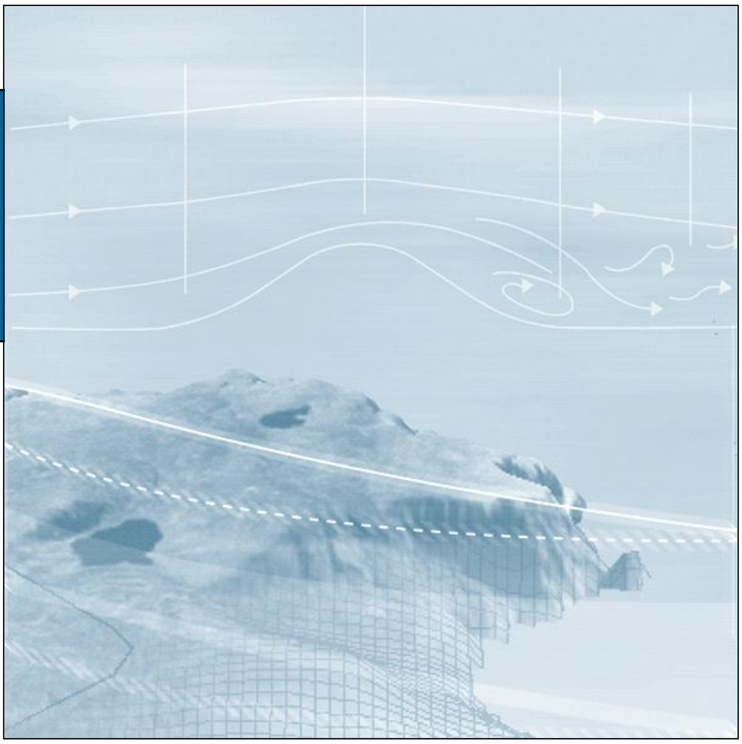
Tutorial project

WindSim 12

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



WindSim | Tutorial project

15th Edition | July 2025

WindSim 12

Contents

Tutorial Project: Hundhammer

In this tutorial project we look at the site of Hundhammerfjellet, or simply Hundhammer, located on the coast of Norway. The tutorial will guide you through all six modules of WindSim, supplying you with the required input.

NOTE: *All inputs have suggested default values, which for most cases would be the recommended value to use. Whenever a default input value has been modified it is highlighted with an orange overlay.*

Open WindSim and start a new project by choosing:

File > New > Project...

The New Project window is opened:

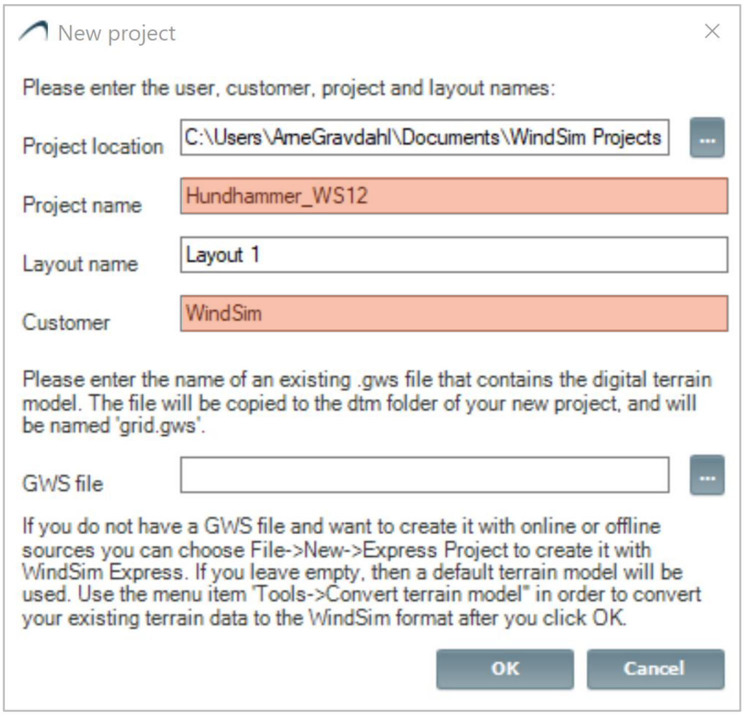


Figure 1 - New Project window.

Set your project location, which is the folder where your project will be stored, by browsing your hard drive, push the button  or use the default path, (C:\Users\User_name\Documents\WindSim Projects), give a project name, leave the Layout name as Layout 1, and finally enter the name of your customer. If no GWS file is specified, then the grid.gws with the terrain from Hundhammer will be automatically copied from the WindSim installation area. In this tutorial, this field is kept empty to load the default terrain data. Elevation and Roughness data are read from the .gws file and imported in the new WindSim project. Click OK in order to create the project.

Terrain

The first step within micrositing is the generation of a 3D model in the *Terrain* module. This involves choosing the horizontal and vertical extension of the volume to simulate. This volume, which we will call the computational domain, is then discretized into a system of hexahedral cells called a grid or mesh. The computational domain is built based on the digital terrain in *.gws format containing information about elevation and roughness.

WindSim can be run either by using the default settings in *Properties* or by specifying your own settings.

NOTE: *If you specify a non-default property value, then it is shown in **bold** type. There are a few exceptions to this rule. Properties associated with terrain extensions are always shown in **bold** type as they are project specific, a given default value equal for all projects can't be set.*

In WindSim Evaluation (EV), the maximum number of cells in the vertical direction has been set to 10, while the total number of cells in all three directions has been limited to maximum 50 000. In the commercial version of WindSim, the maximum number of cells would typically be limited by the available computer resources. Models with millions of cells can be simulated on PCs with a 64-bit operating system. The restrictive limitation in WindSim EV on the number of cells used for the computational model means that the results will not have the accuracy required for a proper micrositing thus WindSim EV it is not intended for commercial work.

