E-Wind: Steady state CFD approach for stratified flows used for site assessment at Enercon

M. Alletto¹, A. Radi², J. Adib³, J. Langner¹, C. Peralta¹, A. Altmikus³ and M. Letzel¹

¹WRD Management Support GmbH, Costumer support, Teerhof 58, 28199 Bremen, Germany
²Enercon GmbH, Z.I. Le Taillis 16, rue des Bosquets, 44840 Les Sorinières, France
³WRD GmbH, Borsigstr. 26, 26607 Aurich, Germany

Email: michael.alletto@enercon.de

Abstract. A new industrial methodology for CFD-based site assessment is presented. It is based on a steady state two equation RANS model and includes the effect of Coriolis force, forests and buoyancy in the equations for turbulence. The computational results are validated with Monin-Obukhov similarity theory, the Høvsøre meteorological mast and one real site experiment. The validation evaluates the profile quality over homogenous terrain, with focus on profile consistency and shape. Different heat fluxes and roughness lengths are considered. In the second validation section the well-known Askervein hill is simulated in order to demonstrate the applicability of the model to a real site.

1. Introduction

With substantial exploitation of wind energy in the near shore regions, simple terrain suitable for wind parks becomes more and more rare. For this reason the wind energy industry is forced to erect turbines in locations of increasing terrain complexity. Unfortunately, the prediction of the atmospheric flow for site assessment and power prediction is far from simple in such terrains. Driven by the demand to lower the risk associated with erroneous predictions of the flow fields in complex sites, an in-house CFD solution was developed at Enercon. The strategic goal was to have full control over the CFD solver quality in order to increase the reliability of the predictions. Furthermore, custom made solutions for assessing risky turbine position will become easier to implement. In the present paper, Enercon’s methodology to calculate the wind flow in the framework of computational fluid dynamics (CFD) is presented. This methodology accounts for most relevant factors influencing the evolution of the fluid flow in complex sites, i.e., steep terrain, roughness changes, forests, atmospheric stability and the Coriolis force. Other factors, such as humidity or mesoscale effects are ignored in the solution presented here. The paper is organized as follows: In Section 2 the CFD methodology is described. In Section 3 the CFD model is evaluated for the standard academic case of a homogeneous flow in flat terrain. Finally in Section 4 the model is tested on the well-known Askervein hill test case.

2. Description of the methodology

The methodology relies on the solution of the 3D Reynolds Averaged Navier Stokes (RANS) equations, with a two equation k-ε turbulence closure (see [1] and [2]). The governing equations are implemented in the open source toolbox OpenFOAM (version 2.3.0). The topography and roughness
information are provided by the user together with the climatology (from wind measurement masts) in order to calibrate the simulation. The calibration process is the adjustment of the simulated shear in order to match the measured wind shear at a given measurement mast as closely as possible. This adjustment is done for each wind sector.

The topography information is processed automatically and a cylindrical computational grid is generated using the commercial grid generator Pointwise (Figure 1). The first step of the meshing is to draw an enclosing circle with the radius $R_{ec}$ around all points relevant for the site evaluation (turbines and mast). This circular area is surrounded by a square with the side length $B$, within which the desired constant horizontal resolution ($H_{res}$) and the desired vertical cell size at the ground ($V_{res}$) are set. Outside the inner square, the grid is stretched towards the outer radius $R_{out}$; also a vertical cell stretching is applied throughout the whole domain. 500m above the highest terrain elevation (denoted as meshULinterfaceHeight in Figure 1), the horizontal resolution is doubled, using the ability of OpenFOAM to interpolate between patches of non-conformal grids.

Within $R_{cut}$, the topography is kept unchanged. Between $R_{cut}$ and $R_{smooth}$ the original terrain is ramped down radially using a $cos^2$ function. Outside $R_{smooth}$ the terrain is flat.

