrhard, J., Kunz, R. and Moussiopoulos, N., "On the performance and applicability of nonlinear two-equation turbulence models for urban air quality modelling", *Environmental Monitoring and Assessment*, Vol. 65, No. 1-2, (2000), 201-209.
- 22 Lien, F.S., Chen, W.L. and Leschziner, M.A., "Low-Reynolds number eddy-viscosity modelling based on non-linear stress-strain/vorticity relations", in W. Rodi, editor, *Engineering Turbulence Modelling and Experiments 3*, Elsevier, Amsterdam, 1996.
- 23 Ishihara, T. and Hibi, K., "Numerical study of turbulent wake flow behind a three-dimensional steep hill", *Wind & Structures*, Vol. 5, No. 2-4 (2002), 317-328.
- 24 Shih, T.H., Zhu, J. and Lumley, J.L., "A new Reynolds stress algebraic equation model", *Comput. Methods Appl. Mech. Eng.*, Vol. 125 (1995), 287-302.
- 25 Meroney, R.N., Leitl, B.M., Rafailidis, S. and Schatzmann, M., "Wind-tunnel and numerical modeling of flow and dispersion about several building shapes", *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 81 (1999), 333-345.
- 26 Fothergill, C.E., Roberts, P.T. and Packwood, A.R., "Flow and dispersion around storage tanks. A comparison between numerical and wind tunnel simulations", *Wind & Structures*, Vol. 5, No. 2-4 (2002), 89-100.
- 27 Ferreira, A.D. , Sousa, A.C.M. and Viegas, D.X., "Prediction of building interference effects on pedestrian level comfort", *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 90 (2002), 305-319.
- 28 Richards, P.J., Mallinson, G.D., McMillan, D. and Li, Y.F., "Pedestrian level wind speeds in downtown Auckland", *Wind & Structures*, Vol. 5, No. 2-4 (2002), 151-164.
- 29 Ketzler, M., Louka, P., Sahn, P., Guilloteau, E., Sini, J.-F. and Moussiopoulos, N., "Inter-comparison of numerical urban dispersion models – Part II: Street canyon in Hannover, Germany", *Water, Air, and Soil Pollution: Focus*, Vol. 2, No. 5-6 (2002), 603-613.
- 30 Theodoridis, G., Karagiannis, V. and Valougeorgis, D., "Numerical prediction of dispersion characteristics in an urban area based on grid refinement and various turbulence models", *Water, Air, and Soil Pollution: Focus*, Vol. 2, No. 5-6 (2002), 525-539.
- 31 Grinstein, F.F. and Fureby, C., "Recent Progress on MILES for High Reynolds Number Flows", *Journal of Fluids Engineering*, Vol. 124 (2002), 848-861.

- 32 Nozawa, K. and Tamura, T., "Large eddy simulation of the flow around a low-rise building immersed in a rough-wall turbulent boundary layer", *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 90, No. 10 (2002), 1151-1162.
- 33 Rehm, R.G., McGrattan, K.B. and Baum, H.R. "Large eddy simulation of the flow over wooded building complex", *Wind & Structures*, Vol. 5, No. 2-4 (2002), 291-300.
- 34 He, J. and Song, C.C.S., "Evaluation of pedestrian winds in urban area by numerical approach", *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 81 (1999), 295-309.
- 35 Rodi, W., "Large-Eddy Simulation of the Flow past Bluff Bodies", in B. E. Launder and N. Sandham, editors, *Closure Strategies for Turbulent and Transitional Flows*, pages 361-391, Cambridge University Press, Cambridge, 2002.
- 36 Wisse, J.A., Krüs, H.W. and Willemsen, E., "Wind comfort assessment by CFD - Context and requirements", in G. Augusti, C. Borri, and C. Sacré, editors, *Impact of Wind and Storm on City life and Built Environment*, pages 154-163, CSTB, Nantes, 2002.
- 37 Hall, R.C. (Ed.), *Evaluation of modelling uncertainty. CFD modelling of near-field atmospheric dispersion. Project EMU final report*, European Commission Directorate-General XII Science, Research and Development Contract EV5V-CT94-0531, WS Atkins Consultants Ltd., Surrey, 1997.
- 38 Baetke, F. and Werner, H., "Numerical Simulation of Turbulent Flow over Surface Mounted Obstacles with Sharp Edges and Corners", *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 35 (1990), 129-147.
- 39 Willemsen, E., private communication, 2003.
- 40 Prevezer, T., Holding, J., Gaylard, A. and Palin, R., "Bluff body asymmetric flow phenomena – real effect or solver artefact?", *Wind & Structures*, Vol. 5, No. 2-4 (2002), 359-368.
- 41 Richards, P.J. and Hoxey, R.P., "Appropriate boundary conditions for computational wind engineering models using the k- $\epsilon$  turbulence model", *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 46 & 47 (1993), 145-153.
- 42 Durbin, P.A. and Petterson Reif, B.A., *Statistical Theory and Modeling for Turbulent Flows*, John Wiley & Sons, Chichester, 2001.
- 43 Piomelli, U. and Balaras, E., "Wall-Layer Models for Large-Eddy Simulations", *Annu. Rev. Fluid. Mech.*, Vol. 34 (2002), 349-374.
- 44 Kondo, K., Tsuchiya, M., Mochida, A. and S. Murakami, S., "Generation of inflow turbulent boundary layer for LES computation", *Wind & Structures*, Vol. 5, No. 2-4 (2002), 209-226.
- 45 Ferziger, J.H. and Perić, M., *Computational Methods for Fluid Dynamics*, Springer Verlag, Berlin Heidelberg New York, 3rd edition, 2002.
- 46 Hirsch C., Bouffieux, V. and Wilquem F, "CFD simulation of the impact of new buildings on wind comfort in an urban area", in G. Augusti, C. Borri, and C. Sacré, editors, *Impact of Wind and Storm on City life and Built Environment*, pages 164-171, CSTB, Nantes, 2002.
- 47 Cowan, I.R., Castro, I.P., and Robins, A.G., "Numerical considerations for simulations of flow and dispersion around buildings", *Journal of Wind Engineering and Industrial Aerodynamics*, 67 & 68 (1997), 535-545.
- 48 Freitas, C.J., "Editorial policy statement on the control of numerical accuracy", *Journal of Fluids Engineering*, Vol. 115, No. 3 (1993), 339-440.
- 49 Roache, P.J., "Quantification of Uncertainty in Computational Fluid Dynamics", *Annu. Rev. Fluid Mech.*, Vol. 29 (1997), 123-160.
- 50 Stern, F., Wilson, R.V., Coleman, H.W. and Paterson, E.G., "Comprehensive Approach to Verification and Validation of CFD Simulations - Part 1: Methodology and Procedures", *Journal of Fluids Engineering*, Vol. 123 (2001), 793-802.
- 51 Leitl, B., "Validation Data for Microscale Dispersion Modeling", *EUROTRAC Newsletter*, Vol. 22 (2000), 28-32. See also <http://www.mi.uni-hamburg.de/cedval>.
- 52 Iaccarino, G., "Predictions of a turbulent separated flow using commercial CFD codes", *Journal of Fluids Engineering*, Vol. 123 (2001), 819-828.
- 53 Roache, P.J., *Verification and Validation in Computational Science and Engineering*, Hermosa Publishers, Albuquerque, New Mexico, 1998.