Run the *Terrain module* by clicking on the **Start** button with the below default settings:

Properties

1: Terrain extension	
Coordinate system	Global
X-range	**318000; 332975**
Y-range	**7180000; 7194975**
Projection	**_UTM _WGS_84 33**
2: Roughness	
Roughness height	Read from grid.gws
3: Numerical model	
Automatic gridding	False
Vertical expansion	Geometrical
Refinement type	Refinement area
Refinement area, X-range	**322991; 327984**
Refinement area, Y-range	**7184991; 7189984**
Height above terrain	Automatic
Horizontal gridding	Maximum number of cells
Maximum number of cells	100000
Ratio additive length to resolution	0.5
Height distribution factor	0.3
Number of cells in Z direction	20
Set low-level uniform cells	True
Uniform vertical cells height	120
Uniform vertical cells number	4
4: Smoothing	
Smoothing type	No smoothing
5: Forest	
Forest	Disregard forest

Figure 1 - Property settings for the Terrain module with default values for the tutorial project Hundhammer.

Upon successful completion, the Module Selector of the *Terrain module* will have a green check and the *Description* pages will automatically be switched to the *Report* pages. Some of the content in the report is presented below.

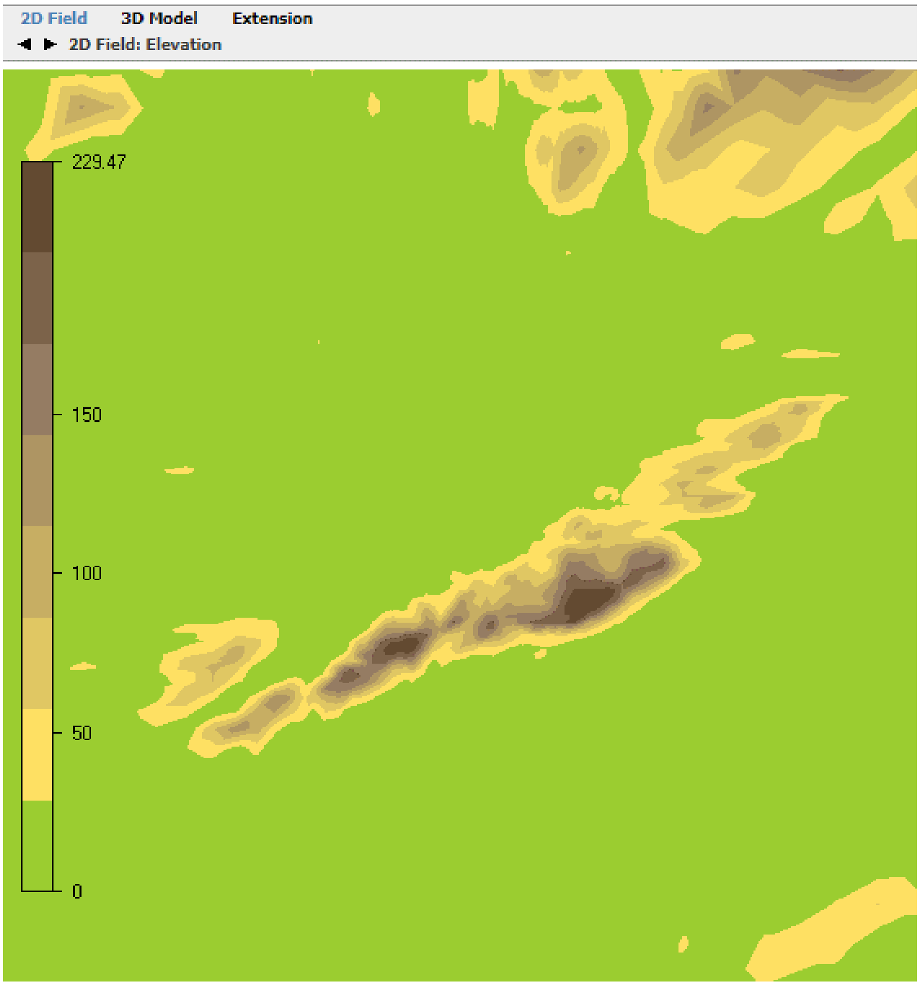


Figure 3 - Plot from Terrain module report; 2D Field: Elevation.

Click on the black arrows above the plots in the *Report* pages to visualize various 2D field data:

- Elevation
- Roughness height (z_0) with both linear and logarithmic scale
- Inclination angle (deg)
- Second order derivative of the elevation
- Delta elevation.

The delta elevation variable gets a non-zero value only when the smoothing option is applied to the model.

Click on **3D Model** and the black arrows to visualize:

- Grid (xy)
- Grid (z)
- Open area

Grid (xy) and Grid (z) contain information about the grid spacing and number of cells used for the discretization in horizontal and vertical directions.

The Open area data is a useful tool to understand if too much blockage has been introduced in the generation of the 3D model. A Wind Field simulation can be viewed as a numerical wind tunnel test; too much blockage would produce unphysical and therefore unacceptable speed-ups. The ratio between minimum and maximum open area is used in the automatic generation of the proper height of the 3D model, see the *Description* pages for details.

When you click on **3D** at the upper right of each plot; a 3D model in the visualization tool GLview is opened. The 3D visualization option is available for all the sections 2D Fields, 3D Model, and Extension. The 3D visualization option is also available in the other modules of WindSim.