The roughness information is processed automatically to generate appropriate (ground) boundary condition for the governing equations. The atmospheric stability information is incorporated by setting the heat flux at the ground and choosing the corresponding 1D inflow profiles from a pre-generated library. This library consists of pre-calculated numerical profiles generated by a 1D version of the CFD model, an approach also used in [3]. Effectively, the change of atmospheric stability is modelled by the reduction or increase of the turbulent momentum transport as function of the applied heat flux at the ground. The discretisation of the convective part of the momentum equation was set to second order upwind scheme, and for the convective part of the other equations (the turbulent kinetic energy $k$, the turbulent dissipation rate $\epsilon$ and the potential temperature $\theta$) a first order upstream scheme was used. The diffusion part of all equations is discretized with a second order scheme.

![Figure 1](image.png)

**Figure 1.** Overview of the meshing strategy: Schematic of the round domain (left). Sketch of the two different mesh resolutions applied (right).

2.1. Governing equations

The equations are based on the work of [1] and [2]:

```
\[
\frac{\partial \rho u_i}{\partial t} + \frac{\partial \rho u_i u_j}{\partial x_j} = 0 \tag{1}
\]

\[
\frac{\partial \rho u_i}{\partial t} + \frac{\partial \rho u_i u_j}{\partial x_j} - \frac{\partial}{\partial x_j} \left( \rho (\mathbf{v} + \mathbf{v}_e) \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right) = -\frac{\partial p}{\partial x_i} - \tau_i - \rho \varepsilon_{ijk} f_j \bar{u}_k - \rho c_d LAD \bar{u}_i \bar{u}| \tag{2}
\]

\[
\frac{\partial \rho \theta}{\partial t} + \frac{\partial \rho \theta u_i}{\partial x_i} = \frac{\partial}{\partial x_i} \left( \mu \frac{\partial \theta}{\partial x_i} \right) + S_\theta \tag{3}
\]

\[
\frac{\partial k}{\partial t} + \bar{u}_j \frac{\partial k}{\partial x_j} - \frac{\partial}{\partial x_j} \left( \nu + \frac{\nu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} = P_k - \varepsilon + B \tag{4}
\]

\[
\frac{\partial \varepsilon}{\partial t} + \bar{u}_j \frac{\partial \varepsilon}{\partial x_j} - \frac{\partial}{\partial x_j} \left( \nu + \frac{\nu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} = \left( C_{e1} + C_{e2} \right) \left( \frac{l}{l_{max}} \right)^{\alpha_k} \frac{\varepsilon}{k} P - C_{e2} \frac{\varepsilon^2}{k} + \left( C_{e1} - C_{e2} \right) \frac{\varepsilon}{k} B - (C_{e1} - C_{e2}) \frac{\varepsilon}{k} S_d \tag{5}
\]

Equation (1) is the continuity equation, (2) is the momentum equation, (3) the energy equation, (4) the equation for the turbulent kinetic energy \(k\) and (5) the equation for the dissipation rate \(\varepsilon\). In the momentum equation (2) the second term \(\tau_i\) on the right hand side represents the geostrophic wind (or driving pressure gradient), the third term the Coriolis force and the forth term the force exerted by the forest on the flow. The geostrophic forcing is set to obtain a geostrophic wind of 17.5 m/s at a latitude of 51° north. \(\rho\) is the fluid density, \(c_d\) the forest drag coefficient and \(LAD\) the leaf area density. The forest source term in the momentum equation describes the drag force exerted by the trees on the flow. It is modelled as a pressure induced resistance (see also [4] and [5]). The influence of buoyancy in the momentum equation is neglected for the sake of numerical stability. The term \(S_\theta\) in the equation (3) for the potential temperature \(\theta\) is used to take into account the heat flux transferred from the canopy to the surrounding fluid:

\[
S_\theta = \alpha_{FH} S_{FH} \frac{\varepsilon}{\sigma_\theta} \tag{6}
\]

