# GT2011-(\*&(\$

# TERRAIN EFFECTS ON WIND FLOW: SIMULATIONS WITH AN IMMERSED BOUNDARY METHOD

# S. Jafari\*, N. Chokani, R.S. Abhari

Laboratory for Energy Conversion Department of Mechanical and Process Engineering ETH Zurich **Zurich, Switzerland** 

\*Email: jafari@lec.mavt.ethz.ch

#### ABSTRACT

The modelling of the wind resource over arbitrary topography is required to optimize the micrositing of wind turbines. Most solvers use classical body-fitted grid for simulations. In such an approach, to cover the wind rose using a rectangular domain, a dedicated mesh must be generated for each direction. Moreover, over complex terrain, additional numerical errors are introduced in the solver due to coordinate transformations. To overcome these challenges and to facilitate the grid generation process, an immersed boundary method is developed in connection with a RANS solver in order to simulate turbulent atmospheric flows over arbitrary topography. In this method, a Cartesian grid is used and the boundary condition on the terrain surface is modelled within the solver using a "direct forcing" approach. With the immersed boundary method a rectangular grid can be used to simulate the flow field for all wind directions and only a rotation of the digital elevation map is required. Ghost cells are used to enforce the desired boundary condition at the immersed surface. The immersed boundary method developed in this work is used to simulate the flow in connection with both Baldwin-Lomax and kw turbulence models. The performance of the method is examined for the flow over a two-dimensional hill. Results are compared with experimental data as well as a classical body-fitted grid to isolate the effect of the boundary conditions. The comparisons show good agreement among all the results. The results for the three-dimensional wind flow simulation over the Askervein Hill test case are also presented, and show the capability of the immersed boundary method in a full-scale scenario.

#### NOMENCLATURE

λ	interpolation constant
$\mu_{\mathrm{T}}$	turbulent eddy viscosity
σ	speed-up
τ	viscous shear tensor
ω	specific dissipation rate
Κ	turbulent kinetic energy
р	pressure
Q	flow vector
Re	Reynolds number
Т	temperature
U	velocity vector
Х	position vector
у	normal distance from wall
$y^+$	non-dimensional distance from wall

Subscripts

g	ghost node
IBC	immersed boundary condition
m	mirror point
S	surface point

# Abbreviations

СР	centre point
GA	grid aligned with geometry
HT	hilltop
IBM	immersed boundary method
OpenMP	open multi-processing

RANS

# INTRODUCTION

It is well known that atmospheric boundary layer flow is significantly influenced by the underlying topography. In order to assess the wind resource for planning and analysis for wind farms, measurements and/or the simulation of atmospheric flow over specific terrain are required [1-3]. In the case of gentle topography and low hills, analytical solutions and simple linear models that yield predictions with acceptable accuracy have been developed [4]. These models have a low computational cost and can be used to obtain accurate flow predictions over smooth hills with sufficiently gentle slopes and attached flows [5]. However, due to simplified assumptions in their formulation, linear models fail to predict the separation region downstream of steep hills. Some empirical relationships have been added to the models to improve their accuracy in separated flow regions, but these relations are often sitespecific and computational time is also significantly increased. With the increase in computational resources and major developments in computational fluid dynamics, there has been an increased focus on the application of non-linear models for wind resource assessment [6]. Raithby et al. pioneered the use of three-dimensional CFD in atmospheric flow simulations over Askervein Hill [7]. Since then, RANS methods with second order closure turbulence models have been used for atmospheric flows over moderately sloped and complex terrains by several authors [8-11]. More detailed modelling of turbulence is possible using large eddy simulations (LES) applied to two-dimensional hills and relatively simple flows [12-13]. Large eddy simulations have also been used for threedimensional cases including Askervein Hill [14]. However, recent results of the Bolund Hill complex terrain test case show that there are still challenges in achieving sufficiently accurate predictions using large eddy simulations [15].