There is a separate section in this document that further explores GLview. For now, we only explain the basic operations of movement:

- Translation with left mouse button
- Rotation with right mouse button
- Zoom with mouse wheel or both mouse buttons simultaneously

Click on **Extension** to visualize the horizontal extension of the generated 3D model which is marked with a grey frame over an elevation contour map:

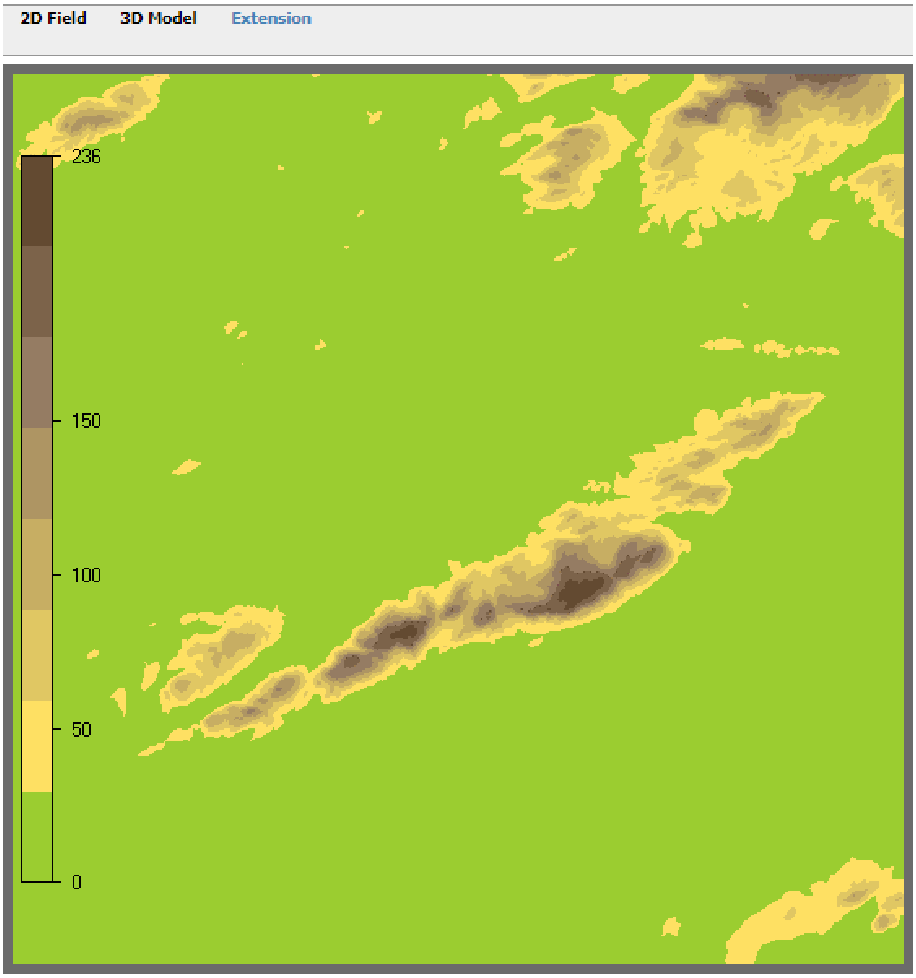


Figure 4 - Plot from Terrain module report; Extension

The contour map for the Extension is based on the original resolution of the .gws file, thus on the maximum possible resolution and not on the 3D model which is coarser than the original digital terrain data. The difference in resolution is particularly clear when comparing the two plots under *2D Field: Elevation* and *Extension.*

NOTE: *If you want to reduce the extension of the model, that is resetting the X- and Y-range of the properties given in Figure 2, then it is convenient to use the visualization tool GLview. You open GLview by clicking "3D" while Extension is displayed in the report, see figure 5.*

Next you should identify the coordinates of the lower left and upper right corner of the new reduced model you will establish. In GLview coordinates can be displayed for any cell in the grid by selecting the cell with the cursor while pressing down the "Ctrl" button. Note that when then "Ctrl" button is held down the cursor changes from and arrow to a plus sign as an indication that the selection mode is active. A cell is selected by pressing the left mouse button. Pressing the right mouse button allows for selection of properties, chose "Display

Polygon info…" and the output would be similar to Figure 5. In Figure 5 a cell with coordinates (5500,3000) have been selected. Hence adding 5500 meters to the X-min value of the X-range and 3000 meters to the Y-min of the Y-range will establish a model with the selected point as the lower left corner.

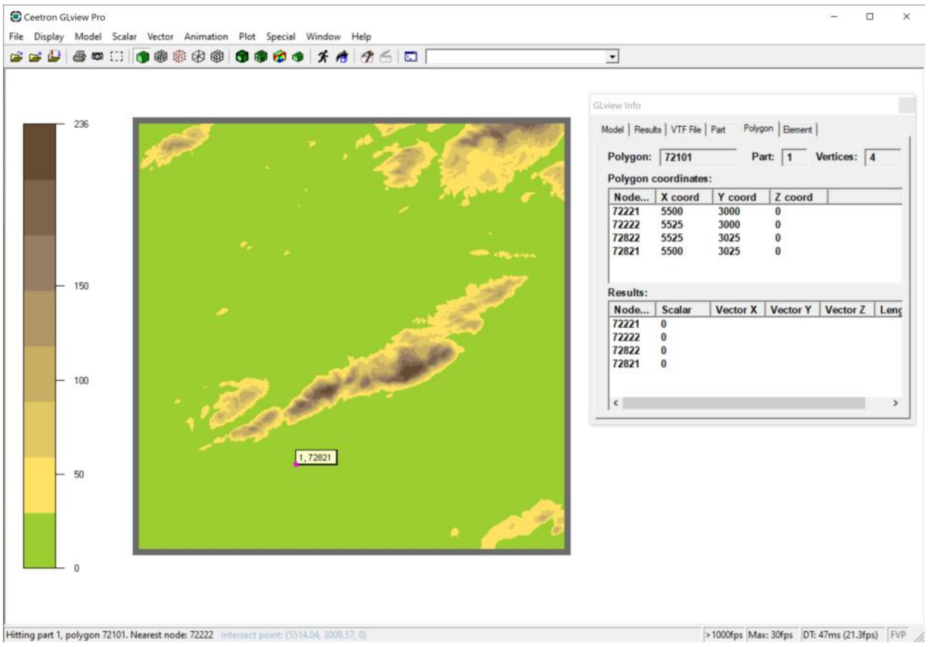


Figure 5 – Displaying cell (polygon) properties in GLview

The same procedure can be used to select a new upper right corner of your model. For more information about the coordinate systems used in WindSim, see the description pages for the Terrain module.

