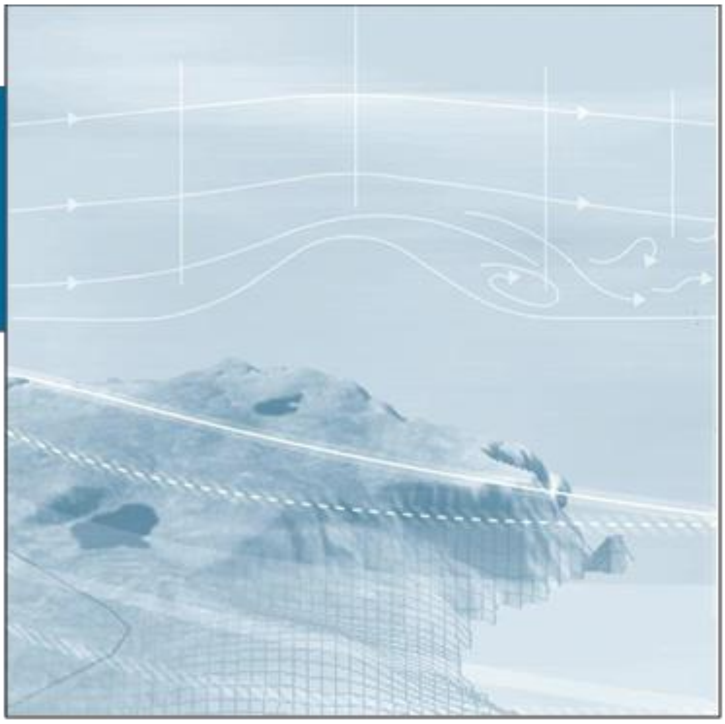
User Handbook

WindSim

WindSim AS
Tollbodgaten
22
N-3111 Tønsberg
Norway
+47 33 38 18 00



WindSim | User Handbook

15th Edition | July 2025

WindSim

Contents

WindSim Desktop 12

About WindSim

Micrositing

WindSim is a modern Wind Farm Design Tool (WFDT). WindSim is used to optimize the wind farm energy production while at the same time keeping the turbine loads within acceptable limits. This is achieved by calculating numerical wind fields over a digitalized terrain. In the wind energy sector this is called micrositing.

By coupling the numerically calculated wind speeds and wind directions against available site-specific climate conditions the optimal position for each turbine can be found. Climate conditions are most frequently given by on-site measurements but could alternatively be derived from meteorological models. WindSim can interface with both of these types of datasets.

The loads on a wind turbine are influenced by wind field characteristics such as; wind shear, inflow angle, and turbulence. Since the wind field modelling is 3D all of these characteristics are calculated and checked to be within acceptable limits for a given turbine type.

The optimization of the energy production and the minimization of the loads could be conflicting processes. The location yielding maximum energy production for a wind turbine could also yield too high loads, and often a compromise must be found: a location with the highest possible energy production still with acceptable loads. Micrositing is an iterative process where various turbine locations and types must be inspected.

Modular Approach

WindSim uses a modular approach with six modules to complete the steps within micrositing.

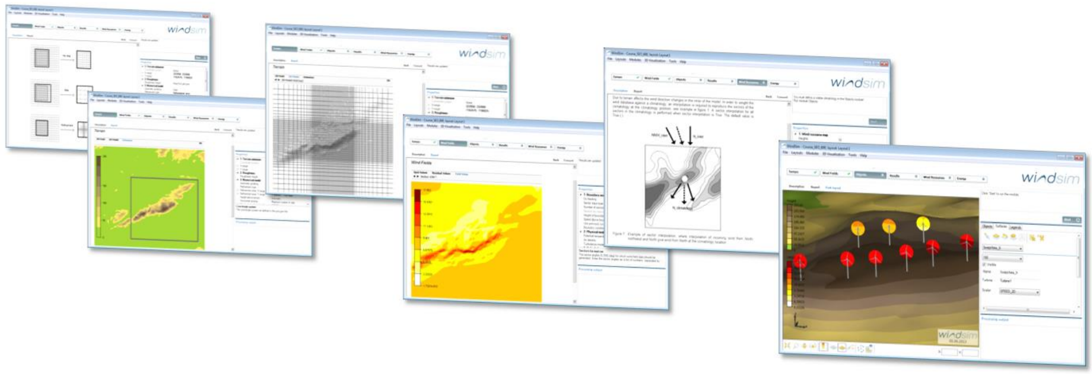


Figure 1 - WindSim is a modular based Wind Farm Design Tool (WFDT)

A full micrositing will require execution of all six modules. The modules must be executed in the right order as there are dependencies between the modules. However, depending on the purpose of the project, it is not always necessary to run all the modules.

WindSim Modules

- ***Terrain***
  Establish the numerical model based on height and roughness data.
- ***Wind Fields***
  Calculation of the numerical wind fields.
- ***Objects***
  Place and process wind turbines and climatology data.
- ***Results***
  Analyse the numerical wind fields.
- ***Wind Resources***
  Couple the numerical wind fields with climatology data by statistical means to provide the wind resource map.
- ***Energy***
  Couple the numerical wind fields with climatology data by statistical means to provide the Annual Energy Production (AEP); including wake losses. Determine the wind characteristics used for turbine loading.

In addition to the modules there are stand-alone *Tools* for data preparation and data post-processing. In particular, *Tools* are used for the import and preparation of terrain and climatology data. WindSim also works with Add-on Modules including the Remote Sensing Correction Tool (RSCT) and Park Optimizer.

About the Technology

Computational Fluid Dynamics (CFD) is used to perform the wind field simulations in WindSim. CFD is a numerical method for solving the fundamental equations of fluid flow. CFD has become a very useful method within many industries. Accurate flow simulations are required within the automotive industry, oil and gas, and of course within the aerospace industry. In these industries CFD has become the standard method for flow calculations.

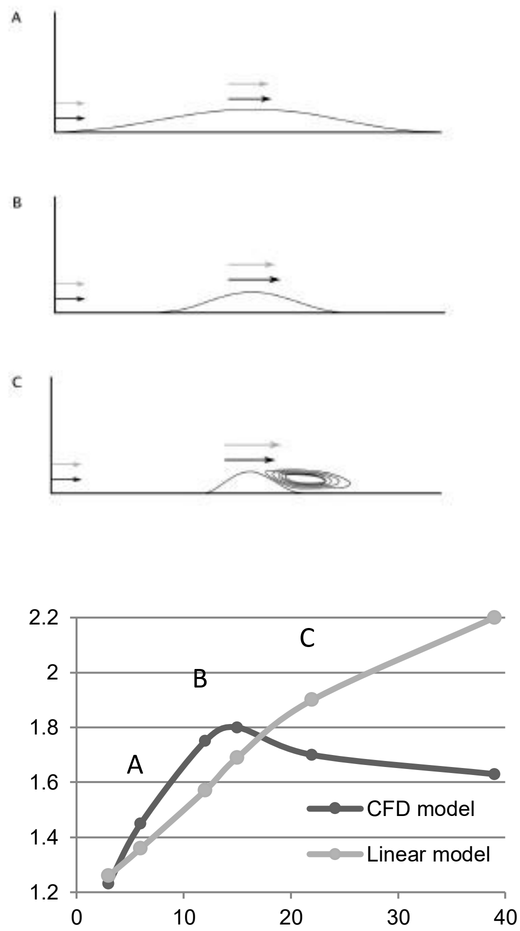


Figure 2 - Upstream speed-up, (Speed hill top/speed inlet). Average inclination angle (degrees) A) 5.7 B) 11.3 C) 21.8 - Linear model (light grey) CFD model (dark grey).

The fundamental behaviour of fluid flow is described by the Navier-Stokes equations. The Navier-Stokes equations are non-linear partial differential equations known to be unstable and difficult to solve. Therefore simplified methods, where the troublesome non-linear terms have been linearized have become popular within the wind energy sector. However, the severe penalty is the reduced accuracy in the results. The differences between the traditional so called linear method and the CFD method could be illustrated by looking at speed-up over a ridge. The speed-up increases with increasing inclination angles until the flow separates, as seen in the upper part of Figure 2.

This behaviour is captured by a CFD method. Even for smaller inclination angles, when the flow does not separate there is a significant difference in predicted speed-up between the linear and the CFD methods.

For an inclination angle above 20 degrees (Case C) the flow separates. The recirculation acts as an extension of the terrain, the ridge becomes more like a plateau and the speed-up is reduced.

The Navier-Stokes equation is time averaged introducing terms for the speed fluctuations. A turbulence model is required to close the equation set of the so-called Reynolds Averaged Navier-Stokes equation (RANS). The RANS equations are discretized in a computational domain and integrated with a finite-volume procedure.

There are various solvers and solver strategies implemented in the software. Turbulence is taken into account using various types of two equation models like the k-ε model.

We invite you to learn more about the Bolund Experiment. The Bolund experiment is a field campaign that provides datasets for validating numerical models of flow in complex terrain and was the basis for a unique blind test comparison of flow models. The CFD methods—including WindSim—showed the lowest errors among the various methods used. Find out more about The Bolund Experiment (windsim.com).

WindSim Evaluation versus WindSim Commercial

WindSim Evaluation (EV) contains all the features of the commercial version but is intended for evaluation use. WindSim EV can be downloaded for free. The only restriction of WindSim EV is the maximum number of cells used to discretize the computational domain. The total maximum numbers of cells are 50 000, and there are 10 layers of cells vertically. The limit on the number of cells; as will be shown in the following paragraphs, have two effects:

- It smoothes the given height and roughness data
- It introduces significant discretization errors

The results obtained with WindSim EV are therefore not intended for commercial work. However, the user will be able to explore all the features of the software with the evaluation version, and also appreciate the power of the commercial version.

Terrain

The first step in the set-up of flow field simulations is the generation of a 3D model of the area of interest. This is done in the Terrain module. But first the basis for the 3D model must be available, which is a 2D dataset with elevation and roughness data in *.gws* format.

Digital terrain conversion

When starting a new project you will be asked for a *grid.gws* file containing the elevation and roughness data of the terrain in a regular grid. The *grid.gws* file can be viewed by using the *Tools → View terrain model* menu item. If no *.gws* file is specified a demo file is automatically copied into the dtm (digital terrain model) projects folder.

Conversion from other formats is performed by clicking the *Tools → Convert terrain model* menu item. If you need extensions in order to perform the conversion of your specific data, see examples of supported **third party formats**. Alternatively, you can convert your terrain model directly to the *.gws* format by using your own tools, see: **terrain field data**.

Properties

1. Terrain extension

Coordinate system

The coordinates in the *grid.gws* file can refer to any global orthogonal system. This coordinate system is called system 3 in the *grid.gws* file. After generation of the 3D model, a local coordinate system (referred to as system 1) is introduced in the lower left-hand corner. Later in the Objects module the placements of objects can be referenced to by either the global or local coordinate system.

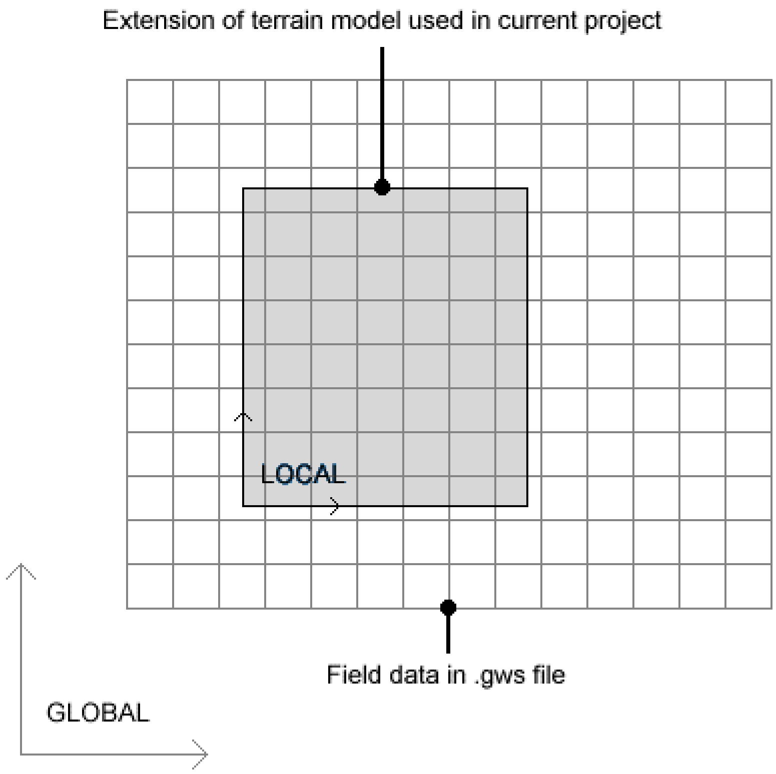


Figure 3. Sketch of the defined coordinate systems.

X-range and Y-range

Extensions in the east-west direction and in the south-north direction. The specified coordinates may be increased to fit the nearest node in the *grid.gws* file, as interpolation of the underlying data is avoided. Default values are the minimum and maximum extensions in the *grid.gws* file [m].

Projection

The parameters that are shown under Projection identify the global coordinate system as given in the *grid.gws* file distinctively. Given is the Projection system, the Datum and the Zone. From that information WindSim calculates the EPSG code and stores that information in the *grid.gws* file for later use in WindPRO and Google export.

2. Roughness

Roughness height

By default, variable roughness heights are read from the grid.gws file. Alternatively, a constant roughness height can be imposed in the model by specifying a non-zero value for the roughness height.

The roughness height is defined in the log-law

$$\frac{U}{U\tau} = \frac{1}{\kappa}\ln(z/z0),$$

where

U - wind velocity,

U_τ - friction velocity $\sqrt{\frac{\tau_0}{\rho}}$,

τ_0 - shear stress,

ρ - air density,

κ - von Karmans constant ($\kappa = 0.435$),

z - coordinate in vertical direction,

z_0 - Roughness height.

3. Numerical model

With the Automatic gridding option set to False the grid on the ground level specified by the Refinement type as well as the grid in vertical direction can be adjusted by the user.

Vertical Grid

The grid in vertical direction is determined by the following parameters being relevant for all choices of Refinement type except the specification of the whole grid by loading a refinement file.

Vertical Expansion

Enable the choice to have the type of vertical expansion, Arithmetical or Geometrical expansion.

Height above terrain

The height above the terrain is defined as the vertical distance between the highest elevation point in the 3D model and the upper boundary. In order to set a proper value for this height, two seemingly contradictory requirements must be balanced. On the one hand, the distribution of nodes in the vertical direction should be as dense as possible for obtaining accurate numerical solutions, near the ground (see also *Height distribution factor* and *Number of cells in Z direction*). This requirement implies that the upper boundary should be placed as near the ground as possible. Yet, on the other hand, if the upper boundary is too close to the ground this would impose a blocking effect when the flow field passes over mountains.

As a rule of thumb the fraction between the minimum open area z_{min} and the maximum open area z_{max} between the ground and the upper boundary, calculated as the model is traversed in west-east and south-north direction, should be larger than 95%. By setting *Height above terrain* to Automatic a height will be calculated satisfying the relation $\frac{z_{min}}{z_{max}} > 0.95$. In some cases the automatic procedure might fail. The procedure will not capture the case with a ridge along the diagonal of the model, as the traverse in west-east and south-north direction will not detect significant changes in open area. But blocking might occur when the incoming flow is perpendicular to the ridge. Likewise for an isolated island or mountain the height of the upper boundary will be reduced (as soon as the 2D modelled area) as the total modelled area is increased, which could lead to blocking in extreme cases. Alternatively, a specific height could be given.

Height distribution factor

The cell distribution in the vertical direction follows an arithmetic or geometrical sequence. If arithmetical the height distribution factor gives the fraction between the cell at the ground and the cell at the upper boundary. If geometrical the height distribution factor defines the proportion of width increase of each cell height compared to the lower one. The vertical cell distribution could be inspected under the 3D model section in the terrain report. Default value is 0.1 [-] if arithmetical expansion, 0.3 [-] if geometrical.

Number of cells in Z direction

Number of cells in the vertical direction. Default value is 20 [-].

Low-level uniform cells

Defining the low-level uniform cells enable to set the first vertical cells to have uniform grid spacing, the above cells follow the expansion defined by previous options. Uniform vertical cells number *n* and height *h* define the level properties and resolution. The vertical grid area of height h is discretized by *n* cells. The levels above the uniform area are expanding, depending on the height of the models and the expansions settings. All cells' heights above the uniform area are set to have the same cell height as in the low-level area or higher. Compression and expansion in the vertical grid are not allowed as it could produce unrealistic solver results.