\(\alpha_{FH}\) is set equal to 1 in the upper half of the forest and 0 everywhere else. \(S_{FH}\) represents the magnitude of the forest heat source term, and is set to \(S_{FH} = 2 \frac{\partial}{\partial x}\). \(H_T\) is the tree height and \(\partial\) the (sensible) heat flux applied on the bare ground. Factor 2 is used since the heat source term is only active in the upper half of the trees. The above expression is derived from two requirements: First: the heat transmitted from the forest to the above fluid should be equal to the heat flow at the bare ground of the patches without forest. Second: The magnitude is derived from the assumption that all the heat generated inside the forest is transmitted to the surrounding fluid at the top of the forest.

The influence of buoyancy is only taken into account in the equations for turbulence (4) and (5) by the term

\[
B = \beta g_i \frac{\nu_t}{\sigma_\theta} \frac{\partial \theta}{\partial x_i} \tag{7}
\]

Note that the model of the buoyancy production by [6] is used instead of the model described in [2]. \(\beta = 0.0033 \, 1/K\) is the thermal expansion coefficient, \(g_i\) the gravitational acceleration, \(\nu_t\) the turbulent viscosity and \(\sigma_\theta\) the turbulent Prandtl number, where:

\[
\sigma_\theta = \frac{\phi_h}{\phi_m} = \begin{cases} 0.74 & \text{if } Ri_\theta \geq 0 \\ 0.74(1 - 15Ri_\theta)^{-1/4} & \text{if } Ri_\theta < 0 \end{cases} \tag{8}
\]
$R_i_g$ is the gradient Richardson number $R_i_g = -B/P$. $R_i_g$ is a modified Gradient Richardson number [2]:

$$R_i_g = -\frac{B}{P + \left[\frac{a_B}{\sigma_B}\right]}$$  \hspace{1cm} (9)

$R_i_g$ is introduced by [2] in order to avoid singularities for small turbulence productions $P$. The coefficient $a_B$ is set to achieve equilibrium between turbulent kinetic energy production by shear $P$ and dissipation $\epsilon$ in the stable case, and between buoyancy production $B$ and dissipation $\epsilon$ in the unstable case [1]:

$$a_B = \begin{cases} 
(1 - \frac{l}{l_{max}}) & \text{for } R_i_g \geq 0 \\
1 - \left(1 + \frac{c_{e2}}{c_{e1}}\right) \frac{l}{l_{max}} & \text{for } R_i_g < 0 
\end{cases}$$  \hspace{1cm} (10)

There are two main differences to [1] and [2]: First, the equations are solved in a steady context. Second, a simple tabular relation between the maximum turbulent length scales $l_{max}$ and stability class is used, instead of estimating $l_{max}$ from the location of the geometric center of the turbulent kinetic energy along the vertical direction, as proposed by [7]. This way, the computational effort is reduced.

shows these relationships, which were interpolated from [8].

**Table 1.** Tabular relation between Monin-Obukhov length $L$ and maximum turbulent length scale $l_{max}$ (interpolated from [8]).

| Monin-Obukhov Length $L$ [m] | -50 | -60 | -80 | -130 | $\infty$ | 660 | 350 | 230 | 160 | 110 | 80 | 70 |
|-----------------------------|-----|-----|-----|------|--------|-----|-----|-----|-----|-----|-----|-----|
| Max. length scale $l_{max}$ [m] | 200 | 160 | 120 | 81 | 41 | 35 | 30 | 25 | 21 | 16 | 13 | 10 |

The relevant model coefficients are listed in **Table 2**. Note that $a = 3$ is chosen in equation (5) in order to achieve a better match with the log profile shape close to the ground for the neutral case.

**Table 2.** Coefficients of the $k-\epsilon$ turbulence model.

| Coefficient | $C_{\epsilon1}$ | $C_{\epsilon2}$ | $\kappa$ | $C_\mu$ | $\sigma_\epsilon$ | $\sigma_k$ | $a$ |
|-------------|----------------|----------------|---------|--------|-----------------|-----------|-----|
| value       | 1.44            | 1.92            | 0.4     | 0.09   | 1.3             | 1         | 3   |

2.2. Boundary conditions

For the top domain boundary, a zero gradient boundary condition is used for all variables except pressure and potential temperature. For the latter, a constant value of 300K is applied. At the side boundaries of the circular domain the precomputed numerical profiles are used as inflow condition, if the flow direction is pointing inside the domain. If the flow direction is pointing outside the domain, a zero gradient condition is used. This way a unique boundary condition is applied for the circular part of the domain, automatically taking care of the twisted velocity profiles due to the Ekman spiral. On the terrain surface, the OpenFoam wall function `nutkAtmRoughWallFunction` is used to compute the shear stress at the wall. For the turbulent dissipation rate $\epsilon$ the function proposed by [9] is applied:

$$\epsilon = \frac{C_0^{0.73} k^{1.5}}{\kappa(z+z_0)}$$  \hspace{1cm} (11)
$z_0$ is the aerodynamic roughness length. Regarding the potential temperature $\theta$, a logarithmic distribution of the temperature profile is assumed between the ground and the first cell center adjacent the wall, according to [10]. This assumption is enforced by setting the turbulent heat transfer coefficient $\alpha_t$ to:

$$\alpha_t = \frac{\rho u^+ \kappa}{\sigma_\theta \ln \left( \frac{H_{res}}{z_0} \right)}$$  \hspace{1cm} (12)

### 2.3. Generation of the inflow profiles

In order to obtain a homogenous solution of the governing equations with OpenFOAM, the flow in a rectangular box is simulated. The grid with 3 cells in the homogeneous $x$ and $y$ directions and 160 cells in wall normal direction is used. The first cell has a height of 3 m. The domain height is 6000 m. Cyclic conditions are imposed at all 4 vertical side patches. The boundary conditions described in the previous section are applied at the bottom and top boundary. Since only a quasi-steady state can be achieved for stratified cases, a suitable termination condition is used, and the resulting profiles are collected in a library.

### 3. Validation of the homogenous profiles

In order to validate the homogenous profiles different steps are performed: First it is checked that the profile shapes are maintained over flat terrain, simultaneously ensuring independency to domain size. After that, the homogeneous solutions close to the ground are compared to Businger profiles [11]. This is done in order to verify that the relationship between the fluxes and the gradients close to the ground are in accordance with observations. Profile shapes at higher altitudes are compared against the Høvsøre meteorological mast data [12] and Leipzig profiles [13].

#### 3.1. Preservation of the homogenous profiles

In order to confirm the preservation of the pre-computed homogenous profiles over flat terrain, the flow in a round domain with a radius Rout=10 km is simulated. The resolution in the inner square is $H_{res} = 25$ m, with a ground cell height of $V_{res} = 3$ m, resulting in a mesh size of 1.2 Mio cells. Figure 2 shows the evolution of velocity (SU: speed-up) and turbulence intensity (TI) over the streamwise domain extent for $z_0 = 0.05$ m at various heights and heat fluxes. It is obvious that the velocity magnitude does not change more than 1.5%, and the turbulence intensity not more than 4% between the domain inlet and the outlet for these two extreme cases. For all remaining heat fluxes, the variation of the velocity magnitude and the turbulence intensity are less. The slight increase of the shear in the stable case (Figure 2 a) can be explained as follows: Since the procedure does not obtain a fully steady state, the flow close to the ground keeps cooling when traversing the domain. This leads to a diminishing of the turbulence, hence to an increase of shear. The unstable case shows an increase of velocity close to the ground until the mid of the domain (Figure 2 b). Through the increased turbulent mixing –induced by the heating of the ground– higher momentum fluid can reach the ground from the bulk. This causes a slight acceleration of the flow close to the ground. For the turbulence intensity the interpretation is not so clear.
3.2. Comparison with Monin-Obukhov similarity theory