Even though the capability of RANS models has been demonstrated for atmospheric flows, to achieve sufficient accuracy in wind prediction, further improvements are required in several areas including turbulence modelling, surface roughness, boundary conditions and grid generation. This work looks at grid generation. Most of the current flow solvers employ structured or unstructured body-fitted grids over the specified topography. Since most of the solvers use finite difference or finite-volume methods with relatively low (that is first or second-order) accuracy, fine meshes are required to obtain good results. Apart from the fact that some solvers are not capable of handling complex geometries, the grid generation process often requires significant computational time to achieve a satisfactory balance between the desired grid size and the skewness of the grid cells. The latter is more problematic over steep landscapes. Moreover, most algorithms use a terrain following coordinate transformation, which introduces additional numerical error in the solver when used over very steep terrain. In addition to the above considerations, in wind resource assessment for the micrositing of wind turbines, multiple wind directions must be investigated. In order to cover a wind rose of interest using conventional rectangular domains, grids must be generated for each wind direction. This is a lengthy and tiresome procedure. On the other hand, the use of one circular computational domain amplifies the uncertainties in the definition of inflow boundary conditions. To overcome these problems, in this study an immersed boundary method has been used in a RANS solver in order to model the flow over any arbitrary topography using a single Cartesian grid. The method does not incur significant additional costs in addition to the basic computational costs. Furthermore, changes in surface geometry simply require modification of the orientation of the topography without any further modification of the code or the grid.

The immersed boundary method (IBM) was first introduced by Peskin for low Reynolds number biological flows [16]. However, its application was later successfully extended to simulate the flow over arbitrary complex geometries using RANS solvers, large eddy or direct numerical simulations [17-19]. In the IBM the presence of the surface is modelled by an external body force acting on the grid nodes in the vicinity of the surface. Peskin [16] modelled the force acting on the flow by moving solid boundaries with a spring system. Later, several authors extended the method by using "feedback forcing" in the momentum equation to set the desired boundary condition [20]. However, in this approach, the computational cost is significantly increased for transient flows [21]. To overcome this problem, the "direct forcing" approach was introduced. In this method the forcing function due to the immersed surface is included in the equations solved using ghost nodes [22]. This approach is used here. In addition to higher stability, the advantage of this method is that the main part of the solution algorithm is calculated only once for a specific simulation and remains unchanged during the remainder of the computation. This significantly decreases computational overheads. While the immersed boundary method has been applied to wind predictions using LES [23], in this study the implementation of the method with a RANS solver and its application to wind flow over two and threedimensional terrains are presented for the first time.

## NUMERICAL METHOD

The computations are done using our in-house code "MULTI3", which is a second order compressible RANS solver. The algorithm is based on an explicit, finite-volume node-based Lax-Wendroff method developed by Ni [24].

The Navier-Stokes equations are discretised using a finite volume method with vertex storage in unstructured hexahedral cells. The finite volume formulation uses a central cell-vertex variable location, meaning that the numerical scheme used in this study is a central scheme and the state variables are stored at the vertices of the computational cell. To prevent highfrequency oscillations, second and fourth order numerical smoothing is added. To speed-up the convergence, a local time stepping approach with a multiple-grid algorithm is used [24]. The eddy viscosity can be obtained using either the zeroequation Baldwin-Lomax, one-equation Spalart-Allmaras, or two-equation k- $\omega$  turbulence models, and is solved separately from the mean flow. The solver has been parallelized using OpenMP and can run parallel in shared memory architecture. More information on the solver can be found in Burdet and Abhari [25].

#### **Immersed boundary method**

As mentioned above, in this study, the "direct forcing" approach is used to simulate the no-slip boundary condition at solid boundaries of the computational domain. In this approach, the solid surface divides the whole computational domain into three regions, (i) physical cells, (ii) interfacial cells and (iii) ghost cells (Figure 1). Ghost cells are "dead" cells within the computational domain; these cells do not carry any meaningful physical values. Instead, the grid nodes in the vicinity of the surface (ghost nodes) are used to impose the desired boundary condition at the surface. With the known location of ghost nodes, each ghost node is linked to the immersed surface through a surface point, X<sub>s</sub>. The surface point located on the virtual surface is the point on the virtual surface that is closest to the ghost node. The normal line connecting the ghost nodes and surface points also specifies the mirror points that are located within the physical domain (see Equation 1). Note that  $\lambda = 1$  for a linear implementation.