Refinement type

By default no refinement of the grid in the ground plane is performed. By setting the *Automatic gridding* option to False additionally a refinement area can be specified, then a denser distribution of nodes will be allocated within the refined area.

Refinement area, X-range and Y-range

Extensions in the east-west direction and in the south-north direction of the refinement area. By default the area is given in the centre with an extension equal to 1/3 of the total area extension. The cell distribution is uniform within the refined area with an increasing cell size towards the borders. The data used for setting up the refinement area is stored in the current project under the folder dtm in the file simple_refinement.bws.

Horizontal gridding

- Maximum number of cells:
  The 3D model will consist of $n_x \cdot n_y \cdot n_z$ cells. The number of cells in *z* direction is set in *Number of cells in z direction*, default value is 20 [-]. The number of cells in x and y direction is set according $n_x \cdot n_y \cdot n_z$ *maximum number of cells*, by skipping nodes in the field data set. Whenever a regular model is created then the extension of the model might be slightly modified as no interpolation of the underlying field data is performed, see figure 4. If Refinement is used, then an interpolation is performed anyway and the number of cells in the model will be approximately adjusted to the specified *Maximum number of cells*.

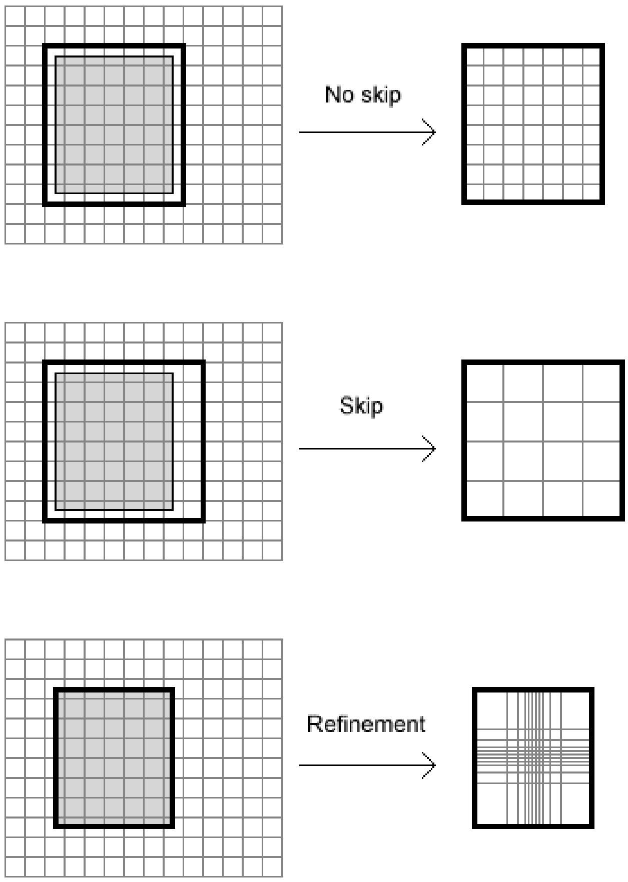


Figure 4. Field data extraction fitting the set maximum number of cells. In the upper case the maximum number of cells has been given a large value and the finest resolution possible for a regular grid model was created. In the middle case the maximum number of cells is smaller and the extraction procedure skipped every second node in the field data set. In the lower case refinement is set and the extraction procedure interpolates the field data set giving a model with approximately the maximum number of cells specified by the user.

The obtained number of cells and corresponding grid resolution is found under the 3D model section in the terrain report. The computing time required is exponentially proportional to the number of cells. Consider splitting a large model into several smaller models using the nesting technique as described under the Wind Fields module or use the refinement technique in order to handle large terrain areas.

- Horizontal resolution:
  In the case of refinement described with a refinement area the computational grid can also be constructed by assigning the *Horizontal resolution* in the refined area, combined with a law for the expansion of the grid in the zones surrounding the refined area.
- Ratio additive length to resolution

The expansion of the grid is defined by the ratio of the additive length δ to the horizontal resolution set in the inner refined area.

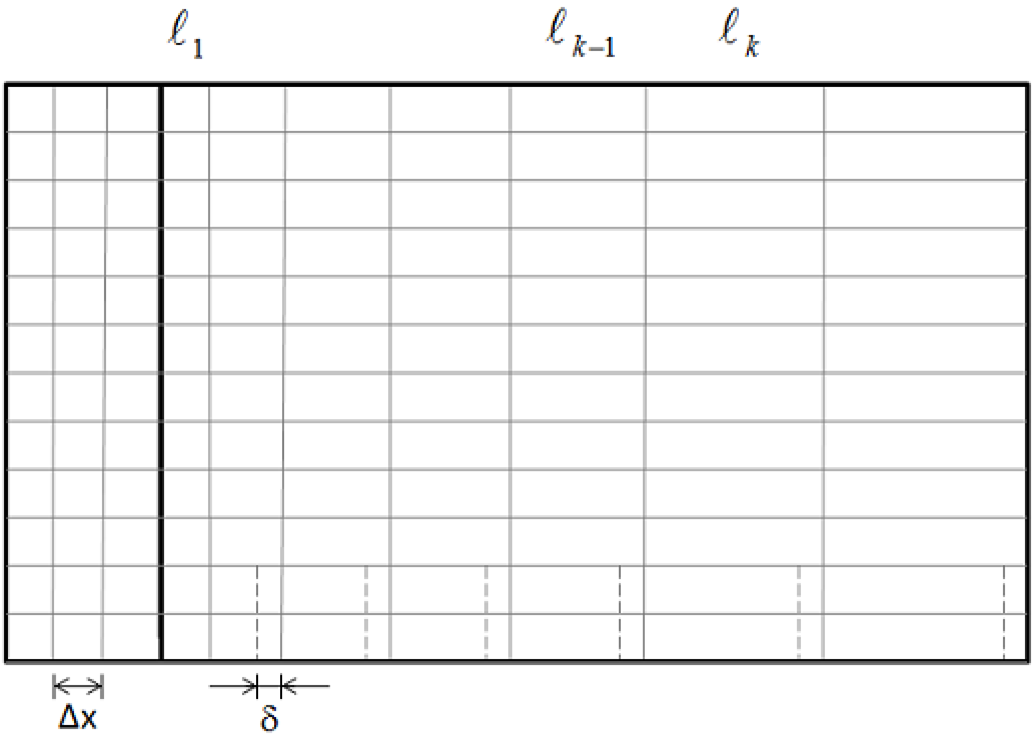


Figure 5. Sketch of the expansion law applied to the horizontal grid.

Expansion law, arithmetic sequence, for a zone on the right side of uniform grid. A fixed length δ is added to the length of a general cell $k-1$ to obtain the length of the successive cell k. The additive length δ is calculated from the Ratio additive length to resolution $\delta / \Delta x$, where

$$L = \sum_{k=1}^{7} l_k$$

with $l_k = l_{k-1} + \delta$ and $l_1 = \Delta x$

Refinement/blocking file

Files with the extension .bws are blocking or refinement files and can be loaded separately, syntax and examples.

Actuator disc

The actuator disc is the concept to model a turbine as a disc that is providing forces to the air flow which are equivalent to the actual forces exchanged by a real wind turbine and the flow. With WindSim it is possible to generate a grid modelling the wind turbines as a set of actuator discs. In order to design a computational grid with a set of actuator discs an object file (.ows) has to be defined which contains the layout of the wind farm. Additionally, the number of spacings can be specified, referring to the number of spacings used to discretise the rotor diameter.

- Object file:
  An actuator disc will be designed over each wind turbine. The layout of the wind farm is therefore loaded from an object file (.ows). The same layout has to be loaded in the Objects module of WindSim where it can also be produced as an output file. Thus the standard procedure in order to include the actuator disc model can be as follows: first run the Terrain module and neglect the actuator disc option therein, then jump to the Objects module, place all turbines as explained in the corresponding "description pages" and export your windfarm layout by using the Export → Export objects (.ows) dropdown option, finally return to the Terrain module and import the Object file here with the actuator disc enabled. The same layout has to be loaded in the Objects module of WindSim, to reach the goal is useful to load the same object file in the Objects module. In order to run WindSim with the actuator disc option enabled, the module to be run after the Terrain module has to be the Objects (having the same layout included in the .ows of the Actuator disc), and, after the Objects, the Wind Fields.
- Number of spacings:
  The computational grid is designed having as reference length a subdivision of the rotor diameter which will be the minimum grid resolution, both horizontal and vertical, in the area of the wind farm. The grid will expand gradually from the wind farm region to the borders and top of the domain. The default value is 16 that has proven to bring the discretization errors to an acceptable low value, therefore it is suggested to keep the number of spacings to 16 or higher.

4. Smoothing

Smoothing type

Various smoothing types can be used to smoothen the terrain. In some cases a wind field simulation fails to give a physical solution, the solution procedure diverges. Smoothing of the terrain is a technique for battling divergence.
Divergence could be caused by abrupt changes in the inclination between adjacent cells. Areas with abrupt changes in the inclination would be found in narrow valleys or at sharp mountain peaks. Typically, these areas would not be of interest for placing wind turbines, and as such a slight modification of the terrain could be accepted.
Terrain smoothing should be used with care, as excessive use will significantly change the heights of the terrain. Both the second order derivatives, which is a measure of the smoothness, and the changes between the original heights and the heights after smoothing are given in the report. Inspect these results, and make sure that areas of interest have not

been significantly changed. To smooth areas that are not of interest will speed-up the wind field simulations.

No smoothing

By default no smoothing is applied.

Bi-linear smoothing

Smoothens the terrain by marching along x and y directions sequentially. Bi-linear smoothing is a line smoothing routine.

Gaussian Smoothing

Smoothens the terrain by providing a Gaussian weighting to the neighboring nodes. *Gaussian smoothing* is a surface smoothing routine. In most cases, *Gaussian smoothing* will smoothen the terrain more than the *Bi-linear smoothing*. An example will be a saddle point, where the second order derivative is positive along one of the axis and negative along the other, which the *Bi-linear smoothing* might fail to smooth to the given *Terrain smoothing limit*.

Terrain Smoothing Limit

In mathematical terms changes in the inclination are represented by the second order derivatives of the elevation. On specifying a value for the *Terrain smoothing limit*, all places where the second order derivatives are higher than the smoothing limit are smoothed in an iterative procedure, until the second order derivatives are less than the specified limit.

Smoothing radius

The Smoothing radius is used to define the area where the smoothing is applied, in accordance with the below setting of *Gradual smoothing type*. *The Smoothing radius* is defined as the radius of a circle centered in the mid-point of the simulation domain.
The *Smoothing radius* has a value between 0 and 1, reaching the value 1 at the corners of the simulation domain.

Gradual Smoothing Type

The *Gradual smoothing type* is used to weight the smoothing in the inner or outer part of the simulation domain, preserving a continuous smoothing in the whole domain. The settings of the property variables *Terrain smoothing limit*, *Smoothing radius* and *Gradual smoothing* type allows for a wide range of terrain smoothing combinations.

Inner

Applies gradual smoothing from the center to the *Smoothing radius* and thereafter constant smoothing towards the border of the model, see definition sketch.

Outer

Applies no smoothing from the center to the *Smoothing radius* and thereafter a gradual smoothing towards the border of the model, see definition sketch.

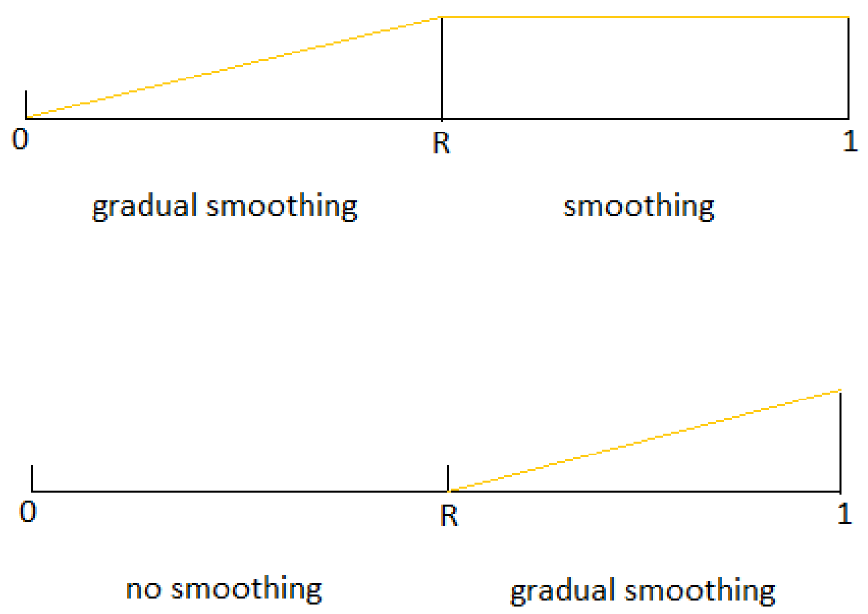


Figure 6. Definition sketch for the Gradual smoothing type equal to Inner (top) and Outer (bottom), where R is the non-dimensional Smoothing radius.

5. Forest

Forest

By default the existence of a forest is disregarded. By changing the dropdown menu to *Base on roughness height* a forest can be considered by specifying a collection of parameters in the *Forest setup*. The forest is then associated to a certain roughness height present in the file *grid.gws* and is modelled by configuring the associated grid-cells. Here, source/sink terms S_1, S_k and S_ϵ are introduced in the governing equations as explained in the description of the Wind Fields module where they are specified.

Forest setup

Roughness height

The roughness height to that the forest is assigned. A forest is established for all locations with this given roughness height in the *grid.gws* file. By default the *Roughness height* is set to 0.5 [m].

Forest height

The height of the forest. Default value is 30 [m].

Forest resistive force constant C_2:

The resistive force constant C_2 is used to represent the forest as a resistive force proportional to the velocity squared

$$S_{2,i} = -\rho C_2 \mid u \mid u_i,$$

where *i* indicates the component in space and $\mid u \mid$ is the magnitude of the wind velocity. By default the C_2 is set to 0.1 [1/m]. Note that the parameters *Forest porosity* and *Forest resistive force constant* C_1 no longer are used for the forest modelling in *WindSim* as they are incompatible with the current solvers.

Forest turbulence sources

Sources of turbulence are introduced in the κ and ϵ transport equations. Formally the model has been implemented as in Sanz (2003) [1] and Katul (2004) [2] with the model constants revised to be compatible with the default set of constants of the standard $\kappa - \epsilon$ model,

$$S_k = C_2(\beta_P \mid u \mid^3 - \beta_D \mid u \mid k)$$

$$S_\epsilon = C_2 \frac{\epsilon}{k}(C_{\epsilon 4}\beta_P \mid u \mid^3 - C_{\epsilon 5}\beta_D + \mid u \mid k)$$

With $\beta_P = 1.0$, $\beta_D = 6.51$, $C_{\epsilon 4} = 1.24$ and $C_{\epsilon 5} = 1.24$.

Forest cell count in Z direction

The number of cells in the vertical direction used for the forest can be specified by this variable. The cell distribution is uniform. The remaining number of cells used above the forest, is *Number of cells in Z direction - Forest cell count in z direction*. The vertical cell distribution could be inspected under the 3D model section in the Terrain report. By default the *Forest cell count in Z direction* is set to 3 [-].

References

[1] Sanz, C. "A NOTE ON k - ε MODELLING OF VEGETATION CANOPY AIR-FLOWS." Boundary-Layer Meteorology, Kluwer Academic Publishers, No.108, pp191-197, 2003.

[2] Katul, G.G., Mahrt, L., Poggi, D. & Sanz, C. "ONE- AND TWO-EQUATION MODELS FOR CANOPY TURBULENCE." Boundary-Layer Meteorology, Kluwer Academic Publishers, No.113, pp81-109, 2004.