Figure 3 shows the simulated dimensionless velocity gradients $\Phi_m = \frac{\kappa z}{u_*} \frac{\partial U}{\partial z}$ as a function of the Monin-Obukhov length $L = \frac{-u_*^3 \theta_0}{\kappa g} q$ for $z_0 = 0.05$ m and $0.03$ m. $u_*, \kappa, \theta_0, g, q$ and $\frac{\partial U}{\partial z}$ are the friction velocity at the ground, the von Kármán constant, the potential temperature at the ground, the gravitational acceleration, the heat flux at the ground and the derivative of the velocity with respect to the coordinate normal to the ground. The dimensionless gradients match the experiments reasonably well for heights close to the ground. The gradients are increasing with increasing stability, and decreasing with decreasing stability. This is in agreement with the Monin-Obukhov similarity theory (MOST) which is based on the measurements of [11].
3.3. Comparison with tall measurement towers

Figure 4 (a-c) shows the comparison of the simulated wind profiles ($z_0 = 0.05$ m), normalized with friction velocity at 10 m above ground, to the experiments of [12], and to the Leipzig data [13] (for $z_0 = 0.3$ m). It is obvious that the match with the experiments of [12] is good for the unstable case, and reasonable for the neutral case. Regarding the stable case, the simulated profiles have a considerably lower shear compared with the experiments. A possible explanation for the differences could be the assumption of height-independent geostrophic wind in the simulations (see e.g. [12]). Additionally, the current methodology is based on a steady-state approach. Unsteady phenomena, like high sheared profiles associated with nocturnal inertial oscillations (low level jets), cannot be captured using a steady approach. However, the change of the shear with stability is clearly visible. Since the present methodology will be used to calibrate the simulation against measurements, the current results are acceptable from an industrial point of view. On the other hand, the comparison with the Leipzig data is good. The veer of the wind with height is particularly well captured.

Figure 4. Comparison of homogenous simulation results to measurements. From left to right (a-c): Most stable, neutral and most unstable cases, compared to Høvsøre mast data [12]. Most right (d): comparison to the Leipzig velocity profile [13].
4. Application in real terrain

As first test case in a real terrain, the well-known Askervein hill experiment was chosen (see [16] for a detailed description of the site). The main focus of this test case is the determination of the optimal resolution for the application of the CFD-methodology on real project sites. Furthermore, thanks to the detailed measurements, a general picture of the model prediction quality can be obtained. Four simulations with different horizontal resolutions (\(H_{\text{res}} = 50\,\text{m}, 25\,\text{m}, 15\,\text{m} \,\text{and} \,10\,\text{m}\)) and a ground cell height of \(v_{\text{res}} = 3\,\text{m}\) are performed. Based on the results of previously performed simulations of a 3D hill [15] (not shown here), \(V_{\text{res}} = 3\,\text{m}\) is assumed to be small enough to resolve the possible recirculation region behind the hill. The analysed resolutions correspond to 1.0 Mio, 3.3 Mio, 8.0 Mio and 16.0 Mio cells, respectively. Note that if not specified otherwise, all velocities are scaled with the velocity at the reference location \(RS\) at 10 m (see [16] for the details).

Following conclusions can be drawn from the results shown in Figure 5: The overall agreement between the simulations and the measurements is very good regarding all measured quantities and positions along and normal to the hill. The agreement with the reference site \(RS\) is very good close to the ground for this neutral run (Figure 5a). For altitudes above 100 m the shear is under predicted. The turbulence intensity at \(RS\) is well captured. Very good agreement with the measurements could be achieved regarding the fractional speed-up along line A (Figure 5b).