In order to produce a higher resolution grid, which means smaller discretization errors in the area of interest where the wind farm has to be constructed, a new extraction of a smaller area will be done.

Assign a new extension to the terrain model in the Properties panel by new X-range and Y-range. Define a refinement area to design an even denser grid in the area of interest, where the wind farm has to be located:

Properties

1: Terrain extension	
Coordinate system	Global
X-range	322950; 332000
Y-range	7182475; 7190025
Projection	_UTM _WGS_84 33
2: Roughness	
Roughness height	Read from grid.gws
3: Numerical model	
Automatic gridding	False
Vertical expansion	Geometrical
Refinement type	Refinement area
Refinement area, X-range	325966; 328984
Refinement area, Y-range	7184991; 7187509
Height above terrain	Automatic
Horizontal gridding	Maximum number of cells
Maximum number of cells	100000
Ratio additive length to resolution	0.5
Height distribution factor	0.3
Number of cells in Z direction	20
Set low-level uniform cells	True
Uniform vertical cells height	120
Uniform vertical cells number	4
4: Smoothing	
Smoothing type	No smoothing
5: Forest	
Forest	Disregard forest

Figure 6 - Property settings for the Terrain module, reduced area with refinement.

In this way you have cropped from the original .gws file an area only covering the ridge of Hundhammer, where the wind farm will be located. The borders of the computational domain are still quite far from the area of interest in order to avoid too heavy boundary effects.

NOTE: *The X- and Y-range of the refinement area is by default set to be in the centre of the model. Whenever a resetting of the X- and Y-range of the total modelled area is done, also a resetting of the refinement area should be done to keep the refinement area in the centre. A simple way to achieve this is to set the property Refinement area to "No refinement" and then reselect it to "Refinement area". At this stage the X- and Y-range of the Refinement area will be based on the newly set X- and Y-range of the total model and you will automatically obtain the values given in Figure 6.*

Click "Start" on the right-hand side of your screen to run the Terrain module again.

Click on **Terrain > Report > Extension** to get the Figure 7 and Figure 8 in the Report frame, which shows the extension of the new 3D model compared to the original gws terrain data.

NOTE: *When refinement is applied, the horizontal resolution varies. Its minimum and maximum values are available in the* ***Report > 3D Model*** *and* ***Grid (xy)****.*

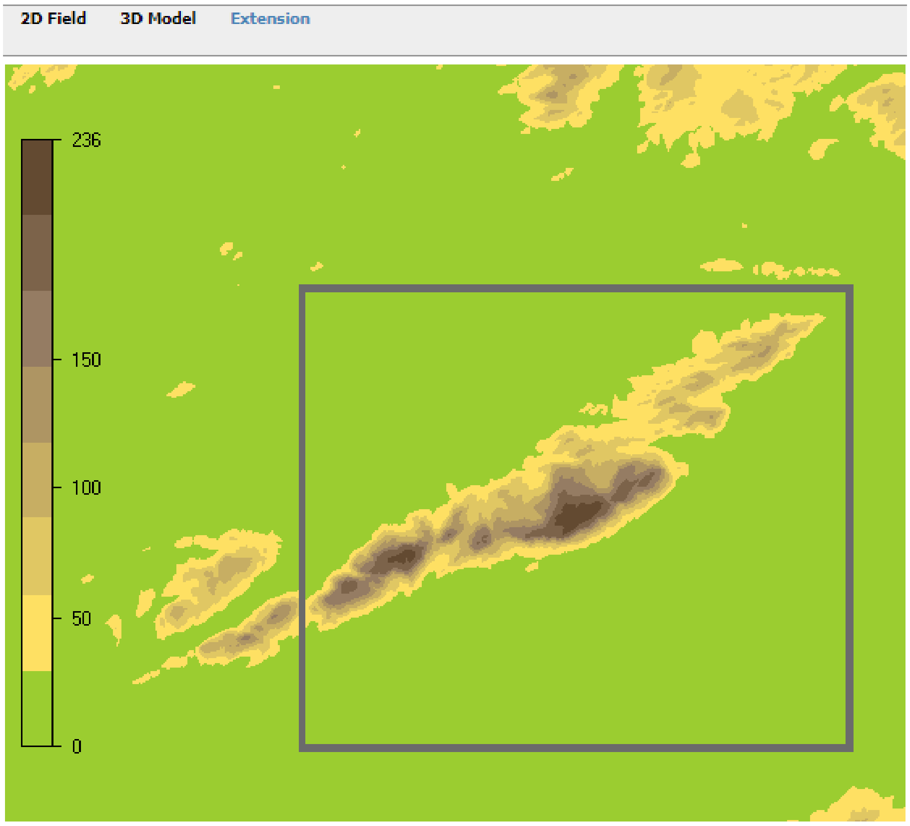


Fig 1. Digital terrain model - Extension.

The digital terrain model, marked as a box, is extracted from grid.gws.

x-min	x-max	y-min	y-max	x-extent	y-extent	resolution
322950.0	332000.0	7182475.0	7190025.0	9050.0	7550.0	Variable

Table 1. Digital terrain model.

x-min	x-max	y-min	y-max	x-extent	y-extent	resolution
318000.0	332975.0	7180000.0	7194975.0	14975.0	14975.0	25.0

Table 2. Data in grid.gws

Figure 7 - The digital terrain model, marked as a box, with given extensions is extracted from grid.gws.

Click on **Terrain > Report > 3D Model,** and then select **Grid (xy)** using the black arrows to obtain the report on the horizontal discretization of the generated 3D Model shown in Figure 8.