$$X_m = \frac{(1+\lambda)X_s - X_g}{\lambda} \tag{1}$$

The overall idea of the immersed boundary method is to impose the proper flow field on  $X_g$  based on the flow vector on  $X_m$ , so that the desired boundary condition on the immersed surface,  $X_s$ , is fulfilled. Since the mirror points are not necessarily located at computational nodes, the full immersion of the boundary condition necessitates the interpolation of the flow field to corresponding points within the physical domain. Here a weighted average interpolation scheme is used for this purpose. After determining the flow vector at mirror point,  $Q_m$ , the flow vector at ghost node  $Q_g$  is specified to set the required no-slip condition of  $Q_s$  at the surface. If the immersed boundary condition is of a Dirichlet type, the specified flow field is set at the ghost nodes accordingly. For a no-slip boundary condition with linear interpolation, Equation 2 gives the flow vector at the ghost node:

$$Q_{g} = (1 + \lambda)Q_{IBC} - \lambda Q_{m} \qquad (2),$$

where for the flow vector  $\boldsymbol{Q}_{\text{IBC}}$  the velocity is zero.

Furthermore, a Neumann boundary condition may be required to specify the gradients at the wall. This boundary condition is given as following:

$$Q_{IBC} = \frac{\partial Q}{\partial n} \longrightarrow Q_g = \lambda Q_m - (1 + \lambda) Q_{IBC} (X_s - X_g)$$
(3)

which can be reduced to Equation 4 for the pressure at the noslip wall.

$$Q_g = Q_m \tag{4}$$

In order to initiate a search for ghost nodes in the vicinity of the immersed surface, it is necessary to define the entire immersed surface. The topographical data is read as discrete points from a digital elevation map (DEM) model. Bicubic interpolation is used to specify the entire terrain.



**Figure 1.** Schematic diagram of the computational stencil used for the implicit IBM. The immersed surface (shown as a red line) divides the domain into physical, interfacial and ghost cells.



Figure 2. Illustration of the implicit boundary condition at the point  $X_s$ , which is on the immersed boundary. The computed flow at the mirror point  $X_m$  is used to specify the flow at the ghost point  $X_g$  such that the boundary condition is implicitly fulfilled on the immersed surface.

The search for mirror and surface nodes is undertaken once at the beginning of the computation. A gradient descending algorithm is used to find the surface points that have the minimum distance to the ghost node (Figure 2). Depending on the initial guess, as few as 10 iterations are required. In general, the computational time required for the geometric calculations is less than 1% of the overall computational time. Similar to classic boundary conditions, the interpolation of the flow field at mirror points and the specification of the desired boundary condition at ghost nodes is undertaken at every iteration. The accuracy of the method for the basic solver is examined for a low Reynolds number (laminar) flow around a circular cylinder. Figure 3 shows the comparison between the surface pressure coefficient calculated using the present approach and experimental measurements [26]. Velocity streamlines are also shown in Figure 3. The agreement between the pressure coefficient predicted by IBM and the experimental data is quite good. The separation occurs at  $\theta$ =129.8° compared to a measured value from Coutanceau and Bouard [27] equal to 126.2°. The implementation of the IBM for the vertex base solver, separate from turbulence modelling is validated through this case.



**Figure 3.** Simulation of Reynolds number Re=40 flow over a circular cylinder. Upper plot: comparison of IBM solution to experimental results. Lower plot: streamlines superimposed on pressure contours.

#### **Turbulence models**

The immersed boundary method developed in the present work can be used with either Baldwin-Lomax or k- $\omega$  turbulence models. More complete details of these models can be found in [28] and [29] respectively, but salient features of the models with regard to the implementation of IBM are given below. In the two-layer algebraic Baldwin-Lomax turbulence model, the turbulent eddy viscosity is given by:

$$\mu_T = \begin{cases} \mu_{T_{inner}} &, y \le y_m \\ \mu_{T_{outer}} &, y \succ y_m \end{cases}$$
(3)

where y is the normal distance from the wall and  $y_m$  is the distance at which the inner and outer eddy viscosity are equal. Inner layer eddy viscosity is calculated from Prandtl's mixing length theory and depends on the absolute value of the vorticity and the mixing length. The characteristic mixing length is defined as below as:

$$l_{mix} = 0.4y[1 - \exp(-\frac{y^{+}}{26})]$$
(4)

where the non-dimensionalized distance  $y^+$  is given by Equation 5.

$$y^{+} = \frac{\rho_{wall} u_{\tau} y}{\mu_{wall}}$$
(5)