Wind Fields

Generation of the wind fields

Based on the generation of the 3D model in the preceding **Terrain** module, the simulation of the wind fields can now start. The wind fields are determined by solving the Reynolds Averaged Navier-Stokes equations (RANS). The standard $\kappa - \varepsilon$ model is applied as turbulence closure. Since the equations are non-linear the solution procedure is iterative. Starting with the initial conditions, which are guessed estimations, the solution is progressively developed by iteration until a converged solution is achieved.
The flow variables that are solved are:
Pressure p
Velocity components (u, v, w)
Turbulent kinetic energy k
Turbulent dissipation rate ε

In more complex situations additional flow variables such as **temperature** T can be included. The governing equations of the WindSim CFD model correspond to the RANS equations, assuming steady-state (derivatives in time are set to zero) and incompressibility (constant density). The conservation of momentum in the horizontal direction is expressed as

$$u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\nu \left[\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right] - \overline{u_i u_j}\right) + \frac{1}{\rho}\left(S_{1,i} + S_{2,i}\right) \qquad i = \{1,2\},$$

while mass conservation is ensured by $(\partial u_i / \partial x_i = 0)$. Here ρ refers to the air density and ν is the air viscosity. The terms indicated by $S_1$and S_2 correspond to forest resistive forces as introduced in the **Terrain** module description. The conservation of momentum in the vertical direction has an additional forcing term when thermal effects (atmospheric stability) are taken into

$$u_j \frac{\partial u_3}{\partial x_j} = \frac{\theta_0 - \theta}{\theta_0} g - \frac{1}{\rho}\frac{\partial p}{\partial x_3} + \frac{\partial}{\partial x_j}\left(\nu \left[\frac{\partial u_3}{\partial x_j} + \frac{\partial u_j}{\partial x_3}\right] - \overline{u_3 u_j}\right) + \frac{1}{\rho}\left(S_{1,i=3}\right),$$

where g is the gravitational acceleration, θ is the potential temperature and θ_0 is its reference value. For neutral simulations $\theta = \theta_0$ and therefore the extra forcing term vanishes. The potential temperature is influenced by advection, thermal diffusion and turbulent heat transfer, expressed as

$$u_i \frac{\partial \theta}{\partial x_i} = \frac{\partial}{\partial x_i}\left(\alpha \frac{\partial \theta}{\partial x_i} - \overline{u_i \theta'}\right),$$

where α is the kinematic molecular diffusivity for heat in air. The turbulent terms in the Equations above are parametrized as follows

$$\overline{u_i u_j} = -\nu_T \left[\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right] + \frac{2}{3}\delta_{ij},$$

$$\overline{u_i \theta'} = -\frac{\nu_T}{\sigma_\theta}\frac{\partial \theta}{\partial x_i},$$

where ν_T is the turbulence viscosity and $\sigma_\theta = 1$ corresponds to the turbulent Prandtl number for heat transfer. Additional information concerning the setup for the applied $\kappa - \varepsilon$ closure can be found under *Turbulence model*. More details on the physical background of the model can be found in reports and publications on the WindSim web site.

Boundary and initial conditions

Along the borders of the 3D model, information about the flow field must be supplied. The wind flow solution obtained by a steady-state RANS model depends on these selected boundary conditions. These boundary conditions make explicit assumptions about the wind conditions, such as wind speed, wind direction, temperature and turbulence. The boundary conditions can be specified in two ways, either as analytical profiles or by nesting.

The analytical wind profiles are empirical profiles over flat terrain. The vertical profiles are dependent on the roughness height and if the temperature is activated also on the stability of the atmosphere. There are associated profiles for the turbulence. In case of a neutral atmosphere the wind profiles are log profiles. Applying the log profile along the border is equivalent to the placement of an infinite flat terrain upstream of the 3D model. This might be a bad assumption, therefore the results along the borders should be treated with care.

The nesting technique involves results from a former 3D model that fully covers the current 3D model. The profiles at the boundaries are interpolated from the former 3D model. This former model can be a WindSim model or a mesoscale meteorological model. Likewise the initial conditions are interpolated from the former 3D model. Applying the nesting technique from a WindSim project reduces the inaccuracies introduced by applying the log profiles. It is highly recommended to start simulations in a large area and make refined models around the areas of interest using the nesting technique.

Using the **nesting technique from a meteorological mesoscale model** gives the possibility of simulating special weather patterns. The information from the mesoscale model is interpolated to the WindSim grid as boundary and intial conditions. Meteorological data has to be given in an ***.xyz format** and needs to be converted into the WindSim format ***.dws**. This can be done using the meso-microscale coupling console under Tools → Open meso-microscale coupling console.

To calculate the annual energy production from simulations nested with a meteorological mesoscale model, WindSim provides a methodology that utilizes the average mesoscale conditions per directional sector, described in Duran et al. [1]. This can be done using the meso-microscale coupling console under Tools → Open meso-microscale coupling console. The console will use the provided *.xyz timeseries and produce one *.xyz per directional sector. These files can be then converted into *.dws files, using the same console.

Computer resources

Be aware that 3D simulations can be quite demanding regarding computer resources and time. Typically, the simulation time is exponentially proportional to the number of cells in the model. It could be advantageous to split large models into smaller ones with the nesting technique. Running several simulations simultaneously is not recommended.

Accuracy and convergence

For descriptions on the accuracy and convergence of the wind field simulations please refer to **accuracy** and **convergence**. For general advice on how to achieve accurate and faster converging results with WindSim software, see **Best Practices and Recommended Settings**.

Properties

Boundary and initial conditions

Do nesting

Specify whether nesting from a WindSim project or from a mesoscale meteorological model should be activated or not. By default nesting is disregarded.

Project file name

In case nesting should be done *From a WindSim project* the name of the WindSim project as a .ws file has to be specified. The nesting project must surround the current project, except at the upper boundary. If the vertical extension of the current project goes above the nesting project a constant extrapolation will be applied.

Folder for mesoscale data

In order to do nesting *From mesoscale* the folder name where the ***.dws** files for this project are stored has to be specified. The nesting project must surround the current project completely.

Sectors for next run

The wind fields will be simulated for the sectors given in this list. Any wind direction can be given, but if climatology data is then introduced, it is recommended to simulate the same sectors as those found in the climatology data. For further processing of the modules **Wind Resources** and **Energy** there must be a correspondence between the sectors in the climatology and the sectors for which flow simulations are performed. Sector 0 gives wind from North, sector 90◦ gives wind from East etc. Default values are 12 sectors from 0 to 330◦.

Sector Input Type

The list containing the *Sectors for the next run* can either be inserted manually by choosing the option *Manually set sector angles* (the angles of interest have to be separated by a semicolon) or the list can be generated automatically by specifying the *Number of sectors* with the *Uniform distrubution of sector angles* option, here the full range of 360◦ is equally distributed into the indicated *Number of sectors*.

Cases for next run

When nesting is done from a mesoscale meteorological model then the wind field will be simulated for the cases given in this list. Any case number until 1000 can be given.

Height of boundary layer

The loglog profile as explained in ***Boundary and initial conditions*** is defined from ground up to the boundary layer height, above this height the profile is constant. Default value is 500m.

Speed above boundary layer

The constant speed above the boundary layer height. Default value is 10m/s.

Use previous run as input

If a wind field simulation has been undertaken for a sector, the simulation can be continued by using the restart facility. Restart means that the solution from the former run is used for the initial conditions. In situations when the number of iterations specified in the preceding simulations have not led to a converged solution, a restart could progress the solution further towards convergence. The restart is not yet available for simulations with temperature included and for simulations driven by meteorological data. Default value is False [-].

Boundary condition at top

Three types of boundary conditions could be set at the top: fixed pressure condition, no-friction wall or diffusive link (moving fixed pressure). The fixed pressure condition could be used on complex terrain, while the no-friction wall is for use over flat terrain, diffusive link is like fixed pressure case but imposing a constant speed at the top. The imposition of a constant speed enables to lower the height of the model and use less cells vertically. Consider that no friction wall case is restricting the flow, no mass can flow out or in at all by the top boundary layer. Details on the behavior of the top boundary condition can be found under Documentation on the WindSim web site. Default value is *Fixed pressure*.

Physical models

Potential temperature

Activation of the **temperature** equation. Depending on the type of initialization different boundary and initial conditions are set:

Initialize from Monin-Obukhov length

Depending on the given value for the Monin-Obukhov length the vertical profiles at the inlet are calculated stability dependend and a linear temperature gradient over the whole atmosphere is assumed.

- Reference temperature:
  Potential temperature at sea level. Default value is 288 K. This value is only important for visualization purpose and does not influence the results. Select a value which is reasonable for your area.
- Monin-Obukhov length:
  Stability Parameter with a range of -10000 to 10000 except 0. Default value is 100 m (very stable atmosphere). WindSim leaves the user the freedom to choose this parameter. Reasonable default values are |L| > 1000 for neutral atmosphere, 200 < L < 1000 for stable atmosphere and 0 < L < 200 for a very stable atmosphere. 0 > L > -200 for a very unstable atmosphere and -200 > L > -1000 for an unstable atmosphere. As WindSim by default is setup to run steady state we do not recommend to run simulations with very unstable conditions as this can lead to divergence problems.
- Reference height and Wind Speed in reference height:
  Depending on the stability of the atmosphere the wind profile in the higher elevations differs considerably (see Figure 7). It is therefore more convenient to prescribe a “reference speed” in a “reference height” near the surface (e.g. mean annual wind speed of your climatology). This can be done by these two parameters which are 0 by default. If both values are set to zero the “speed at boundary layer height” is used to calculate the boundary profiles. The boundary layer height has to have a reasonable value as it defines the height where the TKE gets zero and the wind profiles get a constant value. For stable simulations we recommend to use a boundary layer height of around 300 m and for unstable simulations of around 800 m.

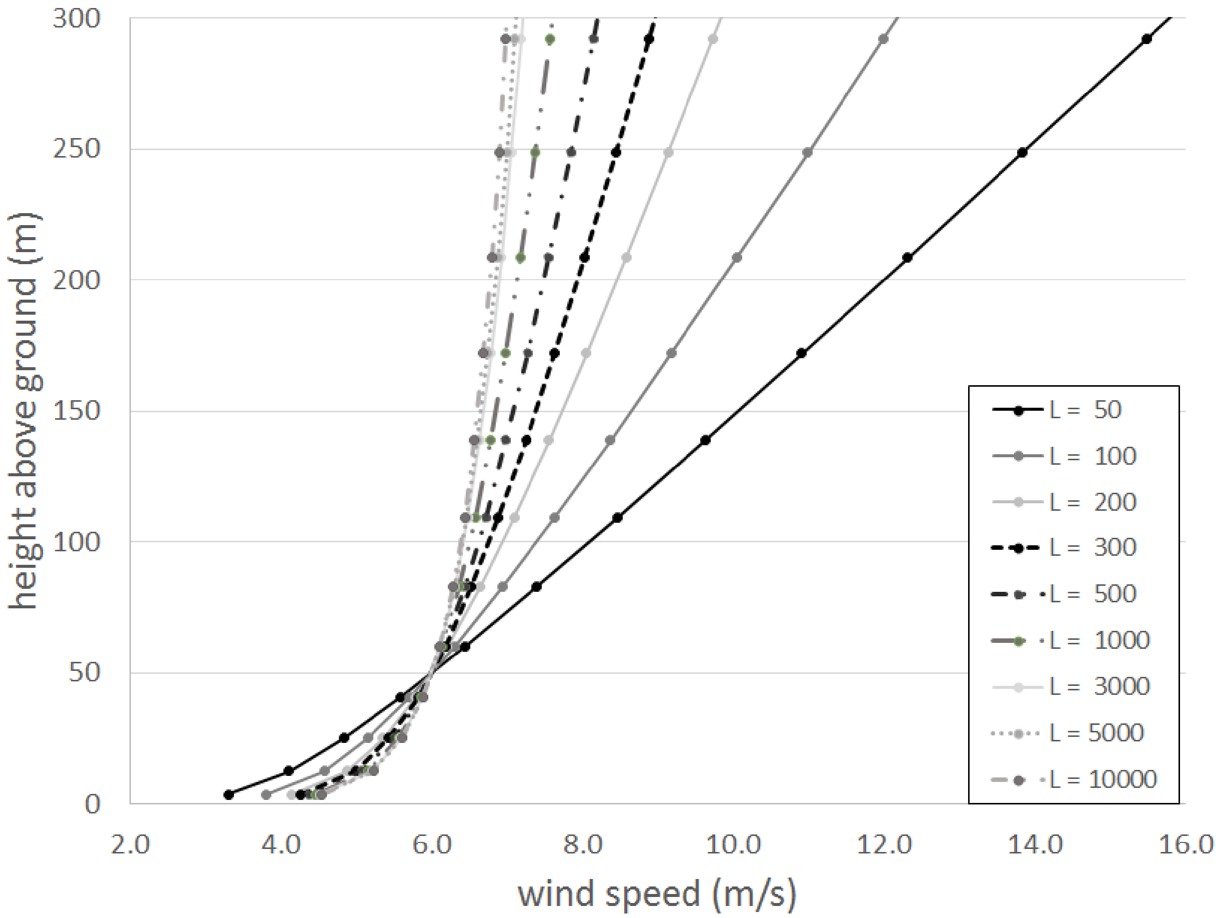


Figure 7. Wind speed profile for different Monin-Obukhov lengths (roughness length of 0.001 m).

- Initialize from mesoscale:

  WindSim provides the possibility to derive the boundary conditions for temperature from a mesoscale meteorological model. Mesoscale temperature values are interpolated for the initialization of the CFD simulation. In Fig. 3, the temperature boundary conditions derived from analytical profiles and from a mesoscale model are compared.

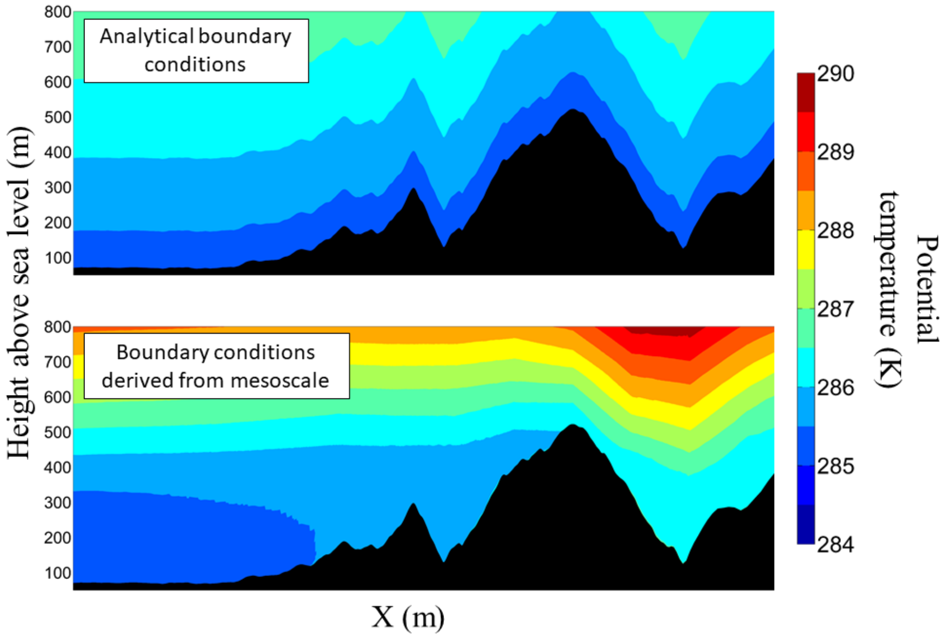


Figure 8. Example of potential temperature fields obtained from an analytical model (top panel) and from a mesoscale model (bottom panel).

Air density

This is the air density used in the CFD simulations [kg/m3].

Turbulence model

The first turbulence model is the standard $\kappa - \varepsilon$ model. The $\kappa - \varepsilon$ model belongs to the family of eddy viscosity models, the eddy viscosity ν_T is calculated by the following analytical equation.

$$\nu_T = C_\mu \, k^2/\varepsilon$$

A version with modified model constants is also available [2], please refer to the section "Library/Papers & Presentations/" at **WindSim** for details. For high turbulent Reynolds numbers, the standard form of the $\kappa - \varepsilon$ model may be summarised as follows

$$\frac{\partial(u_i\, k)}{\partial x_i} = \frac{\partial}{\partial x_i}\left[\frac{\nu_T}{\sigma_k}\frac{\partial k}{\partial x_i}\right] - \varepsilon + P_k + P_b + \frac{1}{\rho}\, S_k$$

$$\frac{\partial(u_i\, \varepsilon)}{\partial x_i} = \frac{\partial}{\partial x_i}\left[\frac{\nu_T}{\sigma_\varepsilon}\frac{\partial\, \varepsilon}{\partial x_i}\right] - C_{\varepsilon 2}\,\frac{\varepsilon^2}{k} + C_{\varepsilon 1}\,\frac{\varepsilon}{k}\,(P_k + C_{\varepsilon 3}\, P_b) + \frac{1}{\rho} S_\varepsilon$$

Here κ is the turbulent kinetic energy and ϵ is the dissipation rate, ρ is the fluid density and ν_T is the turbulent kinematic viscosity. S_ϵ and S_κ are due to the forest resistive forces as introduced in the **Terrain** module description. The additional kinetic and thermal contributions P_k and P_B are given

$$P_k = \nu_T \left[\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right]\frac{\partial u_i}{\partial x_j},$$

$$\text{and} \quad P_b = -\frac{\nu_T}{\sigma_\theta}\frac{g}{\theta}\frac{\partial \theta}{\partial x_3}$$

The term covered by P_B introduces thermal effects to the model and is therefore only present if the *Potential temperature* is included, otherwise $P_B = 0$. The values of the model constants in Standard $\kappa - \varepsilon$ model are given as follows.