2D Field 3D Model **Extension**

◄ ► **3D Model: Grid (xy)**

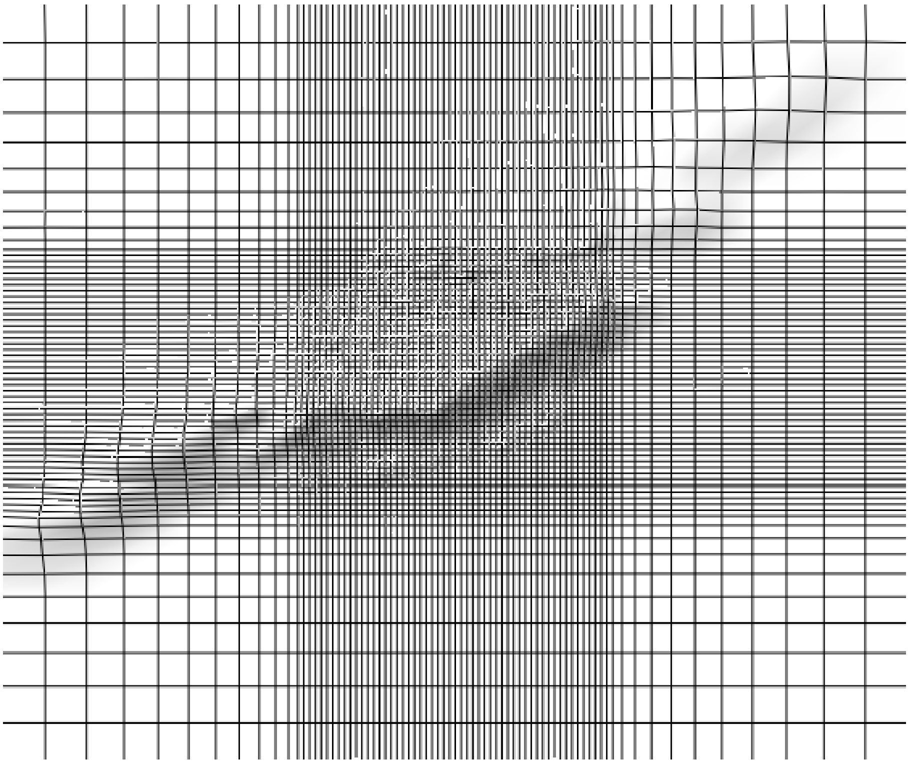


Fig 1. Digital terrain model - Grid (xy).

Body fitted co-ordinates (BFC) are used in grid generation. The above plot displays the resolution at ground level.

	x	y	z	total
Grid spacing, min - max (m)	58.0 - 444.6	58.6 - 398.9	Variable	-
Number of cells	76	65	20	98800

Table 1. Grid data.

Figure 8 - Terrain > Report > 3D Model: Grid (xy)

Click on **Terrain > Report > 3D Model** and then select **Grid (z)** using the black arrows to obtain the report on the vertical discretization of the generated 3D Model, shown in Figure 9.

2D Field 3D Model Extension

◀ ▶ 3D Model: Grid (z)

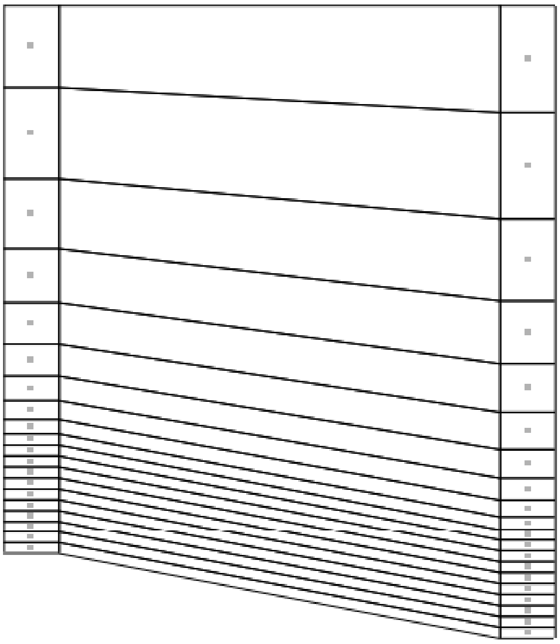


Fig 1. Digital terrain model - Grid (z).

The grid extends 1497.4 (m) above the point in the terrain with the highest elevation. The grid is refined towards the ground. The left and right columns display a schematic view of the distribution at the position with maximum and minimum elevation respectively. The nodes, where results from the simulations are available, are situated in the cell centers indicated by dots.

.	1	2	3	4	5	6	7	8	9	10
z-dist. max (m)	15.0	45.0	75.0	105.0	135.0	165.0	195.0	225.0	255.0	285.0
z-dist. min (m)	15.0	45.0	75.0	105.0	135.0	165.0	195.0	225.0	255.0	285.0

.	11	12	13	14	15	16	17	18	19	20
z-dist. max (m)	315.2	350.3	395.9	455.1	532.2	632.3	762.5	931.7	1151.8	1388.1
z-dist. min (m)	317.8	358.8	412.1	481.3	571.3	688.3	840.5	1038.2	1295.3	1586.7

Table 1. Distribution of the first 20 nodes in z-direction, relative to the ground, at the position with maximum and minimum elevation.

Figure 9 - Terrain > Report > 3D Model: Grid (z).

Click on **Terrain > Report > 3D Model.** Then select **Open area** by using the black arrows to obtain what is shown in Figure 10. The criteria named ***Open area*** account for the ratio of minimum and maximum area of the intersections of the computational domain with vertical planes south-north and west-east directed. The wind field simulations can be seen as numerical wind tunnel “runs”. So, it is important to reduce the blockage effect which produces unphysical speed-up.