In computations using body-fitted grids, y is defined as the minimum distance between a grid point and the closest point on the terrain (no-slip surface). On the other hand, in the IBM implementation the minimum distance to the immersed surface must be used. The minimum distance is determined either as the distance between the node within the domain and all surface points or using the minimum of distance function, the algorithm used for the calculation of the surface points. The wall shear is estimated using the computed velocities at the mirror points (Equation 6).

$$u_{\tau_{IBM}} = \sqrt{\frac{\mu(U_m - U_s)}{\rho_s(X_m - X_s)}} \tag{6}$$

In the k- $\omega$  turbulence model, transport equations are used to solve the turbulent kinetic energy, k and the specific dissipation rate,  $\omega$  (Equation 7).

$$\frac{\partial(\rho u_i k)}{\partial x_i} = \frac{\partial}{\partial x_i} \left[ \left( \mu + \frac{\mu_T}{\sigma_w} \right) \frac{\partial \omega}{\partial x_i} \right] + P_k - \rho k \omega$$
$$\frac{\partial(\rho u_i \omega)}{\partial x_i} = \frac{\partial}{\partial x_i} \left[ \left( \mu + \frac{\mu_T}{\sigma_w} \right) \frac{\partial \omega}{\partial x_i} \right] + \frac{\omega}{k} P_k - \beta \rho \omega^2 \qquad (7)$$

The eddy viscosity is then given as:

$$\mu_T = \rho \frac{k}{\omega} \tag{8}$$

The boundary condition for k is:

$$k = 0 \tag{9}$$

and is applied at a grid point located on the wall for the classical body-fitted grid and at ghost nodes for the IBM implementation. For  $\omega$ , the boundary condition follows Menter [30],

$$\omega = \frac{60v}{\beta_1 {y_0}^2} \tag{10}$$

where  $\beta_1 = 0.075$ ,  $y_0$  is the distance of the first node above the wall for the classical body-conformal grid and for the IBM implementation, either the distance between the mirror and surface points, maximum value of the prescribed quantity or the grid spacing in the direction normal to the immersed wall. The best results over several test cases were obtained using the latter approach.  $\mu_T$  is set to zero for all nodes below the immersed surface for both turbulence models. It should be noted that the k and  $\omega$  behaviour close to the boundary is nonlinear. Hence, the accuracy of the implemented boundary conditions is strongly dependent on the grid resolution. This dependency can be substantially reduced using wall functions. A detailed description of the wall function implementation in connection with IBM is given in Jafari et al. [31]. A sand-grain equivalent roughness height is used in the wall functions to account for surface roughness.

#### **RESULTS AND DISCUSSIONS**

In the following the performance of the immersed boundary method is first examined in the flow over a twodimensional hill. Results from IBM are also compared with those from a body-fitted grid to isolate the effect of wall boundary condition on the results. A three-dimensional test case of flow over Askervein Hill is also examined. The 2D test case is the sinusoidal hill defined by Kim and Lee [32], schematically shown in Figure 4. The chord length and the height of the curved hill are 0.46 meters and 0.07 meters respectively. The unit Reynolds number based on the free stream velocity ( $U_{\infty}=7$  m/s) is  $7.5 \times 10^5$ /meters. For simulations with a classical body-fitted grid, an H-grid with clustering in the x and z directions is used (Figure 5). Also shown in Figure 5 is the corresponding Cartesian grid for the IBM simulations.



**Figure 4.** Definition of the geometry for the 2D hill, *L* is the upwind half-length of the hill at the one-half height of the hill.



**Figure 5.** Typical H-grid (upper plot) and non-uniform Cartesian grid (lower plot) used for the grid-aligned with flow and immersed boundary method simulations, respectively.

For the simulations the upstream boundary is 3C upstream of the windward foot of the hill. The inflow streamwise velocity is a fully developed turbulent boundary layer with a thickness of 0.25 m. The outflow boundary is located 5C downstream of the leeward foot of the hill.