C_μ	σ_k	σ_ε	$C_{\varepsilon1}$	$C_{\varepsilon2}$	$C_{\varepsilon3}$
0.09	1.0	1.314	1.44	1.92	1.0

The values of the model constants in Modified $\kappa - \varepsilon$ model are given as follows.

C_μ	σ_k	σ_ε	$C_{\varepsilon1}$	$C_{\varepsilon2}$	$C_{\varepsilon3}$
0.0324	1.0	1.85	1.44	1.92	1.0

The constants are modified to better fit experimental data of the Atmospheric Boundary Layer, as described in [2].

In **WindSim** two further modifications of the $\kappa - \varepsilon$ model are available: the YAP correction [3] and the RNG (ReNormalization Group) version [4]. The Yap correction adds a term to the ε equation, this correction is reported to provide much-improved predictions in separated and reattaching flowsway. It gives better results in case of flow separation, the models constants are the same as Standard $\kappa - \varepsilon$ model. RNG techniques have been used to develop a theory for the large scales in which the effects of the small scales are represented by modified transport coefficients. The obtained model differs from the Standard $\kappa - \varepsilon$ model in the set of constants used and has an additional term in ε equations. RNG model has proved better results in case of flow separation. The values of the model constants in RNG $\kappa - \varepsilon$ model are given as follows.

C_μ	σ_k	σ_ε	$C_{\varepsilon1}$	$C_{\varepsilon2}$	$C_{\varepsilon3}$
0.0845	0.7194	0.7194	1.42	1.68	1.0

The boundary conditions of model at the inlet are set for κ and ε with vertical profiles as described in [2], C_μ constant affects the κ profiles and is modified accordingly to the turbulence model selected.

Calculation parameters

Solver

During the infancy of Computational Fluid Dynamics, only algorithms with small demands on computing power could survive. Computer memory was then extremely rare and expensive. In this context, use of the Patankar-Spalding SIMPLE algorithm and its descendants (SIMPLER, SIMPLEC, SIMPLEST, PISO...), based on the segregation of momentum and continuity equations was the best strategy to adopt. This strategy used to be activated with the segregated solver which today is obsolete.

But as the number of cells increases, the elliptic nature of the pressure field becomes a penalty and the global convergence of the method strongly slows down. The present state of the technology opens doors to other ways of thinking. The mean amount of RAM on PCs is continually increasing and its price decreasing so that, in many case, refining a grid up to the computer storage capability leads to some huge CPU times to convergence. Therefore, as soon as the bottleneck becomes the required time for the solution, we had some other options.

Convergence problems can be overcome by using the General Collocated Velocity method (GCV). The method uses a segregated pressure-based solver strategy with an additional correction of cell-centre momentum velocity components, which provides faster convergence in comparison with the standard one-step face velocity correction. The pressure-velocity coupling algorithm is based on a linearisation which is similar to the well known SIMPLEC procedure, and is generalised for arbitrary BFC geometries. The linear equation solver is based on the conjugate-residual algorithm with LU preconditioning.

The convergence with an increasing number of cells can also be handled by using the parallel GCV solver. The solver makes it possible to split the simulation area into several domains which are calculated seperately. This speeds up the calculation process if several CPUs are available.

Hypre [6] is a library of high-performance preconditioners and solvers, with BoomerAMG enabled in WindSim. This uses algebraic multigrid (AMG) algorithm for the linear equation solver for the GCV algoritm. The new solver setup shows a better convergence behavior for big models run with parallel option. This solver enables future development to solve on semi-structured grid.

Number of simultaneous sectors

Number of sectors which will be run simultaneously. The maximum number is limited to the number of CPU's of the machine.

Number of CPU's per sector

For the parallel solver version the domain of each sector can be additionally split into several subdomains which are calculated on different CPU's (1,2,4,8...). The number of simultaneous sectors times the number of CPU's per sector should not be higher then the total number of CPU's available.

Number of iterations

With the *Number of iterations* the maximal number of iterations that will be performed in the non-linear solution procedure is determined. The iteration is either stopped if the *Convergence criteria* is fulfilled (see below) or if the *Number of iterations* is reached. The number of iterations necessary to obtain a converged solution will depend on the number of cells.

Convergence criteria

The simulations are stopped automatically when a certain level of convergence is reached. For this purpose the sums of absolute imbalances of the solved variables are calculated and then normalized by a reference value. If the normalized values fall below the given convergence criteria for all solved variables the simulation will stop. The default value is 0.0005).

Convergence monitoring

Coordinate system

The spot value position is referenced to the local or global coordinate system. The local coordinate system has its origin in the lower left-hand corner of the 3D terrain model. The global system is the system specified in the *grid.gws* file. Default value is Local [-]

Spot value X and Y position

The spot value position is by default set to the centre of the terrain at ground level ($n_x/2$, $n_y/2$,1), where n_x and n_y is the number of cells in x and y-direction. The spot value position can be set anywhere in the horizontal plane. This allows monitoring of the development at points of special interest, like climatology and turbine positions. For details refer to **convergence**.

Field value to monitor

The field value allows the user to choose a solved variable and follows the development of this variable as the solution procedure progresses. The field variable is presented for the plane ($n_x/2, n_y/2$,1), where n_x and n_y is the number of cells in x and y-direction. With this information available it should be easier to introduce measures to ensure convergence even for very complex models.

Output

Height of reduced wind database

The results from the wind field simulations are stored in a compressed database consisting of flow variables, derived variables and auxiliary variables. In order to speed up the extraction of the data used in other WindSim modules a reduced database is also stored. In this database only the flow variables u, v, w and k are stored from the ground and up to the specified *Height of reduced wind database* herein. Default value is 300 m.

XY reduced wind database

Activating this option, if the horizontal grid is defined with a refinement type (Refinement area, Refinement file or Actuator Disc) the following modules is performed only on the internal part of the horizontal grid. Similarly to the vertical "Height of the reduced wind database", the solver is solving the equations on the complete grid but the outer part is disregarded by the postprocess. This option enables lighter and faster run of next modules, obtaining a uniform grid out of a refinement case.

Actuator disc

The cells covering the swept area A exert forces directed against the wind in the axial direction. For the new actuator disc method presented in Simisiroglou et al. [5] the thrust force F_i at each cell of the actuator disc is calculated from

$$F_i = \frac{1}{2} \rho \, A_i \left(\frac{U_{1,i}}{1 - a_i} \right)^2 C_T\left(U_{1,i}\right) \qquad (1)$$

Where $(U_{1,i})$, is the velocity of the flow at i-th cell of the disc perpendicular to the disc, a_i is the axial induction factor calculated for each individual cell of the disc, is the surface area of the cell facing the undisturbed wind flow direction and $C_T\left(U_{1,i}\right)$ is a modified thrust coefficient dependent on the velocity at the disc $U_{1,i}$ and ρ is the air density. In most cases wind turbine manufacturers offer C_T as a function of U_∞ the undistributed wind velocity. This C_T is reasonable for the first wind turbine of the row but not for the downstream wind turbines where the flow has been disturbed. Hence, in the present case a C_T which is a function of U_1, representing the velocity at the disc, is needed. This function can be established from the 1D momentum theory by combining the definition of the trust coefficient C_T and the axial induction factor a:

$$C_T = 4a\,(1 - a)\,, \qquad (2)$$

$$U_1 = (1 - a)\,U_\infty\,. \qquad (3)$$

Hence from Eq. (2) and Eq. (3) the following is obtained

$$U_1 = \frac{1}{2} U_\infty \left(1 + \sqrt{1 - C_T(U_\infty)}\right). \quad (4)$$

References
[1] Duran, P., Meissner, C., Casso, P. *"A new meso-microscale coupled modelling framework for wind resource assessment: A validation study."* Renewable Energy Journal, Vol. 160, pp538-554, 2020.
[2] Arne R. Gravdahl. VECTOR AS *"Meso Scale Modeling with a Reynolds Averaged Navier-Stokes Solver Assessment of wind resources along the Norwegian coast"* 31th IEA Experts Meeting – State of the Art of Wind Resource Estimation, Risø 1998.
[3] Yap, C. J. *"Turbulent Heat and Momentum Transfer in Recirculating and Impinging Flows."* PhD Thesis, Faculty of Technology, University of Manchester, United Kingdom, 1987.
[4] Yakhot, V., Orszag, S.A., Thangam, S., Gatski, T.B. & Speziale, C.G. *"Development of turbulence models for shear flows by a double expansion technique."* Physics of Fluids A, Vol. 4, No. 7, pp1510-1520, 1992.
[5] Simisiroglou, N. *"Wind power wake modelling: Development and application of an actuator disc method for industrial utilization."* PhdThesis, p.50, 2018.
[6] V.E. Henson and U.M. Yang, BoomerAMG: a Parallel Algebraic Multigrid Solver and Preconditioner, Applied Numerical Mathematics, 41 (2002), pp.155-177. UCRL-JC-141495.

Objects

Placing the objects

The **Objects** module is used to position turbines, climatologies and transferred climatologies. These objects can be placed directly using the *Park layout* interface.
Alternatively, objects can be read from a .ows files using the *Tools* → *Import objects* menu item. For visualisation purposes various geometrical objects can also be placed into the 3D terrain model by using the OpenGL Pro application. The current *3D* terrain model is accessible within the OpenGL Pro environment by clicking the *3D* button in the upper right corner of the *Report* section.

Climatology conversion

If the climatology data is not given in the WindSim formats .wws, or .tws, a conversion is necessary before the data can be used within WindSim.
One possibility is to convert your climatology data directly to the format by using your own tools, see: **climatology data**.
Alternatively, the conversion from other formats can be performed by clicking the *Tools → Convert climatology data* menu item as described below under *Climatology file*.
The *Convert climatology data* is subject to continuous development, so please contact WindSim AS if you need extensions in order to perform the conversion of your specific data, see examples of supported climatology formats: **third party formats**.

Properties

1. Object definition

Object type

Setting the object type as Turbine, Climatology or TransferredClimatology. Default value is Turbine [-].

Name

Name of the object.

Reliability

Reliability is a value between 0 and 1 that represents the trust in the climatology. The trust can vary due to the type of measurement (e.g. a measurement mast has a higher trust than a lidar) and due to the length of the measured period. If the quality of the measurements is very good (e.g. class 1 anemometer and more than a year of measurements) then the reliability should be set to 1. Accordingly, the reliability should be decreased depending on the trust. The *Reliability* is used in the **Wind Resources** and **Energy** calculation when the results are weighted against all climatologies. In both calculations the climatology weighting factor obtained from the inverse of the distance will be multiplied by the *Reliability* value. Setting the *Reliability* to 0 will cause the climatology to be disregarded in the **Wind Resources** and **Energy** calculation.

Visualisation file

Specification of a file with geometry used for visualisation. For Turbine objects it is recommended to use turbin_n (n is a height in m), which automatically generates simple scaleable geometries in accordance with the legend in the report section. Likewise for climatology and climatology transfer objects it is recommended to use climatology_n (n is a height in m). Default value depends on Object type. [-]

Climatology file

Specification of a climatology file. Climatologies can be converted from other formats by clicking the *Tools → Convert climatology data* menu item. Note that all climatology formats should be compatible with each other and wind data has to be available for all sectors specified in the climatologies to be evaluated in the **Wind Resources** or **Energy** module.
If the object type is TransferredClimatology then the specification of an existing climatology file used within the project will be transferred to a new position. Any x and y coordinates within the model is allowed, but there is a restriction on the z values. The z values must be above the first computational node, which varies according to the resolution in the given model. A warning is given if a value below the allowed minimum is given. The height of the reduced wind database, specified under the **Wind Fields** module defines the maximum possible z value. The transferred climatology will be placed in the climatology folder with the prefix trans_ added to the original file name.
The option of transferring climatologies is useful for cross checking and for successive transfer of climatologies over larger distances. By transferring a climatology to higher altitudes in a refined model, small scale effects are filtered out. Now the transferred climatology can be used in a coarser model for transfer over larger distances. Finally, the climatology can be imported again in new refined models.
Note that transferred climatology objects within a model are not used in the **Wind Resources** and **Energy** modules, as they don't provide any new information. A climatology transferred from an external model will provide new information, these climatologies can be included as ordinary climatology objects. Default value is none [-].

Power curve file

Specification of a power curve file in WindSim format .pws. If the power curve for a certain turbine type is not available in the WindSim database ("%ProgramFiles%/WindSim/WindSim 12.0.0/Data/Objects/Power Curve") you can write it with *Tools → Create power curve (.pws)* or in a text editor taking into account the required **syntax**. Default value is none [-].

Hub height

Turbine hub height in [m]. Default value is 80 m.

Rotor diameter

Turbine rotor diameter in m. Default value is 60 m.

Rotation speed

The rotational speed of the turbine blades or the climatology anemometer. Default value is 10 (rounds per minute) for turbines and 25 (rounds per minute) for climatologies.

Facing wind direction

The direction to which the turbines surface is aligned perpendicular can be determined here. The angle needs to be specified in degrees measured clockwise from North. The default value is 180 degrees, meaning that the turbine is oriented towards the South. This option is only important for visualization purposes.

Number of Sectors

In order to process climatology data given in a .tws format **WindSim** rewrites this information into a frequency distribution format (see **climatology data**). The *number of sectors* that are used in this frequency distribution can be specified here.

Number of Bins

In order to process climatology data given in a .tws format **WindSim** rewrites this information into a frequency distribution format (see **climatology data**). The *number of bins* that are used in this frequency distribution can be specified here.

2. Position

Coordinate system

The position of an object is always referring to a certain reference frame, here given by either the local or global coordinate system. The local coordinate system has its origin in the lower left-hand corner of the 3D terrain model (see **Terrain**) while the global system is the system specified in the *grid.gws* file. Default value is Local [-].

X and Y position

The X and Y position of the objects according to the specified coordinate system. For Climatology objects additionally the z position is shown, the position for climatologies cannot be changed because it is already specified in the .wws file. The .wws file can be opened in a text editor by clicking the *Tools → View climatology data (.wws)* menu item.

3. Noise calculation

Based on broadband

A map of sound pressure level $(Lp)[dB(A)]$ is calculated at the *Noise map height*, optionally accounting for the background sound level. The noise level is calculated for a particular wind

condition specified by the user. Therefore a wind speed at a given height has to be specified by the user. WindSim will calculate for the assigned wind condition the wind speed at each hub position, according to the **Wind Fields** calculations, and the relative sound power level Lw. To estimate the power level, the wind speed at the hub is transferred to the reference height assigned in the .pws file by using a power law with a wind-shear exponent of 1/7.
The sound pressure level $(Lp_i(x,y))$ at a generic point $(P(x,y))$ produced by the i-th wind turbine is given by:

$$Lp_i(x,y) = Lw_i - 10\log\left(2\pi d_i^2\right) - \alpha\, d_i$$

where:
(Lw_i) - broadband sound power level $([\mathrm{dB(A)}])$ of turbine i,
(d_i) - distance of $(P(x,y))$ to the i-th turbine [m],
(α)- broadband attenuation coefficient $([\mathrm{dB(A)/m}])$
While the sound pressure levels due to the different wind turbines, and eventually the background sound pressure level, are summed according to the decibels rules.

$$Lp(x,y) = 10\log\left(\sum_i 10^{0.1\,Lp_i} + 10^{0.1\,Lp_{bg}}\right).$$

Noise map height

The height at which the sound map is extracted. The map height [m] is given as a real number.

Background noise

Optional background noise for the sound pressure level calculation [dB(A)]. A default value of 0 will produce a map not accounting for background noise.

Attenuation coefficient

Broadband attenuation coefficient [dB(A)/m] to account for the noise absorption of the air.

Noise at height

Height above ground level at which the wind speed and direction condition is assigned [m].

Noise coordinate system

Specify which coordinate system Global/Local is used to assign the X and Y coordinate at which the wind conditions (wind speed and direction at *Noise at height*) are given.direction at *Noise at height*) are given.

Conditions X position

X (easting) coordinate of the point used to specify the wind conditions.

Conditions Y position

Y (northing) coordinate of the point used to specify the wind conditions.

Noise at wind speed

Wind speed condition at which the noise map is calculated ($[\mathrm{m/s}]$).

Noise at wind direction

Wind direction condition, measured clockwise from north, at which the noise map is calculated ($[\mathrm{deg}]$).