Minor differences between the different resolutions can be observed in the lee side of the hill. For increasing mesh resolutions the flow at the lee side of the hill is stronger decelerated. A possible explanation for this is the inherent reduction of the numerical diffusion for higher resolutions, which leads to a reduced momentum exchange between the bulk flow and the flow close to the surface. Thus, the flow close to the surface is more affected by the adverse pressure gradient. The dependency of the turbulence intensity (Figure 5d) on the resolution supports this analysis: The closer the flow is to a separated condition, the higher is the turbulence intensity in the region affected by the negative pressure gradient.

The speed-up at the hill top \(HT\) is captured well, whereas the turbulent intensity is over predicted (Figure 5c). The simulated speed-up profile shows a reduction with height, which is typical for flows over smooth obstacles. Regarding the wind direction deviation along the line A (Figure 5f), the resolution has a big influence in the lee side of the hill. This is in line with the observation made regarding the turbulent intensity and the fractional speed-up: For increasing resolution the flow tends to separate more easily in regions of negative pressure, and differences between the solutions are more pronounced. The differences in fractional speedup along line AA (Figure 5g) are small for the resolutions studied, and the agreement with the measurements is good. Note that the results of 15 m resolution and 10 m are nearly identical and hence, grid independency can be assumed.

The answer to the initial question of optimal mesh resolution depends on the zones of interest: If the focus lies only on regions with attached flow, already a simulation with 50 m ground resolution gives good results compared to measurements. If the focus lies on the correct prediction of separated regions, a finer resolution of 15 m would be more appropriate.
Figure 5. a) Scaled velocity and turbulence intensity at upstream reference location RS, b) fractional speed-up, c) wind flow inclination and d) turbulence intensity along line A perpendicular to the hill, e) speed up and turbulence intensity at HT, f) wind direction deviation along line A, g) fractional speed up along line AA; h) Contour map of the Askervein Hill with sampling line definition.

5. Conclusions
A new industrial methodology for CFD based site assessment has been presented. It relies on a steady two equation RANS model, which accounts for the influence of buoyancy only in the turbulence equations. The methodology is carefully validated by means of test cases with increasing complexity. When comparing to experimental data following conclusions can be drawn:
• Although only a quasi-steady state is reached for the stratified cases, the solution (vertical profiles) can be maintained reasonably well in a round domain with a diameter of 20 km.
• The shear of the 1D profiles increases with increasing stability and decreases with decreasing stability, which is in accordance to MOST theory and benchmark observations.
• The profiles reproduce the experiments at a tall measurement tower reasonably well for the neutral and unstable cases. For the stable profiles the shear is too low.
• Good agreement with the Askervein hill experiments is found.

Concluding from the comparison with experiments, the authors are confident to have developed a reliable industrial CFD solution for site assessment. Not demonstrated due to limited space, the present model can also be calibrated against mast measurements on real sites, which reduces the prediction error compared to the uncalibrated model. The exact results uncertainty has yet to be evaluated on a statistically significant number of test cases, which is in preparation in-house (not disclosed here). Regarding the application in an industrial environment, the present methodology opens a variety of new possibilities in order to increase the reliability of the CFD-based site assessment:
• Thanks to the ability of the present methodology to calibrate the simulations in complex terrains, now it is possible to quantify the uncertainty associated with the CFD-based site assessment (of course after having a sufficiently large amount of test cases). It is intended to complement this paper with a benchmark with other state of the arte site assessment tools in the near future.