A quantitative assessment of the immersed boundary method is given in Figures 6 and 7. With the k- $\omega$  turbulence model, the predicted pressure coefficients with the immersed boundary method and body-conformal grid are in excellent agreement with each other, as well as with the experiment, Figure 6. However, the predictions of the Baldwin-Lomax turbulence model are in poor agreement with the experiment at the leeward corner of the hill. Numerical results show that there is a separated flow in this region, which results in an overprediction of the minimum pressure at the hillcrest. However, for the Baldwin-Lomax turbulence model, a comparison of IBM and GA results shows the maximum difference observed for the pressure coefficient is less than 0.03 and they are in fairly good agreement. Profiles of the non-dimensionalized velocity speed-ups defined by Equation 5

$$\sigma = \frac{U(z) - U_0(z)}{H/L U_{\infty}}$$
(5)

at the hilltop are presented in Figure 7. Good agreement is observed between the IBM and the body-fitted grid for both turbulence models in the outer region (z/L>0.2) of the boundary layer. Small discrepancies are observed closer to the wall; these discrepancies are attributed to differences between the grids and the resolution of the boundary layer. Overall, it is seen that both the pressure and velocity speed-up are better predicted using the k- $\omega$  turbulence model. In general, the results demonstrate that the immersed boundary technique is capable of resolving the flow field in high Reynolds number flows.



Figure 7. Comparison of the non-dimensionalized speed-ups at the hilltop.

Figure 8 shows a comparison between the pressure coefficient and the non-dimensionalized axial velocity over the hill with IBM and the body-fitted grid. The overall qualitative agreement between the IBM and the solution obtained with the classical body-fitted grid is quite satisfactory.



**Figure 8.** Comparison of axial velocity (pair of upper plots) and pressure coefficient (pair of lower plots) over the two-dimensional hill in grid-aligned with flow (upper of pair) and immersed boundary method (lower of pair) simulations.



Figure 9. Digital elevation maps of Askervein Hill. Note that the z direction is stretched by a factor 10 compared to the x and y directions. Lower plot shows the actual map of Askervein Hill with its surrounding topography (map A). Upper plot shows map B which omits the elevated topographic features that are downwind of Askervein Hill.

The flow over a three-dimensional hill is also investigated using the newly developed method. The Askervein Hill test case is chosen for this purpose. Askervein Hill is an isolated hill on the Isle of Uist in Scotland. A detailed survey is reported in Taylor et al. [33]. These detailed measurements have been used by various researchers to evaluate linear and non-linear models. Three kilometres to the west of the hill is the North Atlantic Ocean. The predominant winds from the south-west (SW) and south-south-west (SSW) are therefore uninfluenced by anthropological or natural obstacles. This is very important in the accurate definition of the inflow in the computations. As is shown in the lower plot of Figure 9 the digital elevation map of the area, downstream of Askervein Hill is a hill with a similar elevation. Another hill of slightly higher altitude is located to the east. There are two maps available for Askervein Hill. Map A with 63 m resolution which also includes the surrounding topography and map B covering only the isolated hill with 23 m resolution. Other than the resolution, there are few differences between the map A and B including the elevation of the hilltop, the elevation along line B-B and the topography downstream of the hilltop. Kim et al. [10] suggest that this topography has an effect on the downstream flow; but several authors such as Castro et al. [8] only focus on the isolated hill. In the current study, simulations are done using both maps (Figure 9).

For grid generation, as mentioned above, digital elevation maps of the terrain are used. The maps are rotated by 60 degrees, so that the flow direction is collinear to the x-axis going through the centre point (CP), Figure 10. The nonuniform Cartesian grid has a resolution of 16m in the x direction and 20m in the y direction for the clustered region defined by an area spanning 700m by 500m around the centre between CP and HT. The size of the entire domain is 8,000m by 8,000m in x and y directions for map A and 6,000m by 6000m for map B; for both maps the domain is 1,000m in the z direction and the cell height is 3.0m for the entire hill. These dimensions result in a total of 0.6 million computational nodes. The velocity and turbulent kinetic energy inflow profiles are specified following Kim et al. [10]. The roughness height is set to 0.03 m for the entire terrain. The available experimental data include measurements of the speed-ups along the lines A-A, AA-AA, B-B and at the hilltop (HT) as shown in Figure 10.



**Figure 10.** Askervein Hill topography map superposed with the nonuniform Cartesian grid used for the simulations (map B). The grid is clustered around the hilltop (HT) and the centre point (CP) of the computational domain. Also shown is the orientation of the lines (A-A, AA-AA and B-B) along which measurements are available, are also shown.