4. Terrain Complexity

Terrain complexity calculation is performed at each turbine position and climatology position, following the IEC 61400-1:2019 [1]. The terrain points near the turbine position are divided into sectors of different radius and angle amplitude: 5 hub height per 360-degree amplitude and 5,10 and 20 hub height per 30-degree amplitude. For each sector a plane is fit to the points (the plane is not passing through the tower base), then the inclination angle (slope) is detected and the standard deviation of the terrain variation is calculated by the distance of terrain points from the plane.
The TSI (Terrain Slope Idex) and TVI (Terrain Variation Index), as defined in the IEC standard, are obtained by using this information and the energy frequency per direction. The energy frequency that is used is obtained by the first climatology of the layout. TSI and TVI are compared to category limits to define Low, Medium and High (L, M and H) complexity categorization. The highest category of all the sectors cosidered is defined as the complexity category at that position. More details on the calulation are availble in [1].
The results of the terrain complexity following IEC 61400-1:2019 are stored in the layout subfolder "/report/TerrainComplexity" and displayed via the links in object module report. The calculation of complexity Indexes according to previously defined standards are still perfomed and the results are available in the layout subfolder "/report/TerrainComplexityPreviousCalc", see **terrain complexity** for more details.

---

[1] International Electrotechnical Commission (IEC). IEC 61400-1 ed4.0 (2019), Wind turbine generator systems - Part 1: Safety requirements [chapter 11.2.1 Assessment of the topographical complexity]

Results

Exploring the wind database

The results of the wind field simulations are stored in a reduced database covering the vertical extension from ground up to the *Height of reduced wind database* as specified in the **Wind Fields** module. The **Results** module extracts 2D horizontal planes from this database.

For inspection of the 3D set, use the *3D Visualisation → Generate/Open 3D wind visualisation file (.vtf)*

Properties

1. Normalisation

Normalisation variable

The following variables are available:

Speed scalar X - wind speed scalar in East-West direction, $(u\,[\mathrm{m/s}])$.

Speed scalar Y - wind speed scalar in North-South direction, $(v\,[\mathrm{m/s}])$.

Speed scalar Z - wind speed scalar in vertical direction, $(w\,[\mathrm{m/s}])$.

Speed scalar XY - wind speed scalar in horizontal plane, $(\sqrt{u^2+v^2}\,[\mathrm{m/s}])$.

Speed scalar XYZ - wind speed scalar in 3D space, $(\sqrt{u^2+v^2+w^2}\,[\mathrm{m/s}])$.

Velocity vector XY - wind speed vector in horizontal plane, $((u, v, 0))$.

Velocity vector XYZ - wind speed vector in 3D space, $((u, v, w))$.

Direction scalar - wind direction in horizontal plane $([\mathrm{deg}])$.

Direction scalar relative - wind direction in horizontal plane relative to incoming wind direction $([\mathrm{deg}])$.

Turbulent kinetic energy - Turbulent kinetic energy, $(k\,[\mathrm{m^2/s^2}])$.

Turbulence intensity - Turbulence intensity assuming isotropic (k), $(100\sqrt{(4\,k)/[3\,(u^2+v^2)]})[\%]$.

Turbulent dissipation rate/Turbulent Frequency $(\epsilon/\omega\,[\mathrm{m^2/s^2}])$.

Pressure - relative pressure: fixed zero at sea level, minus the hydrostatic term, $([\mathrm{Pa}])$.

Inflow angle - angle with respect to the horizontal, $(\arctan\left(w/\sqrt{u^2+v^2}\right)\,[\mathrm{deg}])$.

Wind shear exponent – exponent (α) of the equation $(\mathrm{speed}_1/\mathrm{speed}_2 = (\mathrm{height}_1/\mathrm{height}_2)^{\alpha})[-]$.

Normalisation type

The following normalisation types are available:

Not normalized - Present results with no normalization.

Normalize against climatology - Normalization against the first visible Climatology object.

Normalize against scalar value - Normalize against the scalar value specified in *Scalar value*.

Scalar value

Value used in normalization when *Normalization type* is *Normalize against scalar value*.

2. Planes

Heights

The *Heights* above ground level for which the results should be generated. Only heights below the *Height of reduced wind database* specified in the **Wind Fields** module are valid. Multiple heights can be given as a semicolon separated list, e.g. 50;60;70.

Sector angles

The sector angles for which the results should be generated. Only sector angles where wind field simulations have been performed are valid. Several sector angles can be given as a semicolon separated list, e.g. 0;30;60;90.

3. Legend

Legend minimum and maximum values

Specification of the legend interval, if both minimum and maximum are set to zero the full range will be given. Default value is zero [-].

Wind Resources

Accumulation of results - the wind resource map

Before running the **Wind Resources** module at least one climatology must exist and all sectors defined in that climatology must exist in the wind database. The wind resource map is established by weighing the wind database against the climatology. If several climatology objects are available the wind resource map will be based on them all, by interpolation of the inverse distance to the climatology objects.

The **Wind Resources** module contains a tool for area classification. Finding high speed connected areas where the areas are grouped according to the wind speed and size. The possible power production in the area is also estimated.

Properties

1. Wind resource map

Heights

Here the height for which the results should be generated has to be specified in units of meter [m]. With a semicolon separated list there is also the possibility to generate results for multiple heights. Note that only heights below the "Height of reduced wind database" in the **Wind Fields** module are valid. Default value is 50 [m].

Sector interpolation

The wind direction for a simulation in the wind database refers to the wind direction at the inlet. Due to terrain effects the wind direction changes in the inner of the model. In order to weight the wind database against a climatology, an interpolation is required to reproduce the sectors of the climatology at the climatology position, see example in figure 9. A sector interpolation for all sectors in the climatology is performed when sector interpolation is True. The default value is True [-].

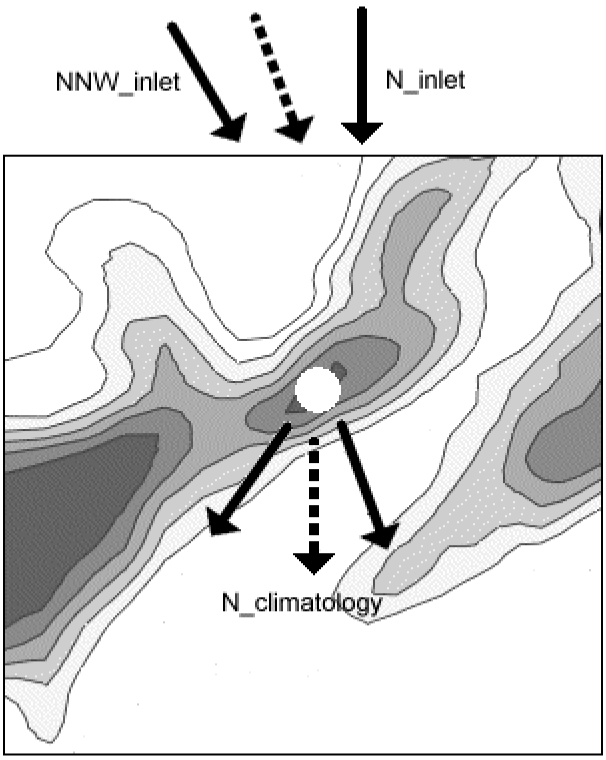


Figure 9. Example of sector interpolation, where interpolation of incoming wind from North-northwest and North give wind from North at the climatology location.

Wake effects

Wake effects can either be assessed by analytical models or by CFD based methods itself. Regarding a CFD based approach, WindSim provides the modeling of a single turbine within an actuator disc approach which is directly included into the CFD calculation. In order to setup a calculation containing the actuator disc model one has to return to the **Terrain** module an choose the actuator disc refinement option as described in the **Terrain** description.

Analytical methods are attractive since they are simpler and less computational demanding than CFD based methods. In the following the three analytical wake models that can be used in WindSim are described, they are all single wake models calculating the normalized velocity deficit $(\delta v = (u - v)/u)$, as shown in figure 10. Since the area which is influenced by wake effects is modeled for instance as a cone, all models are rotationally invariant as represented in figure 10, where the x-axis is shown at hub height.

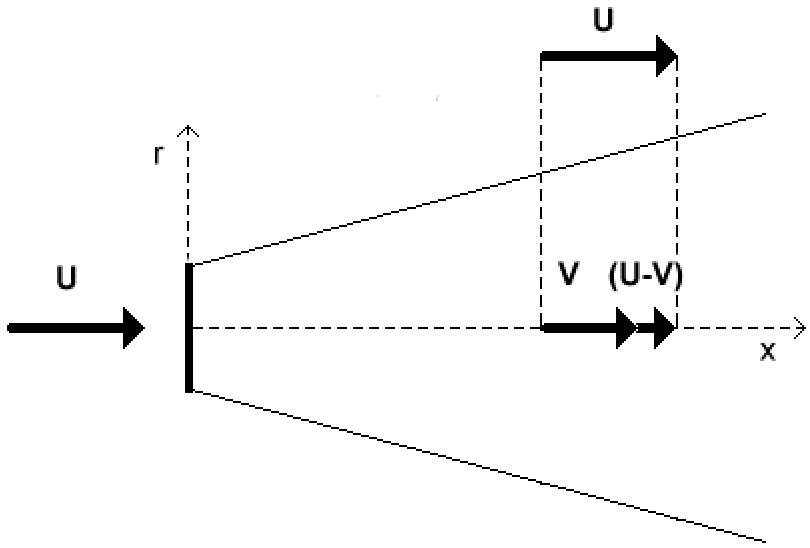


Figure 10. Definition sketch of wake effects.

Since the velocity deficit is calculated based on the wind database established in the module **Wind Fields** the analytical wake modeling is a post-processing procedure. Consequently, the mutual interaction between the wakes as well as the interaction between the wakes and the terrain can not be captured correctly within this approach. Alternatively, each turbine can be modeled as a disc within the actuator disc approach in the **Terrain** module. The presence of all turbines can therefore be included in the CFD simulation and wake-wake effects as well as terrain-wake effects can be considered.

Model 1

Model 1 is based on momentum deficit theory and is often referred to as the "Jensen model" [1]. This model gives a simple linear expansion of the wake, determined by the wake decay factor kk. The wake decay factor increases with increasing level of ambient turbulence, a typical range is from 0.04 to 0.075

$$\delta V = \frac{1}{9}\,(C_T\,A_r\,x^{-2})^{1/3}\left[r^{3/2}\,(3\,C_1^2\,C_T\,A_r\,x)^{-1/2} - (35/2\pi)^{3/10}(3\,C_1^2)^{-1/5}\right]^2$$

Where

C_T - thrust coefficient

$k = A/l\,n(h/z_0)$

$A \ = \ 0.5$

h - hub height [m]

z_0 - roughness height [m]

Model 2

Model 2 is derived from the turbulent boundary layer equations and a similarity assumption. It is often referred to as the "Larsen model"[2].

$$\delta V = \frac{1}{9}\,(C_T\,A_r\,x^{-2})^{1/3}\left[r^{3/2}\,(3\,C_1^2\,C_T\,A_r\,x)^{-1/2} - (35/2\pi)^{3/10}(3\,C_1^2)^{-1/5}\right]^2$$

where

C_T - thrust coefficient [-]

$A_r = \pi\,D^2/4$

D - rotor diameter

$C_1 = (D/2)^{5/2}(C_T\,A_r\,x_0)^{-5/6}$

$x_0 = 9.5\,D^4/(2\,R_{95})^3 - 1$

$R_{95} = 0.5\,(R_{nb} + \min(h, R_{nb}))$

$R_{nb} = \max\big(1.08\,D, 1.08\,D + 21.7\,D\,(I_a - 0.05)\big)$

I_a - ambient turbulent intensity at hub height

Model 3

This model introduces a turbulent depending rate of wake expansion [3].

$\delta V = C_T^{1/2}\,(1.666/k_1)^2\,(x/D)^{-p}\,\exp(-r^2/b^2)\,/32$

where

C_T - thrust coefficient [-]

$b = k_1\,\left(C_T^{1/4}/0.833\right)\,D^{(1-p/2)}\,x^{p/2}$

D - rotor diameter

$p = k_2\,(I_a + I_w)$

$I\_w \ = \ k\_3\ \ (C\_T/\max\,(I\_a, 0.03))\,(1 - \exp(-4\,(x/10\,D)^2))$

I_a - ambient turbulent intensity at hub height

$k_1 = 0.27$

$k_2 = 6.00$

$k_3 = 0.004$

Wake Decay Factor

If Model 1 is chosen to simulate wake effects the corresponding wake decay factor k as described above can be manually specified to a fixed value. If the default option *Automatic* is active the wake decay factor is determined as described in the corresponding model description.

Roughness

Here the roughness height at the position of the turbine should be given as an input parameter to wake Model 1. This value can either be read from the .gws file or it can be specified manually as a constant value. By default the roughness height is read from the .gws file.

Ambient turbulence intensity

The ambient turbulent intensity at the turbine position is an input parameter to wake Model 2 and wake Model 3. This value can either be read from the wind database or it can be specified manually as constant value. By default the ambient turbulent intensity is read from the wind field database.

Number of sub-sectors

In order to refine the calculation of wakes for one sector the sector is additionally divided into sub-sectors for which the wakes are calculated. The total wake representative for the sector is subsequently calculated by an appropriate average. The default *Number of sub-sectors* is 30[-].

Influence range

The influence range in units of rotor diameters determines the area where the wake calculation is performed. Here one has to specify a minimum and a maximum value (seperated by a semicolon) corresponding to the lower and upper radial limit of the waked area. The lower limit is used to disregard wake effects in the near-field that might not be represented correctly by the considered wake model. The upper limit is given for computational reasons, to avoid calculations in the far-field where wake effects could be neglected. The default range is (1;50) [Rotor diameter].

Multiple wakes model

When more than one turbine influence the velocity at the considered location the velocity deficits calculated by the analytical single wake models are combined to obtain a total wake deficit. In the literature several multiple wake models have been proposed to describe the combined wake effect on the basis of the single wake contributions. In WindSim the linear superposition of the wake defects (1) and the square root of the sum of the squares (2) is supported. The total wake deficit δv is composed of the single velocity deficit δv_i associated to one turbine i, where the sum over all turbines is performed as follows.

(1) Based on linear superposition $\delta v = \sum_i \delta v_i$

(2) Based on sum of squares of velocity deficits $\delta v = \sqrt{\sum_i \delta v_i^2}$

Air density correction

Activating Export to WAsP format the wind power density calculation is activated and density calculation is done in every grid point. Choose the desired density correction. The default value is "no correction".

No correction

The density at all points equals the density in the power curve file.

Fixed Value

The density at all points equals the given value.

Individual 1

The density is calculated at each point from the U.S. Standard Atmosphere considering elevation changes $\rho = 1.225 - (1.194 \cdot 10^{-4})\, z$, where z is the height above sea level. This will give a good long-term average value of air density in moderately complex areas.

Individual 2

The density is calculated at each point from an isothermal atmosphere approximation to account for elevation changes and mean site temperature $\rho = \frac{p_0}{R \cdot T} \exp\left(\frac{-g}{R \cdot T} z\right)$, where $p_0 \simeq 1013.25\,\text{hPa}$ is the standard sea level atmospheric pressure, $g \simeq 9.8\,\text{m/s}^2$ is the gravitational constant, T is the temperature in units of Kelvin as specified under *Reference temperature*, R is the gas constant and z the height above sea level.

Individual 3

The density is calculated at each point from the standard atmosphere with an assumed temperature lapse rate that is determined by the input parameter *Temperature gradient* below. The pressure is again assumed to be $p_0 \simeq 1013.25\,\text{hPa}$ at sea level. Additionally, a reference temperature at a certain reference height needs to be specified.

- Reference height:
  Here the *Reference height* associated to the indicated *Reference temperature* has to be determined. Default value is 0[m].

- Reference temperature
  Here the *Reference temperature* as input parameter for an *Air density correction* of the type *Individual 2* or *Individual 3* has to be specified. As reference temperature the mean annual measured temperature can be used. Default value is $288.15\ [\mathrm{K}]$.
- *Temperature gradient*
  Here the *Temperature gradient* which is used for the *Air density correction* as described under *Individual 3* should be given. Default value is $-0.0065\ [\mathrm{K/m}]$.