NOTE: *The ideal height of the 3D model is computed automatically in order to keep this ratio above an acceptable value, see Description for details.*

2D Field 3D Model Extension

◄ ► 3D Model: Open area

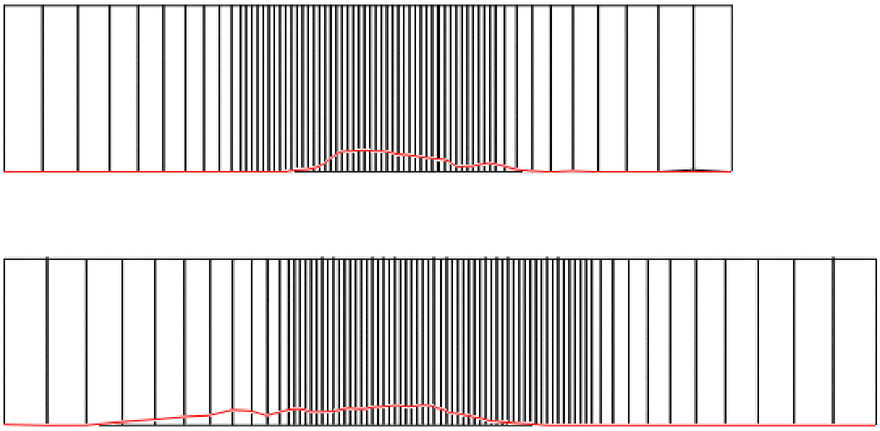


Fig 1. Digital terrain model - Open area.

The open area between the ground and upper boundary is calculated as the model is traversed in west-east and south-north direction. The maximum area is displayed as black rectangles while a red profile displays the ground level of the minimum area. The upper plot is for the traverse in west-east direction and the lower plot for the traverse in south-north direction. If the fraction between the minimum and maximum open area becomes too small, blocking effects might lead to unphysical speed-ups.

	Min (m^2)	Max (m^2)	Min/Max
Open area, west-east traverse	12887583	13033187	0.9888
Open area, south-north traverse	15127418	15656499	0.9662

Table 1. Open area data

Figure 10 - Terrain > Report > 3D Model: Open area.

Wind Fields

Once the generation of the 3D model has been completed in the *Terrain* module, the CFD simulations of the wind fields can start. The wind fields are determined by solving the Reynolds Averaged Navier-Stokes equations (RANS). The standard k-ε model is one option for turbulence closure. The RANS equations are discretized and integrated with a finite-volume method. Starting with the initial conditions, which are guessed estimates, the solution is progressively resolved by iteration until a converged solution is achieved.

There are four possible ways to solve the RANS equations in WindSim:

- GCV; a General Collocated Velocity method. Very robust. Always delivers a converged solution.
- Parallel GCV; a parallel solver of GCV splitting a numerical model in sub-models and use separate CPUs on each sub-model.
- GCV+HYPRE; HYPRE is a library of high-performance preconditioners and solvers, with BoomerAMG. This uses algebraic multigrid (AMG) algorithm for the linear equation solver for the GCV algorithm. The new solver setup shows a better convergence behaviour for big models run with parallel option.

- Parallel GCV+HYPRE; same as above including parallelisation, that is splitting a numerical model in sub-models and use separate CPUs on each sub-model.

For more information concerning the solver you can look at the “Description” in *Wind Fields* module.

Select the *Wind Fields* module and run the module with default settings. A window will pop up to let you follow the development of the wind field during the iterative solution procedure The *Field value* monitored by default is the *Speed scalar XYZ* at ground level, that is at the first Z level, which is the 2D field displayed in Figure 11. If several sectors are solved simultaneously on different processors, then one window will open for each sector being solved.