Figure 11. Comparison of the predicted and measured nondimensionalized speed-ups along line A-A on Askervein Hill. The predictions are from simulations using the immersed boundary method.

The predicted speed-ups along lines A-A and AA-AA and B-B are compared to experiment in Figures 11 and 12 and 13. Also shown in the lower portion of the figures is the elevation

change along the respective lines of measurements. The speedup is defined based on the undisturbed velocity upstream of the hill at the reference station. The results are also compared to simulations of Walmsley and Taylor [34] and Undheim et al., [35]. The prediction over line A-A is generally in good agreement with experiment. However, the computed speed-up does not follow the steep decrease of  $\Delta S$  on the lee side of the hill using either map A or B. At x=0, speed-up is 0.69 compared to the 0.86 of the experiment in the simulations for the isolated hill. Including the downwind hills does not affect the flow downstream but changes the maximum speed-up significantly. This difference is thought to be due to uncertainties in defining the digital elevation map, as well as limitations in turbulence and roughness modelling. Local minima are observed at about x=-700, -350 and -200 m resulting from the changes in topography. The latter is more pronounced in the current simulation, but has also been observed by Undheim et al. [35].

The over-prediction of the low velocity in the wake, downstream of the hill, where non-linear effects are dominant, can be due to turbulence modelling uncertainties in the prediction of flow separation.



**Figure 12.** Comparison of the predicted and measured nondimensionalized speed-ups along line AA-AA shown in Figure 10 on Askervein Hill.

The predictions over line AA-AA, Figure 12, match well with the experiment except at x=400m and x=600m, where the speed-up is over predicted. The local minimum at x=-500m in the field measurements due to the change in topography (see the elevation map) is captured correctly with the IBM method. The observed increase in velocity downstream of CP (x=0m) as well as the sharp change in velocity at x=450m are consistent with the elevation change and are also observed in other simulations [10, 35].

The effect of topography on the predicted and measured speed-ups, along line B-B, are compared in Figure 13. The predictions are obtained using map A, and show good qualitative agreement with the measured wind speed-up.



Figure 13. Comparison of the predicted and measured nondimensionalized speed-ups along line B-B on Askervein Hill (map A).



Figure 14. Comparison of the predicted and measured nondimensionalized speed-up at hilltop.

The predicted speed-up at the hilltop is shown in Figure 14 and compared to the simulations of Undheim et al. [35]. Except for region very close to the surface (z<5m), the IBM prediction matches well with Undheim's simulations, as well as with the experimental data. The difference close to the surface can be reduced by using finer grids or a higher order interpolation close to the surface.

The near surface velocity vectors, 5 m above the ground are also shown in Figure 15. In Figure 15, the velocity vectors are superimposed on the contour plot of velocity field. Accelerated and decelerated flow regions can be observed, as expected, at the hilltop and downstream of the hill.

In general, the immersed boundary method is observed to perform well in flowfield predictions for full-scale scenarios. At potential hub heights, where the measured speed-up is than 60%, the predicted speed-up is within 1% of the measurement. Thus it is evident that the immersed boundary method that has been developed in the present work can provide desired accuracy that the wind industry requires for the optimum micrositing of wind turbines.



Figure 15. Near surface velocity vectors superimposed on velocity contours over Askervein Hill. Note that the z direction is stretched by a factor 5 compared to the x and y directions.

#### CONCLUSION

An immersed boundary method has been developed in connection with LEC's RANS solver MULTI3 for high Reynolds number flows. The method works in connection with zero-equation Baldwin-Lomax and two-equation k-ω turbulence models. The current implementation of IBM is based on a ghost node approach assuming a linear variation of the flow vector at the boundary. The implementation of the method into the RANS solver is first validated for a low Reynolds number flow over a circular cylinder. The capability of the model to simulate high Reynolds number flows similar to those of interest in wind applications is also examined. The flow over a two-dimensional hill was studied and the results were then compared to the simulations obtained using a bodyfitted grid to isolate the effect of boundary conditions. Good agreement with experiment demonstrates the capability of the immersed boundary method for high Reynolds number flows. In addition, to show the capability of the model in handling full-scale scenarios, a three-dimensional flow was simulated over Askervein Hill. The general agreement of the predicted speed-up over the hill was good. Using Cartesian grids for wind flow simulations significantly facilitates the grid generation process to cover the whole wind rose. Furthermore, the immersed boundary method enables the solver to handle highly

complex geometries with large gradients and sharp corners with higher stability and accuracy.