Distance weighting

If more than one climatology is present, the wind resource maps named "All" are combining the results of the single climatologies using as weighting factor the inverse of the distances between climatologies and grid node. In this way the results for the climatology closer to the node influences the result at the node more. This is applied, by default, using the simple invers of the distance weighting r_0/r where r is the distance and r_0 the appropriate normalization factor. Modifying the distance weighting value changes the exponent that will be applied. Values higher than 1increase the weights of the closest climatologies. E.g. if the distance weighting is 2 then r_0^2/r^2 is used to define the weighting, 3 will use r_0^3/r^3 . The climatology weighting factor obtained from the inverse of the distance will be multiplied by the reliability to find the actual weighting factor of a climatology to a grid node. See also description of Reliability under Objects.

2. Legend

Legend minimum and maximum values

Specification of the legend interval, if both minimum and maximum are set to zero the full range will be given. Default value is zero [-].

3. Export

Export to ASCII format

By enabling the *Export to ASCII format* option the wind resource map is exported as an ASCII file. The annual mean wind speed, TI, inflow angle, wind shear exponent, and TI 15m/s is given. A link to the file will be provided in the report. The default value is False [-].

Export to WAsP format

Exporting the wind resource map to the WAsP .rsf and .wrg formats. A link to the files will be provided in the report, likewise, plots of the Weibull scale and shape parameters. The default value is False [-].

Export all

Exporting the WAsP files weighted against all climatologies. This procedure is quite time and memory demanding and should only be activated if it is needed. The default value is True [-].

Type

The WAsP files can be exported for different areas: the complete grid, the refinement area if there is refinement, and a user defined area. For the user defined area, the range in global coordinates and the horizontal resolution can be specified.

X-range

Here the *X-range* of the area that should be exported can be indicated referring to the global coordinate system.

Y-range

Here the *Y-range* of the area that should be exported can be indicated referring to the global coordinate system.

Resolution

Here the *Resolution* of the area that should be exported can be indicated. By default a *Resolution* of 100m100m is assumed.

Export to Surfer format

Exporting the wind resource map to .grd formats. A link to the files' folder will be provided in the report. The exported files are available in "*project**layout*\energy\grd\" and "*project*\windfield\grd\". Activating WAsP Export, "Complete Grid" or "Refinement area", more maps are exported: Weibull-A and k and Sector frequency. The default value is False [-].

4. Cross-checking

The purpose of the wind flow simulations in wind resource assessment is to extrapolate the wind measurements. These extrapolations consist in obtaining the wind speed at a target location T (typically a wind turbine) by multiplying the measured wind speed at a reference point R by a factor (Figure 11.1b). This factor is called the speed-up ratio SU, and it is calculated as $SU(R,T) = u_T/u_R$where u_R and u_T are the modeled wind speeds at points R and T (Figure 11.1a).

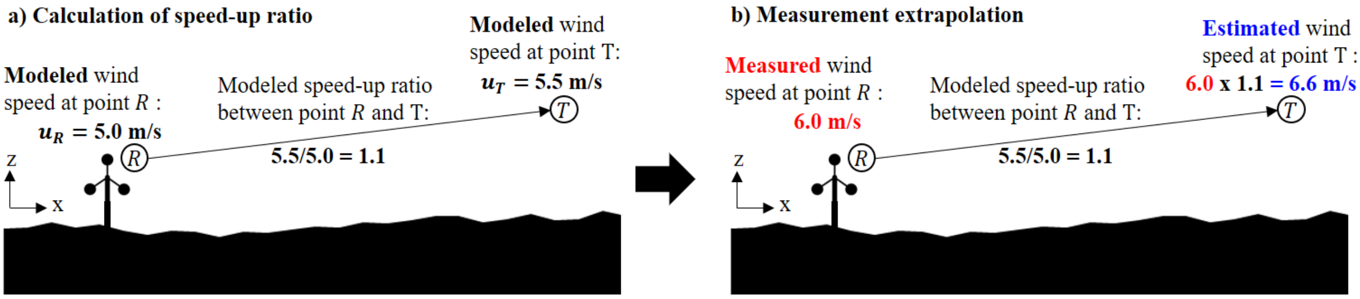


Figure 11. Example for the calculation of the speed-up ratio (a) and the measurement extrapolation (b).

Therefore, the performance of a model is quantitatively evaluated by comparing the modeled SU with the measured one, between selected pairs of measurements. Specifically, the mean values of SU and wind speeds are utilized to calculate the so-called crosscheck prediction $XPE(R,T) = \frac{\overline{SU(R,T)} \cdot \overline{u_{M,R}} - \overline{u_{M,T}}}{\overline{u_{M,T}}}$. It is possible to calculate one XPE per pair of measurement points.

The cross-checking functionality now estimates the errors of the numerical model in predicting measurements on the basis of XPE values. Whenever multiple measurements exist, represented as climatologies with the given time history format **.tws**, a cross-checking can be performed. All possible climatology pair combinations are then cross-checked automatically, defining one climatology as the reference and the other climatologies as targets. In a project with for instance three climatologies all six combinations will be cross-checked as shown in figure 12.

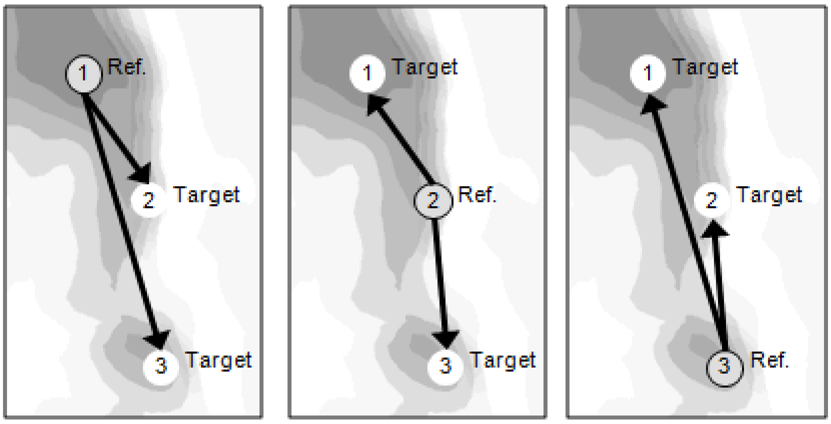


Figure 12. Cross-checking combinations in a project with three climatologies.

The cross-checking is based on concurrent time series. It calculates the ratios of the wind speeds (speed-ups) and ratios of the standard deviations for all concurrent time records of one reference and one target. These ratios are calculated in two different ways; filtered and unfiltered. The filtered way includes wind speeds inside the operational range of a typical wind turbine, the default range is from 3 to 25 (m/s). Whereas the unfiltered way includes all wind speeds except calm conditions when the wind speed is zero.

Hence two sets of errors are estimated based on filtered and unfiltered data. The errors obtained by filtering should be closer to the errors obtained from the numerical model for the operational range of wind speeds for a wind turbine. Results presented in the report tables are based on filtered data. In the underlying files linked in the report there is also unfiltered results.

Wind Speed

With this option it is possible to limit the range of wind speeds that should be used for cross-checking. The wind speed range can be determined at the reference point or at the target point or for both.

Wind speed ref

By specifying a certain wind speed range, the cross-checking procedure filters for wind speeds at the reference point that are within this range.

Wind speed target

By specifying a certain wind speed range, the cross-checking procedure filters for wind speeds at the target point that are within this range.

Wind Speed st. dev

With this option it is possible to limit the range of the standard deviation of the wind speeds that should be used for cross-checking. The range of the standard deviations can be determined at the reference point or at the target point or for both.

Wind Speed st. dev ref

By specifying a certain range of standard deviation, the cross-checking procedure filters for the standard deviations at the reference point that are within this range, all others are neglected.

Wind Speed st. dev target

By specifying a certain range of standard deviation, the cross-checking procedure filters for the standard deviations at the target point that are within this range, all others are neglected.

References

[1] Katic, I., Højstrup, J., Jensen, N.O. *"A Simple Model for Cluster Efficiency."* EWEC Proceedings, 7-9 October 1986, Rome, Italy.

[2] Larsen, C. G. *"A Simple Wake Calculation Procedure."* Risø-M-2760, 1988. (http://www.risoe.dk/rispubl/VEA/veapdf/ris-m-2760.pdf)

[3] Ishihara, T., Yamaguchi, A., Fujino, Y. *"Development of a New Wake Model Based on a Wind Tunnel Experiment."* Global Wind Power 2004. (http://windeng.t.u-tokyo.ac.jp/posters/2004_gwp_poster.pdf)

Energy

The annual energy production (AEP)

The annual energy production, AEP is calculated for all visible Turbine objects. If several climatology objects are available the AEP based on each climatology is calculated separately. Any discrepancies between the AEP's based on different climatologies are easily accessible. A climatology is given by a time series, or its frequency distribution which presented graphically in the wind rose. Additionally a climatology is given by its Weibull distribution. The AEP is calculated for all representations.

Properties

1. Calculations

Air density correction

In general, every power curve is given for a specific air density. If the air density given in the power curve of the considered turbine is assumed to differ from the air density at the position of the turbine a correction can be applied with following approaches. By default *No correction* will be applied.

No correction

The given power curve is not corrected. If the density at the turbine position equals or is approximately equal to the density specified in the power curve this approach should give proper results, otherwise one of the following corrections should be chosen.
Fixed value, Individual 1, Individual 2 & Individual 3
See description page of **Wind Resources** .

Method for density correction

Two different correction methods can be applied to the power curve depending on the power control system of the WECS (EN 61400-12).

Pitch-regulated WECS

In the case of pitch-regulated wind turbines the power output of the WECS is calculated by entering in the original power curve with a corrected wind speed. The corrected wind speed is obtained by the wind speed times the fraction (air density AEP)/(air density power curve) at the power 1/3.

Stall-regulated WECS

In the case of stall-regulated wind turbines the fraction (air density AEP)/(air density power curve) is used in the AEP calculations as a multiplication factor of the reference power curves.

Sector interpolation

See description page under **Wind Resources** .

Wake effects

See description page under **Wind Resources** .

Height of reference production

By default the energy production is calculated at the measurement mast position in the measurement height and in 80m above ground level. This *Height of reference production* can be changed to any value and additional heights can be added in terms of a semicolon separated list, i.e. 80;100.

Rotor equivalent wind speed

If the rotor equivalent wind speed (REWS) calculation is activated, the wind speed at turbine position is not only calculated at hub height, additional heights in the rotor area are taken into account. The total number of heights contributing to the REWS depends on the number of fractions defined under *Number of fractions*. To compute the rotor equivalent wind speed, the wind speeds of all contributing heights are weighted according to the surface fraction that belongs to each height, as shown in Figure 13. For example, if the *number of fractions* is 3, the REWS will be calculated as a weighted sum of the speeds at the hub height, hub height +2/3 radius of the turbine and hub height −2/3 radius of the turbine. The weighting depends on the surface portion of the circle belonging to each height (see Figure 1). The same logic is applied to 5,7 and 9 rotor fractions cases.

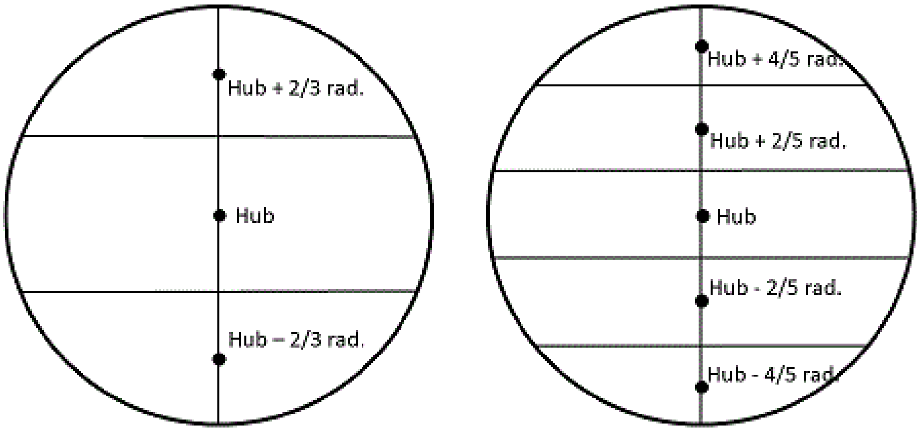


Figure 13. Definition sketch of rotor fractions for rotor equivalent wind speed calculation.

Distance weighting

See description page of **Wind Resources** . In the energy calculation the distance is calculated between climatology and turbine position at hub height.

Manual weighting

For the energy calculation weighted against all climatogies, the weighting of the results of the single climatologies is given manually in a table for each turbine. The values reported by default are obtained using the inverse of the distance with the exponent defined in the *Distance weighting* parameter and the *Reliability* parameter of the climatology (see **Objects** module). To weight a turbine only by a single climatology, modify the line of the turbine and set the values of that climatology to 1 and the values of all the other climatologies to 0. After modification save the file by pressing ok. Values between 0 and 1 will lead to manual weighting. A value of -1 in a line will reset to the default calculation for that turbine using the weighting factors obtained from the inverse of the distance multiplied by the reliability.

2. Export

Export power history

The visible climatologies of type time history (.tws) are transferred to the hubs of each visible turbine; the relative .tws files are placed in the object folder of the project. ASCII files containing the electrical power output of each WECS for each record of the time history files are furthermore created and saved into the report folder. A link to the power history files will be provided in the report.

Export weighted Power history

All time histories transferred to the hub height of each visible turbine used to calculate power output (“powerhistory_clim…_wes_....dat”) are loaded and weighted with the inverse distance. By this procedure a weighted time series is obtained and associated ASCII files which contain the electrical power output are created. A link to the power history files will be provided in the report section called “powerhistory_clim_all_to_wecs_....dat/.csv”.
Note: the time series of the whole layout, i.e. the time series of each climatology need to be consistent with each other. They must be of the same length, with the same time step as well as providing the same time period.

Export rotor profiles

An ASCII file with rotor profiles at all visible Turbine positions is produced. Data is extracted from the wind database in the corners of a square centred at the hub. A link to the file will be provided in the report.

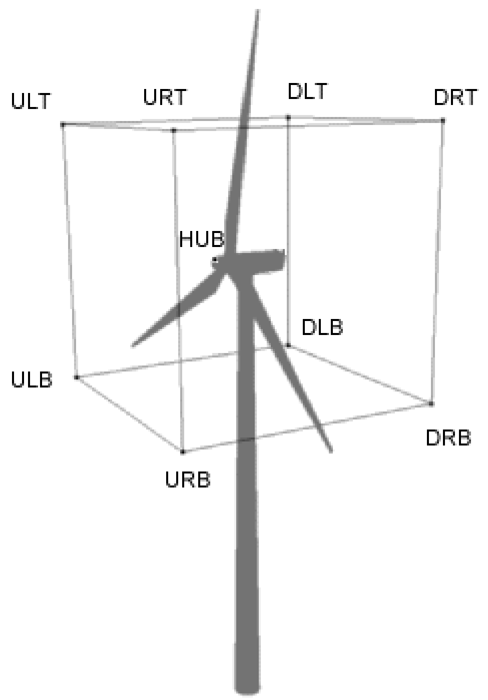


Figure 14. Definition sketch of rotor profiles.

The rotor profile file contains the same variables as the vertical profiles, see above. The variables are given at the hub as well as in the corners of a square centred at the hub, the extension of each side is equal to the rotor diameter as determined in the module **Objects** . The following syntax is used for the corners (compare to Figure 14): Upstream (U), Downstream (D), Right (R), Left (L), Top (T) and Bottom (B). The upstream plane of the cube is perpendicular to the incoming wind direction. The position in the vertical direction is referred to as: above sea level (asl), above ground level (agl) and referred to HUB (rth). The default value is False [-].

Export turbine assessment

By activating the option *Export turbine assessment* important sectorwise parameters at the turbine positions are written into a file. The basic information that are included are the annual energy production (without and with wakes losses), the Weibull parameters and the wakes losses. In case the IEC classification is executed additional parameters are written as the representative turbulence intensity (Irep), the effective turbulence intensity (Ieff), the wind shear exponent and the inflow angle.

Export vertical profiles

An ASCII file with vertical profiles at all visible Turbine positions is produced. The profiles are extracted from the wind database from ground level up to the *Height of reduced wind database* specified in the **Wind Fields** module. A link to the file will be provided in the report.