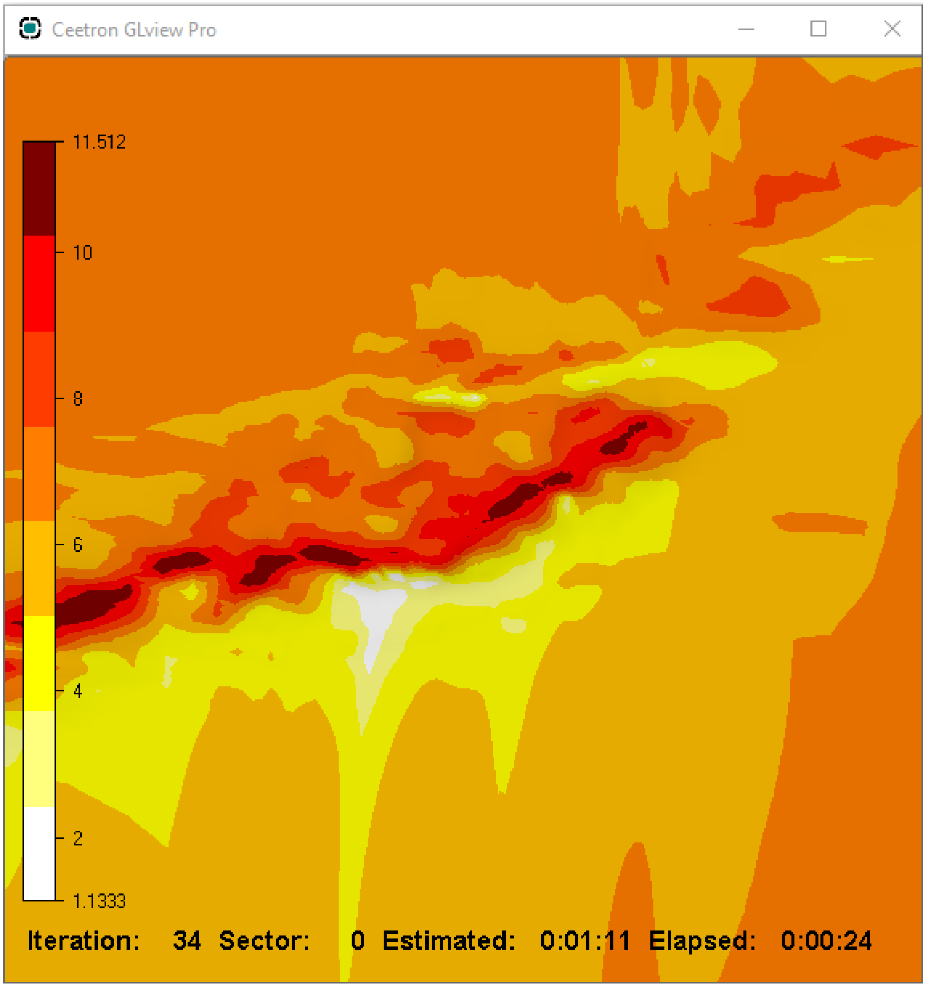


Figure 11 - One window will pop up for each simultaneous sector being run, by default the Field value monitored is the Speed scalar XYZ.

By default, the software monitors the magnitude of the velocity vector at the ground level (at the centre of the ground adjacent cells). If the simulations have reached a converged solution, you should not be able to see any further change in this plot.

In the *Report* pages you can check whether the modelling has been carried out correctly, by inspecting the graphs of the Spot and Residual values for all the solved variables. This is shown in Figure 12 and Figure 13.

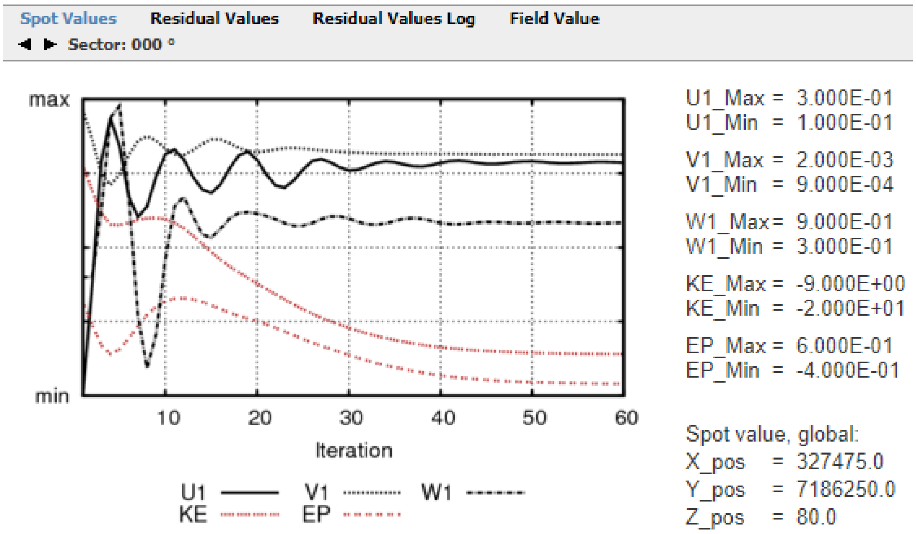


Figure 12 - Wind Fields > Report > Spot Values.

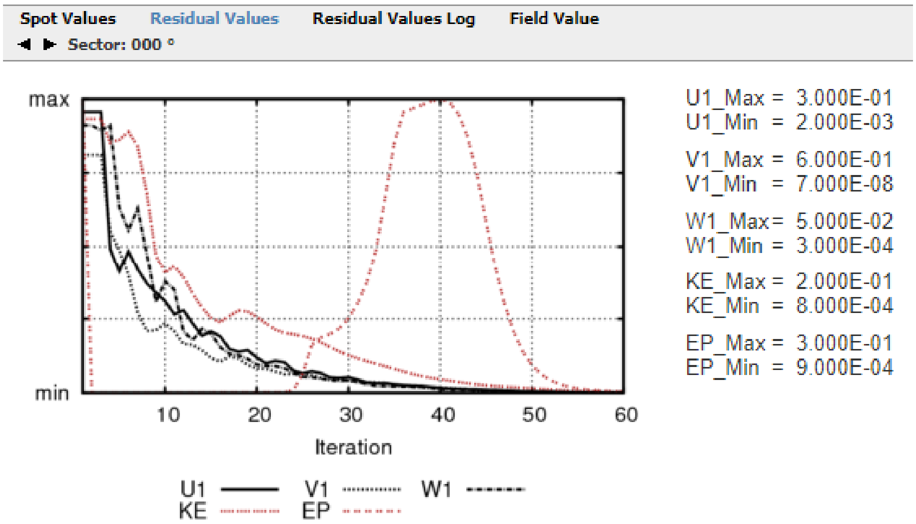


Figure 13 - Wind Fields > Report > Residual Values.

The report pages for the Spot and Residual values are completed with three tables regarding setting and status of the performed calculations.

In the Table 1 on the following page there is a summary of Boundary and Initial Conditions adopted for the simulations; Table 2 reports the Calculation parameters and the current status of the simulations while the Table 3 provides further information about Physical models and output for the wind field simulations. The parameters summarized in these tables are all explained in the description pages of the *Wind Fields* module.

Date	Time	Sector	Nesting	Height_BL	Speed_BL	Top_BC	Restart
27.12.22	13:01:46	000	No	500.0	10.0	fix pres.	No
27.12.22	13:03:18	030	No	500.0	10.0	fix pres.	No
27.12.22	13:04:51	060	No	500.0	10.0	fix pres.	No
27.12.22	13:06:27	090	No	500.0	10.0	fix pres.	No
27.12.22	13:08:07	120	No	500.0	10.0	fix pres.	No
27.12.22	13:09:44	150	No	500.0	10.0	fix pres.	No
27.12.22	13:11:21	180	No	500.0	10.0	fix pres.	No
27.12.22	13:12:59	210	No	500.0	10.0	fix pres.	No
27.12.22	13:14:39	240	No	500.0	10.0	fix pres.	No
27.12.22	13:16:23	270	No	500.0	10.0	fix pres.	No
27.12.22	13:18:11	300	No	500.0	10.0	fix pres.	No
27.12.22	13:19:54	330	No	500.0	10.0	fix pres.	No

Table 1. Boundary and initial conditions for the wind field simulations.