# ACKNOWLEDGEMENT

The first author thanks her current and former colleagues at LEC, Christian Fuchs, Altug Basol and Bob Mischo, for their valuable suggestions and helpful discussions.

### REFERENCES

- [1] Salmon, J.R., Teunissen, H.W., Mickle, R.E., and Taylor, P.A., "The Kettles Hill Project: Field Observation, Wind-Tunnel Simulations and Numerical Model Predictions for Flow over a Low Hill," *Boundary-Layer Meteorol.*, Vol. 43, 1988, 309-343.
- [2] Frehlich, R., Meiller, Y., and Jensen, M.L., "Measurements of Boundary Layer Profiles with In Situ Sensors and Doppler Lidar," J. Atmospheric and Oceanic Technology, Vol. 25, 2007, 1328-1340.
- [3] Mansour, M., Kocer, G., Lenherr, C., Chokani, N., and Abhari, R.S., "Seven-Sensor Fast-Response probe for Full-Scale Wind Turbine Flowfield Measurements," J. Engineering for Gas Turbines and Power, Vol. 133, 2011.
- [4] Taylor, P.A., "Some Numerical Studies of Surface Boundary-Layer Flow above Gentle Topography," *Boundary-Layer Meteorol.*, Vol. 12, 1977, 439-465.
- [5] Bowen, A.J., and Mortensen, N.G., "Exploring the Limits of WasP the Wind Atlas Analysis and Application Program," *Proc. of European Wind Energy Conference*, Goteborg, Sweden, 1996.
- [6] Undheim, O., "Microscale Flow Modelling Possibilities and Limitations," *Proc. of European Wind Energy Conference*, Athens, Greece, 2006.
- [7] Raithby, G.D., Stubley, G. D., and Taylor, P.A., "The Askervein Hill Project: A Finite Control Volume Prediction on Three-Dimensional Flows over the Hill," *Boundary-Layer Meteorol.*, Vol. 39, 1987, 107-132.
- [8] Castro, F.A., Palma, J.M.L.M., and Silvia Lopes, A., "Simulation of the Askervein Flow. Part1: Reynolds Averaged Navier-Stokes Equations (k-ε Turbulence Model)," *Boundary-Layer Meteorol.*, Vol. 107, 2003, 501-530.
- [9] Maurizi, A., Palma J.M.L.M., and Castro, F.A., "Numerical Simulation of the Atmospheric Flow in a Mountainous Region of the North of Portugal', *J. Wind Eng. Ind. Aerodyn.*, Vol. 74, 1998, 219-228.
- [10] Kim, H.G., Patel, V.C., and Lee, C.M., "Numerical Simulation of Wind Flow over Hilly Terrain," J. Wind Eng. Ind. Aerodyn., Vol. 87, 2000, 45-60.
- [11] Ferreira, A.D., Lopes, A.M.G., Viegas, D.X., and Sousa, A.C.M., "Experimental and Numerical Simulation of Wind Flow Around Two-Dimensional Hills," *J. Wind Eng. Ind. Aerodyn.*, Vol. 54/55, 1995, 173-181.
- [12] Allen, T., and Brown, A., "Large-Eddy Simulation of Turbulent Separated Flow over Rough Hills," *Boundary-Layer Meteorol.*, Vol. 102, 2002, 177-198.