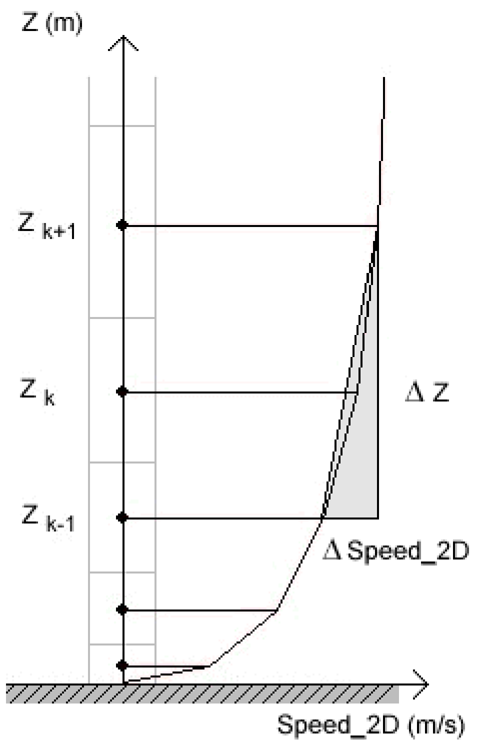


Figure 15. Definition sketch of vertical profiles, e.g. Speed_2D. Variable values are given in cell centres.

The vertical profile file contains the following variables:

u - wind speed scalar in East-West direction $[\mathrm{m/s}]$

v - wind speed scalar in North-South direction $[\mathrm{m/s}]$

w - wind speed scala in vertical direction $[\mathrm{m/s}]$

$\mathrm{Speed\_2D}$ - wind speed scalar in horizontal plane $\sqrt{u^2+v^2}$ $[\mathrm{m/s}]$

Inflow - angle with respect to the horizontal, $\arctan\left(w/\sqrt{u^2+v^2}\right)$, $[\mathrm{deg}]$

Shear - derivate of $\mathrm{Speed\_2D}$ with respect to vertical direction $\Delta\mathrm{Speed\_2D}/\Delta z$ $[1/\mathrm{s}]$, where
$\Delta\mathrm{Speed\_2D} = \mathrm{Speed\_2D_{k+1}} - \mathrm{Speed\_2D_{k\text{-}1}}$ and $\Delta z = z_{k+1} - z_{k-1}$

$\mathrm{Shear\_low}$- derivate of $\mathrm{Speed\_2D}$ with respect to vertical direction, $\Delta\mathrm{Speed\_2D}/\Delta z$ $[1/\mathrm{s}]$, where
$\Delta\mathrm{Speed\_2D} = \mathrm{Speed\_2D_{k}} - \mathrm{Speed\_2D_{k\text{-}1}}$ and $\Delta z = z_{k} - z_{k-1}$

$\mathrm{Shear\_high}$ - derivate of $\mathrm{Speed\_2D}$ with respect to vertical direction, $\Delta\mathrm{Speed\_2D}/\Delta z$ $[1/\mathrm{s}]$, where $\Delta\mathrm{Speed\_2D} = \mathrm{Speed\_2D_{k+1}} - \mathrm{Speed\_2D_{k}}$ and $\Delta z = z_{k+1} - z_{k}$

KE - turbulent kinetic energy $[\mathrm{m^2/s^2}]$

TI - turbulent intensity assuming isotropic KE, $100 \cdot \sqrt{4\,KE/3}/\sqrt{u^2+v^2}$ $[\backslash\%]$

α - wind shear power exponent $\alpha_k = (\alpha_{k-1} + \alpha_{k+1})/2$ [-] with

$$\alpha_{k-1} = \ln(\mathrm{Speed\_2D_{k\text{-}1}}/\mathrm{Speed\_2D_{k}}) \,/\, \ln(z_{k-1}/z_k)$$
$$\alpha_{k+1} = \ln(\mathrm{Speed\_2D_{k+1}}/\mathrm{Speed\_2D_{k}}) \,/\, \ln(z_{k+1}/z_k)$$

The default value is False [-].

3. IEC Classification

In this option of the **Energy** module the suitable IEC class for each turbine of the current layout is calculated, as described in the standards IEC 61400-1 [2,3,4]. The IEC classification of the turbines is performed for both the 2nd edition [2] and 3rd edition [3] of the standards, accounting for the amendment 1 (2010) [4] of the 3rd edition. Each .tws is transferred to each hub position, and then the parameters Vref, Iref (mean value and standard deviation) are computed for the transferred .tws. The reference velocity, an extreme wind (10 min average) with a recurrence period of 50 years, is computed by a Gumbel fitting the peaks of transferred wind speeds. The Gumbel fit can be performed in different ways depending on the way to detect peaks and the fit method applied: Annual maxima with Harris '96 method [5], Independent storms with Harris '99 method [6] and Annual maxima with maximum likelihood method. The methods of fitting are described in details in the relative publications, Annal maxima peaks detection uses one value per year, the maximum speed measure in the year, at least two year of data are needed to enable the fit, but is suggested to have longer time series to make the fit more meaningful. Independent storms method defines the storms as periods exceeding the Storms threshold (m/s) and peaks as their maximum values. To ensure that the peaks are not part of the same storm event the Storms window length (h) value can be tuned: two peaks are considered part of separate storm event if the distance in time is higher than the windows length, if not the higher between the two is used in peaks fit. In case the time stepping of the reference time series (.tws) is not 10 minutes there will be need to change the *Gust factor* accordingly. For the 2nd edition the IEC class is determined also according to the site annual average wind speed Vave. For some sites the restriction over the Vave is more severe than the one on the extreme wind speeds. The 3rd edition substituted the verification over the average wind speed with a more detailed verification of the site probability density function (pdf) over the turbine design pdf, which is a Rayleigh distribution. The check of the site pdf against the design pdf is not present in WindSim as output of the IEC classification, though it can be carried out analyzing the site through the WAT export or the turbines_assessment files. The Iref is given as expected value of the turbulence intensity for the 15 m/s bin, its standard deviation is also computed for the same samples. In the case of classification of wind turbines according to the second edition of the standards [2] a characteristic turbulent intensity is required (84th percentile, mean plus the standard deviation for a normal distribution). For the 3rd edition, as amended in the 2010, the turbulence class is instead determined by veryfing equation (35) of [4] in the range *Wind speeds range Ieff*. All values of Ieff are reported in the energy_IEC_classification.log file which is linked in the report of the "Energy.html" target="main" class="module"> Energy module. In the case of classification according to the third edition, it is verified that the standard deviation of the longitudinal component from the normal turbulence model σ_1 of the wind velocity at hub height is greater or equal to the estimated 90th percentile of the effective turbulence standard deviation (accounting for both ambient and wake turbulence)

$$\sigma_1(V_{\mathrm{hub}}) \geq I_{\mathrm{eff}}(V_{\mathrm{hub}}) \cdot V_{\mathrm{hub}}$$

where

σ_1 - standard deviation of the longitudinal component of the wind velocity,

V_{hub}- wind speed at the hub height,

I_{eff} - effective turbulence intensity.

The verification is performed between for V_{hub} ranging from 60% of the rated wind speed V_r and the cut-out wind speed V_{out}. In the case rated and cut-out wind velocity are not specified in the power curve (.pws) file, the verification will be carried out automatically for V_{hub} varying in the range 7−25 m/s if a user defined wind speed range is not given.

The effective turbulence I_{eff} is defined in the IEC standards as weighted average of turbulence intensity to the power of m, being m the Wöhler (SN-curve) exponent. For the IEC classification m is by default equal to 10, number valid to verify the glass fiber of the blades, the most fragile component of the wind turbine. The effective turbulence is therefore that turbulence intensity that would produce fatigue damage after the same number of cycles of a failure caused by the actual turbulence wind rose.

$$I_{\text{eff}} = \left[\int p(\theta|V_{\text{hub}})\, I^m(\theta|V_{\text{hub}})\, d\theta\right]^{1/m}$$

here

m - Wöhler exponent,

θ - wind direction,

p - frequency of occurrence,

I - representative value (90th percentile) of the turbulent intensity of the wind,

V_{hub} - wind velocity at the hub height.

The turbulence intensity I accounts for the presence of the neighboring turbines with the version of the Frandsen model as proposed in the Annex D of the last standards amendment [4].

The integral defining I_{eff} is therefore discretized in a set of wind directional sectors. Using the subscript s for sector-wise properties, NS number of directional sectors:

$$I_{\text{eff}} \simeq \left[\sum p_S(V_{\text{hub}}) I_S^m\ (V_{\text{hub}})\right]^{1/m}$$

or the general sector s Is is the representative value of the turbulence intensity, accounting for wake induced turbulence by Ns neighboring $(di < 10)$ turbines in the sector s.

$$I_s = \sigma_{\text{eff},s}(\theta_s, V_{\text{hub}})/V_{\text{hub}}$$

$$\sigma_{\text{eff},s}(\theta_s, V_{\text{hub}}) = \left[(1 - N_s\, p_{ws})\, \sigma_{r,s}^m(\theta_s, V_{\text{hub}}) + p_{ws} \sum \sigma_{T,s}^m(\theta_s, V_{\text{hub}})\right]^{1/m}$$

N_s - neighboring turbines within sector s,

p_{ws} - probability to be under wake for a given sector,

$\sigma_{r,s}$- representative value of ambient turbulence intensity for sector s,

$\sigma_{T,s}$- representative value of turbulence intensity in wake condition.

The turbulence intensity under wake condition is modeled with the following expression

$$\sigma_{T,s}(d_i) = \sqrt{\frac{V_{\text{hub}^2}}{\left(1.5 + 0.8\, d_i/\sqrt{C_T}\right)^2} + \sigma_{r,s}^2(\theta_s, V_{\text{hub}})}$$

where

C_T - thrust coefficient of ii-th wind turbine generating the wake,

d_i - distance (in rotor diameters) to the i-th wind turbine.

Bin width

Width of the velocity bin centered at 15 m/s used to compute mean and standard deviation of turbulence intensity.

Gust factor

In the case the .tws file does not contain records averaged over 10 minutes there will be the need to adjust the Gust factor. In case of a .tws with stepping longer than 10 minutes, e.g. a hour, the Gust factor is the averaged ratio of the maximum 10 minutes peaks within the recording period and the mean wind speed in the same time; so a number bigger than unity. The Gust factor will multiply each yearly peak at the turbine location before fitting with a Gumbel distribution.

Wöhler coefficient

The Wöhler coefficient is the exponent m appearing in the definition of the effective turbulence Ieff. It is specific for the material used to build the component that will be verified to fatigue loads. In the case of a complex structure as a wind turbine it has be chosen the maximum coefficient between all components. A coefficient of 10, slightly conservative, is meant for the glass-fibers, generally present in the composite materials of the blades.

Ieff filter

A filter is set, given by the minimum number of samples needed to account for a sector in the Ieff calculation.

Wind speeds range Ieff

According to the IEC standards [3,4] the amended equation (35) in [4] has to be verified ranging from 60% of the rated wind speed (Vr) and the cut-out wind speed (Vout) of the turbine. If the properties of the turbine are unknown [4] recommends to verify in the range of wind speeds 0.2 Vref-0.4 Vref. When the Wind speeds range Ieff is set to its default value "According to IEC standards" the Ieff will be verified in the range 0.6 Vr-Vout; though, if turbine properties are unknown, the check will be performed between 7 and 25 m/s. Alternative to the default value is to set a user defined wind speed range.

WAT Export

Enabling this option allows to export a set of files to the report\WAT folder that can be later imported in Windfarm Assessment Tool (WAT), a free software by Risø and DTU Wind Energy. A basic WAT file (.txt) will be exported, together with turbulence files (.txt), power characteristics (.wtg) and a terrain file (.grd). Turbulence files are obtained by the transferred .tws at each hub position. Power characteristics in .wtg format are converted from the .pws files loaded in the **Objects** module while the digital terrain model (DTM) in .grd format is converted from the .gws file loaded in the current WindSim project.

Excel Export

The ambient and effective turbulence is plotted against the IEC curves for each turbine for each sector and wind speed bin.

Site Compliance Export

All information which is needed for a site compliance study is gathered in one file.

Matrix table Export

The Matrix provides sector-wise binned split of turbine energy production, energy including wake effect, turbulence and speed frequencies. The Matrixes enable to analyze special cases like directional curtailment, sector management systems and more. Tab separate format is easy to load in other software and postprocess.

AEP and IEC report Export

The AEP and IEC results are a collection of information, in a tab separated text format, related to Annual energy production and IEC turbine class suitability analysis, following IEC 61400-1 ed. 4 (2019) standards [7].

References
[1] Risø and DTU Wind Energy. Windfarm Assessment Tool (WAT), software official web site and installation package (MSI) download **http://www.wasp.dk/Products/WAT.aspx**
[2] International Electrotechnical Commission (IEC). IEC 61400-1 ed2.0 (1999), Wind turbine generator systems - Part 1: Design requirements
[3] International Electrotechnical Commission (IEC). IEC 61400-1 ed3.0 (2005), Wind turbine generator systems - Part 1: Design requirements
[4] International Electrotechnical Commission (IEC). IEC 61400-1-am1 ed3.0 (2010), Wind turbine generator systems - Part 1: Design requirements
[5] R.I.Harris. *"Gumbel re-visited - a new look at extreme value statistics applied to wind speeds."* Journal of Wind Engineering and Industrial Aerodynamics, Vol. 59, Issue 1, January 1996, Pages 1-22
[6] R.I.Harris. *"Improvements to the `Method of Independent Storms'."* Journal of Wind Engineering and Industrial Aerodynamics, Vol. 80, Issues 1–2, 1 March 1999, Pages 1-30
[7] International Electrotechnical Commission (IEC). IEC 61400-1 ed4.0 (2019), Wind turbine generator systems - Part 1: Design requirements

WindSim Accelerator

About WindSim Accelerator

WindSim Accelerator is the cloud-native complement to WindSim Desktop, offering on-demand, high-performance computing for micrositing workflows. By leveraging scalable cloud resources, Accelerator enables rapid execution of the Preprocessing and Wind Field modules within the browser, while integrating with WindSim Desktop for the remaining modules.

Purpose and Scope

The primary goal of WindSim Accelerator is to optimize the placement and performance of wind turbines over complex terrain (micrositing) through numerical wind-field simulations. Accelerator focuses on two core operations:

1. **Preprocessing**: Preparation and tiling of digital elevation and roughness data for simulation.
2. **Wind Field Calculation**: High-resolution CFD-based wind-flow modeling across the site.

Remaining steps in the design chain—including object placement, climatology coupling, and energy assessment—continue to be executed in WindSim Desktop, preserving the familiar modular workflow. The cloud architecture unlocks higher-fidelity simulations by distributing the RANS solver across multiple compute nodes, enabling:

- **Finer Mesh Resolutions**: More cells per cubic meter for detailed topographic features.
- **Advanced Solver Strategies**: Hybrid parallelization and dynamic load balancing for large-scale domains.

Modular Architecture

WindSim Accelerator mirrors the modular design of WindSim Desktop, currently comprising three modules:

1. **Map Module**: Interactive visualization and configuration of terrain elevation and roughness datasets.
2. **Model Module**: Set-up of computational meshes and solver parameters based on user-defined settings.
3. **Wind Fields Module**: Distributed execution of the RANS-based CFD solver for wind-flow prediction, with real-time progress tracking.

Continuous Evolution

WindSim Accelerator is under active development, with new features and optimizations released regularly. Upcoming enhancements include expanded postprocessing options, which finally will replace the need for WindSim desktop as a postprocessing tool.

Map

Area of Interest

The first step in setting up a flow field simulation is extracting topography and roughness of the target area. In *Accelerator*, this is done using the *Map module*, which utilizes the *World Elevation Terrain service* provided by *ESRI*. This dynamic terrain service supplies numerical values representing ground surface heights based on a *digital terrain model (DTM*) from multiple data sources. Heights are referenced to sea level (orthometric height = 0), and water bodies above sea level have approximated nominal water heights.

Specifying the coordinates

There are two methods to define the area of interest:

Using the Box Selector:

- Click on the desired location to set the *centre coordinates* of the simulation domain.
- The *box selector* will automatically generate a *rectangular refinement zone* with specified vertical and horizontal dimensions.

Using Latitude and Longitude Coordinates:

- Manually set *latitude and longitude markers* to define the points of interest.
- The system will generate a *rectangular refinement area* based on the minimum and maximum latitude/longitude values.

Refinement and Buffer Zones

The *refinement zone* represents the area where the simulation results are expected to be most accurate. When using *markers*, the area is extended 2 km beyond the markers (such as turbines or meteorological masts) to better capture ground effects.
When using the *box selector*, it is recommended to extend the refinement area to ensure high resolution near critical points, such as turbines.

The *buffer zone* reduces the impact of boundary conditions. At the *inlet*, where the wind enters, the simulation assumes a *logarithmic wind profile* based on surface roughness. As the wind flows over the buffer zone, it develops a *more realistic profile* before reaching the refinement area, reducing boundary effects. The buffer zone is by default set to 10000 m beyond the refinement area. It should be ensured that the edges of the buffer zone do not contain significant features (e.g., large cliffs), as they can create unwanted simulation effects that extend into the refinement area.