• One of the big advantages of having an in-house solution, is that the present methodology can be enhanced in the future by unsteady RANS, diurnal cycles or vortex resolving methods like large eddy simulation or detached eddy simulations, in order to reduce the uncertainties to a desired minimum.
• The ability to perform park simulations, i.e. include the effects of turbine wakes, can be easily achieved.
• Potentially risky turbine positions can easily be identified and visualised, therefore lowering the risk of turbine damage already in the planning phase.

The investment in the development of the in-house CFD solver E-Wind for site assessment improves Enercon’s market position, by increasing the planning quality and lowering the risks thanks to a better uncertainty quantification. E-Wind also constitutes a powerful tool for root-cause analysis.
Bibliography

[1] A. Sogachev, M. Kelly und M. Y. Leclerc, „Consistent Two-Equation Closure Modelling for Atmospheric Research: Buoyancy and Vegetation Implementations,“ *Boundary-Layer Meteorology*, Bd. 145, pp. 307-327, 2012.

[2] T. Koblitzi, A. Bechmann, A. Sogachev, N. Sørensen und P. E. Réthoré, „Computational Fluid Dynamics model of stratified atmospheric boundary-layer flow,“ *Wind Energy*, Nr. 18, pp. 75-89, 2015.

[3] J. Franke, A. Hellsten, H. Schlünzen und B. Carissimo, „Franke, Jörg, ed. Best practice guideline for the CFD simulation of flows in the urban environment,“ Meteorological Inst., Hamburg, 2007.

[4] A. Sogachev und O. Panferov, „Modification of two-equation models to account for plant drag,“ *Boundary-Layer Meteorology*, Bd. 121, p. 229–266, 2006.

[5] R. Chávez Arroyo, J. Sanz Rodrigo und P. Gankarski, „Modelling of atmospheric boundary-layer flow in complex terrain with different forest parametrizations,“ *Journal of Physics: Conference Series* 524, 2014.

[6] D. D. Apsley und I. P. Castro, „A LIMITED-LENGTH-SCALE k-EPSONLON MODEL FOR THE NEUTRAL AND STABLY-STRATIFIED ATMOSPHERIC BOUNDARY LAYER,“ *Boundary-Layer Meteorology*, pp. 75-98, 1997.

[7] G. L. Mellor und T. Yamada, „Development of a turbulence closure model for geophysical fluid problems,“ *REVIEWS OF GEOPHYSICS AND SPACE PHYSICS*, pp. 851-875, 1982.

[8] A. Peña Díaz, S. Gryning, C. Hasager und M. Courtney, „Extending the wind profile much higher than the surface layer,“ *EWEC 2009 Proceedings online*, 2009.

[9] A. Parente, C. Gorle, J. van Beeck und C. Benocci, „Improved k–e model and wall function formulation for the RANS simulation of ABL flows,“ *J. Wind Eng. Ind. Aerodyn.*, 2011.

[10] D. Etling, Theoretische Meteorologie - Eine Einführung, 3. Hrsg., Berlin, Heidelberg: Springer, 2002.

[11] J. A. Businger, J. C. Wyngaard, Y. Izumi und E. F. Bradley, „Flux-profile relationships in the atmospheric surface layer,“ *J. Atmos. Sci.*, pp. 181-189, 1971.

[12] A. Peña, F. Rogier und S. E. Gryning, „The Høvsøre Tall Wind-Profile Experiment: A Description of Wind Profile Observations in the Atmospheric Boundary Layer,“ *Boundary-Layer Meteorology*, 2014.

[13] H. Lettau, „A re-examination of the "Leipzig Wind Profile" considering some relations between wind and turbulence in the frictional layer,“ *Tellus*, pp. 125-129, 1950.

[14] J. Arqvist, A. Segalini, E. Dellwik und H. Bergström, „Wind Statistics from a Forested Landscape,“ *Boundary-Layer Meteorology*, Bd. 156, Nr. 1, pp. 53-71, 2015.

[15] T. Ishihara, K. Hibi und S. Oikawa, „A wind tunnel study of turbulent flow over a three-dimensional steep hill,“ *Journal of Wind Engineering*, pp. 95-107, 1999.

[16] P. Taylor und H. Teunissen, „Askervein ’82: report on the September/October 1982 experiment to study boundary layer flow over Askervein, South Uist. Technical Report MSRS-83-8, Meteorological Services Research Branch,“ Atmospheric Environment Service, Downsview, Ontario, Canada, 1982.