- [13] Porte-Agel, F., Meneveau, C., and Parlange, M.B., "A Scale-Dependent Dynamic Model for Large-Eddy Simulation: Application to Neutral Atmospheric Boundary Layer," J. Fluid Mech., Vol. 415, 2000, 261-284.
- [14] Silvia Lopes, A., Palma, J.M.L.M., and Castro, F.A., "Simulation of the Askervein Flow. Part2: Large-Eddy Simulations," *Boundary-Layer Meteorol.*, Vol. 125, 2007, 85-108.
- [15] Bechmann, A., Berg, J., Courtney, M.S., Jorgensen, H.E., Mann, J., and Sorensen, N.N., "The Bolund Experiment: Overview and Background," *Technical Report*, Riso-R1658(EN), Riso DTU, 2009.
- [16] Peskin, C.S., "Flow Patterns around Heart Valves: a Numerical Method," J. Comp. Phys., Vol. 10, 1972, 252-271.
- [17] Majumdar, S., Iaccarino G., and Durbin, P., "RANS Solver with Adaptive Structured Boundary Non-Conforming Grids," *Center for Turbulence Research, in Annual Research Briefs*, 353-366, 2001.
- [18] Balaras, E., "Modelling Complex Boundaries Using an External Force Field on a Fixed Cartesian Grid in Large-Eddy Simulations," *Computer and Fluids*, Vol. 33, 2004, 375-404.
- [19] Kang, S., Iaccarino, G., and Ham F., "DNS of a Buoyancydominated Turbulent Flow Using an Immersed Boundary Method," J. Comp. Phys., Vol. 228, 2009, 3189-3208.
- [20] Goldstein, D., Handler R., and Sirovich L., "Modelling a No-slip Flow Boundary with an External Force," J. Comp. Phys., Vol. 105, 1993, 354-366.
- [21] Saiki, E.M., and Biringen S., "Numerical Simulation of a Cylinder in Uniform Flow: Application of a Virtual Boundary Method," J. Comp. Phys., Vol. 123, 1996, 450-465.
- [22] Fadlun, E.A., Verzicco, R., Orlandi, P., and Mohd-Yusof, J., "Combined Immersed-Boundary Finite-Difference Methods for Three-Dimensional Complex Flow Simulations," J. Comp. Phys., Vol. 161, 2000, 35-60.
- [23] Lundquist, K.A., Chow, K.F., and Lundquist J.K., "An Immersed Boundary Method for the Weather Research and Forecasting Model," *Monthly Weather Review*, 2010, Vol. 138, Issue 3, 796-817.
- [24] Ni, R., "A Multiple Grid Scheme for Solving the Euler Equations," *AIAA J.*, Vol. 20, 1981, 1565-1571.
- [25] Burdet, A., and Abhari, R.S., "Three-Dimensional Flow Prediction and Improvement of Holes Arrangement of a Film-Cooled Turbine Blade Using a Feature-Based Jet Model," J. Turbomachinery, Vol. 129, 2007, 258-268.
- [26] Grove, A.S., Shair, F.H., Peterson E.E., and Andreas, M., "An Experimental Investigation of the Steady Separated Flow Past a Circular Cylinder," *J. Fluid Mech.*, Vol. 19, 1964, 60-80.
- [27] Countanceau, M., and Bouard, R., "Experimental Determination of the Main Flow Features of the Viscous Flow in the Wake of Circular Cylinder in Uniform Translation. Part 1: Steady Flow," J. Fluid Mech., Vol. 79, 1977, 231-256

- [28] Baldwin, B.S., and Lomax, H., "Thin-Layer Approximation and Algebraic Model for Separated Turbulent Flows," *AIAA Paper* 78-257, 1978.
- [29] Wilcox, D.C., "Turbulence Modelling for CFD," 1st ed., DCW Industries, La Canada.
- [30] Menter, F.R., "Zonal Two Equation k-ω Turbulence Model Predictions," AIAA Paper 93-2906, 1993.
- [31] Jafari, S., Chokani N., and Abhari, R.S., "An Immersed Boundary Method for Efficient Simulation of Wind Flow over Complex Terrain," *European Wind Energy Conference*, 2011, Brussels, Belgium.
- [32] Kim, H.G., and Lee, C.M., "Pollutant Dispersion Over Two-Dimensional Hilly Terrain," *KSME International Journal*, Vol. 12, 1998, 96-111.
- [33] Taylor, P.A., Mason, P.J., and Bradley, E.F., "Boundary Layer Flow over Low Hills," *Boundary-Layer Meteorol.*, Vol. 39, 1987, 107-132.
- [34] Walmsley, J.L., and Taylor, P.A., "Boundary-Layer Flow over Topography: Impacts on the Askervein Study," *Boundary-Layer Meteorol.*, Vol. 78, 1996, 291-320.
- [35] Undheim, O., Andersson, H.I., and Berge, E., "Non-Linear, Microscale Modelling of the Flow over Askervein Hill," *Boundary-Layer Meteorol.*, Vol. 120, 2006, 477-495.