Checking Elevation Map Resolution

To check the resolution of the elevation map at a given location:

- Use the interactive map.
- Scroll to the area of interest.
- Right-click on the map to view the data source and resolution used at that site.

Setting Up Surface Roughness

Two *roughness datasets* are available , The Corine Land Cover 2018 (CLC2018) Roughness (Europe) and Sentinel-2 Roughness (Global). CLC2018 is one of the datasets produced within the frame of the Corine Land Cover programme referring to land cover / land use status of year 2018. Sentinel-2 is a 10-meter resolution land cover time series of the world, produced by Impact Observatory, Microsoft and Esri.

Steps to Configure Roughness:

1. Enable the roughness layer in the right-hand toolbar to visualize roughness categories.
2. Adjust roughness category values if needed.
3. Once the roughness setup is satisfactory, click *"Extract Terrain".*
4. After terrain extraction is complete, proceed to the *Model tab* to continue the simulation setup.

Model

Numerical 3D Model

After extracting terrain data using the *Map module*, you can proceed to create a discretized model of the simulation domain. Click on the *Model tab* on the left-hand side to get started.

Converting to Grid File

Once the elevation and roughness data are extracted:

1. The images are converted to the *grid.gws* file format.
2. The full 4000x4000 resolution is retained to avoid interpolation losses.
3. The *grid.gws file* is used to create a 3D computational grid following the terrain.

Settings

Several parameters can be adjusted when generating the 3D model:

Height Above Terrain:

- Defines the height (in meters) of the model above the highest terrain point.
- Higher terrain features require a greater model height.
- An automatic option is available for optimal height selection.

Resolution of Horizontal Cells

The horizontal resolution of our model is based on the maximum number of cells, or by setting the horizontal resolution.
Cells maintain constant size in the *refinement area*, with gradual size increase in the *buffer zone*.

Height Distribution Factor

- The model assumes *geometric vertical expansion* with this factor.

Number of Vertical Cells

- Specifies the number of cells along the z-axis.
- Higher resolution is required near the ground, while lower resolution is sufficient in upper layers.
- *Geometrical expansion* controls vertical cell sizes

For more detailed description of these features, see **WindSim 12 Desktop numerical model**.

Forest Modelling

Configuring Forest Representation

If a forest is present in your selected area, you can enable *forest modelling* to refine your simulation.

1. Enable the Forest Model:
   - Open the *Forest widget*.
   - Click *Add Forest*.
2. Set Forest Parameters:
   - *Roughness Height*: The roughness value representing the forest.
   - *Height*: The actual forest height.
   - *Resistive Force C2*: The resistive force proportional to velocity squared.
   - *Z direction cell count*: The number of vertical cells representing the forest.
   - *Turbulence Sources*: Defines whether the forest affects turbulence.
   - Enter forest values and verify roughness settings.
   - Click *Create 3D Model*.
   - Check the *Forest box* in the result pane to validate placement.

Vertical Grid and Forest Interaction

- The bottom layer of cells is controlled by the forest model.
- Above the forest, *geometrical expansion* determines cell distribution.
- The *grid (z) view* provides an exact distribution.

Multiple Forests

- If multiple forests exist, cell heights must align.
  - Example:
    - Forest 1: 10m height (2 cells of 5m each)
    - Forest 2: 20m height (4 cells of 5m each)
    - Both forests align at the bottom 2 cells.

A closer description of the forest model can be found in **WindSim 12 Desktop forest model**.

Investigating the Model

To verify the model before running the wind field simulations, we should investigate the different layers now created as part of the model, which include:

Terrain Layers

- *Elevation*

- *Roughness*
- *Roughness Logarithmic*
- *Inclination*
- *Second Order Derivative (Curve)*
- *Delta Elevation*

3D Model Layers

- *Grid (XY)*
- *Grid (Z)*
- *Open Area*

Modify settings if necessary and regenerate the 3D model before proceeding to the *Windfields tab*.

Preprocessing in WindSim desktop

Alternatively, a customer may want to use their own data for creating a grid file. The pre-processing then needs to take place in beforehand using WindSim Desktop (see **digital terrain conversion).** A pre-processed project, ready for simulations, can thereafter be uploaded to WindSim Accelerator. The workflow is outlined in the steps below.

Start WindSim desktop

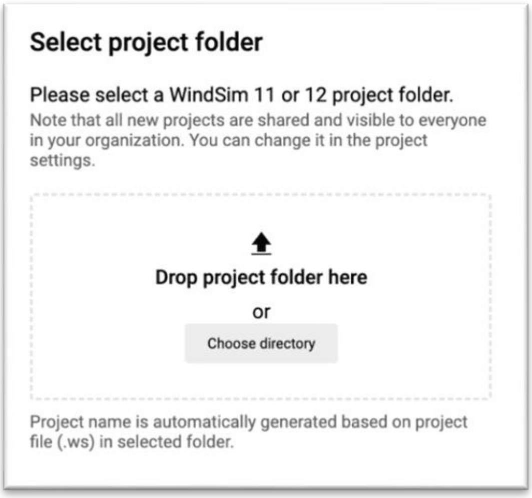
1. Create a new or open existing project in WindSim desktop.
2. Run the Terrain module to establish the computational model to generate the wind database successively.
3. When the Terrain module has been completed, go to the Wind Fields module to specify the Properties for wind fields calculations, or set them directly in the WindFields module in WindSim Accelerator.
4. Save the project, File -> Save.

Upload project in WindSim Accelerator

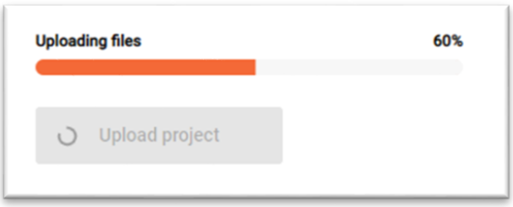
When you log in to WindSim Accelerator, you are directed to the Dashboard and the Project list. The steps below show how to upload the project you created in WindSim desktop.

1. Select *Upload project*

2. Choose the folder where your project and belonging files are located. The new project is visible and accessible to everyone in your organization. You can change it to private in project settings.



3. Upload status is displaying progress.



4. Inspect the wind fields input.

The input parameters can be modified if needed. You can correct potential mistakes or adjust them to changing needs. You can re-run the simulation on a single project multiple times or duplicate it on the project list and run it in parallel. If you want to run a simulation on an existing project (use the previous results), ensure the checkbox “Use the previous run as input” is selected.

Wind Fields

Prerequisites

An advanced project in Accelerator where the Map and Model modules has been successfully run, or an uploaded WindSim 12 project in which the Terrain is already completed.

Settings

- Boundary and initial conditions
  - *Do nesting*: Not implemented on Accelerator yet
  - *Sector input type*: Select which sector you want to run in the simulation
  - *Height of boundary layer*
  - *Speed above boundary layer*
  - *Use previous run as input*: Continues a previous run, this can be enabled when a run did not yet reach a satisfactorily convergence but looks promising.
  - *Boundary condition at top*
    - *Fixed pressure*: Suitable on complex terrain
    - *No-friction wall*: Suitable for flat terrain
    - *Diffusive link (moving fixed pressure):* Like fixed pressure case but imposing a constant speed at the top. The imposition of a constant speed enables us to lower the height of the model and use less cells *vertically*
- Physical models
  - *Turbulence model*:
    - *Standard k-epsilon*
    - *Modified*
    - *k-epsilon with YAP correction*
    - *RNG k-epsilon*
- Calculation parameters
  - Solvers:

        - *GCV (General Collocated-Velocity)*
        - *Parallel GCV (A parallelized version of the GCV)*
        - *Parallel GCV + AMG (Algebraic MultiGrid, uses the AMG to solve the pressure equation)*
    - Convergence settings:
        - Use *manual* definition for specifying
        - Exploratory for initial investigation of a site
        - *Accurate* for recommended convergence rate
- Convergence monitoring: Sets the point of the map where the spot value plots are located. The monitored spot should be set in an area where the conditions are of interest.
- Output
    - *Height of reduced wind database*: For further postprocessing of the data only the lower layer of the air column is of interest. Here you can specify the volume. Having a lower height will speed up postprocessing.
    - *XY reduced wind database*: Only the area inside the refinement area is of sufficient quality and of interest for processing, this option enables a smaller database of the windfields only encompassing that area.

For closer description of flow equations, settings and solution methods please refer to **WindSim 12 Desktop Wind Fields**.

Stability in Accelerator

Accelerator can run simulation with preserved temperature settings that you set in the desktop version. It is possible to do stability simulation on *Accelerator*. Unfortunately, the Web GUI does not show the temperature settings yet, but the settings you set in the Desktop will be preserved after you upload the project to *Accelerator*. In a future version we will show the temperature settings, so you can change them in the Web GUI as well. The stability option, only using *Monin-Obukhov length*, is now added to *Accelerator* and can be selected in *Wind Fields.*

Running Wind Fields

When selecting *Run Wind Fields*, the wind field simulations are submitted to the cloud, and simulations for each sector run independently on separate servers.
The user can follow the progress through the interface, and in the Job status menu, you can

- Follow progress of each sector job
- Refresh job status manually
- Stop all jobs
- Cancel all jobs
- Stop / cancel individual jobs, for each sector
- Check results for each sector after completion
    - *Field value*
    - *Spot value*
    - *Residual values*
    - *Residual values log*

Simulation time and credits approximation

The total wall-clock time for a simulation can be estimated using the relationship

$$T_{sim} = N_{iter} \times N_{cells\ (million)} \times F_{emp}$$

where N_{iter} is the number of solver iterations, N_{cells} the mesh size in millions of cells, and F_{emp} an empirical factor that captures the combined performance of the hardware, solver algorithm and parallel efficiency.

The number of iterations depends on the complexity of the site. For a medium-complexity case, a good rule of thumb is roughly 100 iterations per million cells ($N_{iter} \approx 100 \times N_{cells\ (million)}$). Highly complex sites may require significantly more iterations, and simpler problems may converge in fewer than 100 iterations per million cells. The chosen convergence criteria (residual tolerances) directly influence both iteration count and solution accuracy: tighter tolerances reduce numerical error but can multiply iteration requirements, whereas looser criteria speed up runs at the expense of potential under-resolution or instability.

The table below summarizes the approximate credit requirements for different project sizes and shows how many projects each Accelerator plan supports.

		Number of projects		
Model size (M cells)	**Credits (12 sectors)**	**Light Plan** 18 000 credits	**Plus Plan** 60 000 credits	**Premium Plan** 120 000 credits
5	300	60	200	400
10	1200	15	50	100
15	2700	7	22	44
20	4800	4	13	25

Postprocess results in WindSim desktop

- When the simulations are completed or stopped manually by the user, the results filed are stored in a project.zip file. The project.zip file can be downloaded by selecting *Download project*.
- Unzip project.zip to the project folder on your local machine/server.
- Run the script *RunReports.bat* script to generate results. The script is located in the downloaded project folder. The script will generate results for Spot values, Residual values and Residual values log and display these in WindSim desktop when you open the project file.
- As in WindSim desktop, you can restart a simulation from the end state if it does not converge. Be aware that the existing data will be overwritten unless you duplicate your project.

Best Practices and Recommended Settings

This section provides best practices and recommended settings for running wind simulations efficiently and accurately. These guidelines are based on experience and testing to help users configure simulations for optimal performance.

Default Settings: Are They Suitable for All Sites?

Default settings can work well for many sites, but they might not be the best fit for every location. Users should evaluate their site characteristics and make necessary adjustments using available configuration options such as:

- Forest Model: Used when the simulation includes forested areas.
- Top Boundary Conditions: Selection depends on terrain complexity.
- Turbulence Models: Different models are available depending on flow characteristics.
- Grid Size: Determined based on project size and computational resources.

When Should You Start Tweaking Models?

There are several indicators that a model may need adjustments:

- Non-Convergence: If the model does not converge, factors such as boundary conditions and locations, grid resolution, and terrain complexity may need revision.

- Mismatch in Vertical Profiles: If vertical profiles do not align with observations, consider adjusting atmospheric stability settings.
- Keep in mind that well behaving solutions can be physically wrong, due to choices or simplification of modelling. It is therefore essential to interpret the results in a physical context and validate them against observational data.

Top Boundary Conditions: Understanding and Selection

Selecting the appropriate top boundary condition for accurate simulations:

- Fixed Pressure: Recommended for complex terrain as it maintains realistic pressure gradients.
- No-Friction-Wall: Suitable for flat terrain, ensuring that momentum remains consistent.
- Diffusive Link (Moving Fixed Pressure): Used for simple terrain while maintaining a constant wind speed at the top boundary.

Turbulence Models: Selection Guidelines

Turbulence models impact the accuracy of flow predictions. The following options are available:

- Standard k-ε Model: Commonly used but not ideal for complex terrain due to excessive vertical mixing.
- Modified k-ε Model: Adjusted for atmospheric conditions, offering better performance in complex terrains.
- RNG k-ε Model: Best suited for cases with recirculation and complex flows, providing improved convergence.
- YAP Correction: Can improve performance in separation zones and is generally recommended.

When to Use Smoothing

Smoothing should generally be avoided unless necessary due to divergence in the Wind Fields module. Consider using smoothing if:

- Grid Skewing Causes High-Speed Anomalies: A smoothing factor around 0.1 should be the initial attempt.

- Persistent Divergence Issues: Gradually lower the second-order derivative threshold and use the "Smoothing Radius" to avoid excessive smoothing near wind farm locations.

Smoothing is currently only available in the desktop version.

Grid Size: Considerations and Recommendations

Grid resolution impacts computational efficiency and accuracy. Considerations include:

- Desktop Users: Hardware resources (CPU, memory) should be evaluated.
- Typical Resolution: 10 - 50 m resolution is standard.
- Complex Terrain: Higher resolution is preferred.
- Vertical Layers: 25-30 layers are sufficient for flat terrain but should be increased for finer horizontal resolutions.
- Bottom Layers: The lowest three layers should have similar thickness for numerical stability.

Solvers: Selection Based on Project Needs

Choosing the right solver depends on the project scale and computational resources:

- GCV Solver: Preferred for its strong convergence behaviour.
- Parallel GCV-AMG Solver: Best for large projects exceeding 4 million cells as it speeds up computations.

Stability: Key Considerations

Understanding atmospheric stability conditions is essential for accurate wind simulations:

- Observation Data Analysis: Determine whether stable conditions dominate the site.
- Monin-Obukhov Length (MOL): Should be greater than 50 for valid simulations.
- Negative MOL Values: Should be avoided as they lead to non-physical results.

Additional Considerations

Extensions of the Simulation Area

- The area should be large enough to include key terrain features influencing wind flow.
- Ideally, boarders should be placed along ridges rather than valleys to avoid artificial speed-up effects.

- If terrain cutting leads to unwanted effects, terrain smoothing can help minimize errors (only desktop).

Cell Aspect Ratio

- Divergence issues may arise if the cell length/cell height ratio exceeds 10.
- Keeping cells as close to a cubic shape as possible enhances stability.

Maximum Number of Cells (only desktop)

- 1.2 GB RAM is typically required per 1 million cells.
- Large simulations should ensure adequate virtual memory allocation to prevent slowdowns.

Vertical Gridding

- At least 6-8 cells should be placed in the first 100 m above ground for accurate flow representation.
- The first cell should be at least twice the roughness element height.

Refinement Area Size

- Keep rotor diameters at a reasonable distance from the refinement edges.
- Ensure grid stretching is not excessive near the boundaries.

Forest Modelling

- Used for forested areas larger than 1 km in diameter.
- Avoid forested areas at the inlet as they can create unphysical effects propagating in the domain.
- Must be configured based on the site’s forest characteristics.
- Ensure appropriate wind profiles within forested regions to avoid divergence.

Nesting vs. Refinement (only desktop)

- Refinement is preferred over nesting, which can restrict flow conditions.
- If nesting is necessary, the outer coarse model should have a resolution no coarser than 100-200 m.
- The fine model should maintain a resolution of 10-30 m within the refinement area.

Postprocessing Considerations

Wake Modeling

- The Jensen model performs best, especially with the sum of squares method for multiple wakes.

- For large turbine arrays, averaging results from different wake calculation methods can improve accuracy.

Cross-Checking and Quality Improvement

- Cross-checking should only be performed for time series covering the same period without long-term corrections.
- Large errors may indicate roughness inconsistencies, non-neutral atmospheric effects, or thermal/mountain wind systems.
- Compare LIDAR measurements with simulations to refine model accuracy.

Following these best practices ensures a more accurate and stable wind simulation process. Users are encouraged to experiment within these guidelines and reach out to WindSim if further assistance is needed.