Date	Time	Sector	Solver	Conwiz	#Iter	#Iter exe	Time exe	(S)
27.12.22	13:01:46	000	GCV	No	100	60	00:01:13	(C)
27.12.22	13:03:18	030	GCV	No	100	60	00:01:11	(C)
27.12.22	13:04:51	060	GCV	No	100	60	00:01:12	(C)
27.12.22	13:06:27	090	GCV	No	100	59	00:01:15	(C)
27.12.22	13:08:07	120	GCV	No	100	65	00:01:19	(C)
27.12.22	13:09:44	150	GCV	No	100	58	00:01:14	(C)
27.12.22	13:11:21	180	GCV	No	100	58	00:01:14	(C)
27.12.22	13:12:59	210	GCV	No	100	59	00:01:14	(C)
27.12.22	13:14:39	240	GCV	No	100	61	00:01:16	(C)
27.12.22	13:16:23	270	GCV	No	100	61	00:01:19	(C)
27.12.22	13:18:11	300	GCV	No	100	65	00:01:22	(C)
27.12.22	13:19:54	330	GCV	No	100	58	00:01:17	(C)

Table 2. Calculation parameters and progress for the wind field simulations.

Date	Time	Sector	Turb mod	Coriolis	Latitude	Stability	Reduced_z
27.12.22	13:01:46	000	Standard	No	-	No	300
27.12.22	13:03:18	030	Standard	No	-	No	300
27.12.22	13:04:51	060	Standard	No	-	No	300
27.12.22	13:06:27	090	Standard	No	-	No	300
27.12.22	13:08:07	120	Standard	No	-	No	300
27.12.22	13:09:44	150	Standard	No	-	No	300
27.12.22	13:11:21	180	Standard	No	-	No	300
27.12.22	13:12:59	210	Standard	No	-	No	300
27.12.22	13:14:39	240	Standard	No	-	No	300
27.12.22	13:16:23	270	Standard	No	-	No	300
27.12.22	13:18:11	300	Standard	No	-	No	300
27.12.22	13:19:54	330	Standard	No	-	No	300

Table 3. Physical models and output for the wind field simulations.

Figure 15 - Wind Fields report

In the ***Wind Fields > Report > Field Value menu***, it is also possible to see an animation of the development of the chosen variable during the iterative process. In well converged simulations, the calculated variables should not change any further as the iterative process reaches the end. A screenshot of the monitored variable is shown in Figure 16.

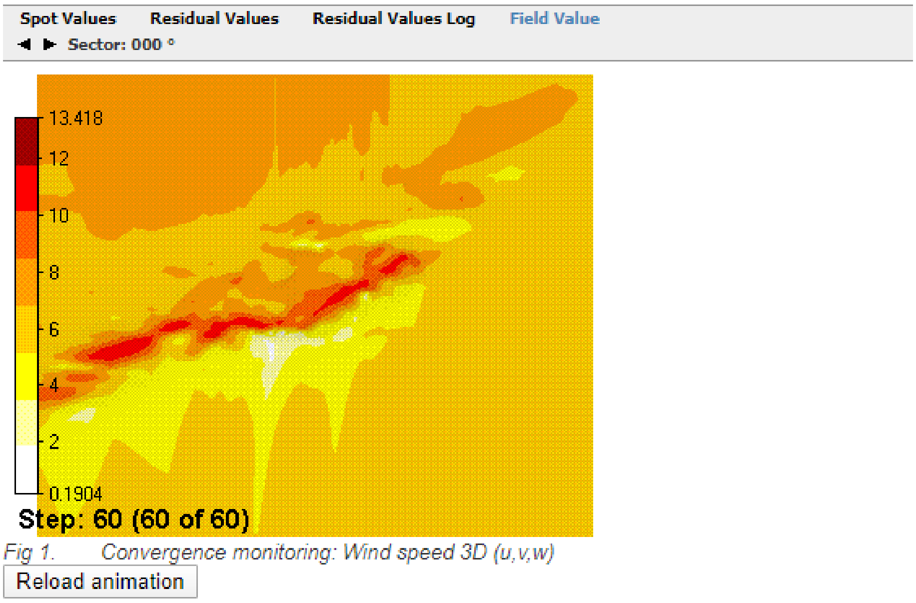


Figure 16 - Wind Fields > Report > Field Value.

Objects

The ***Objects module*** is used to:

- Place turbines in the wind farm
- Process climatologies
- Place transferred climatologies

Geometrical objects can also be placed within the 3D terrain model for visualization purposes. At the actual Hundhammer site there are 17 turbines and 2 climatologies. This layout is already established. It can be read by using the command:

Tools > Import objects **(.ows)...**

Load the file:

C:\Program Files\WindSim\WindSim 12.0.0\Data\Objects\Hundhammer.ows

Note that the name "Program Files" might vary according to which operating system you run on your computer

17 wind turbines will be added to the park layout. The following window message will pop-up if the import is successful:

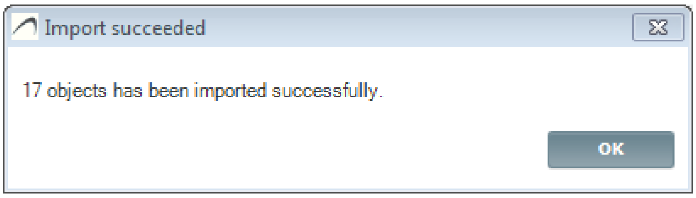


Figure 17 - Successful import of objects from an .ows file.

An alternative to the procedure described is to introduce new objects interactively in the Park Layout of the *Objects module* using the Toolbox on the right-hand side:

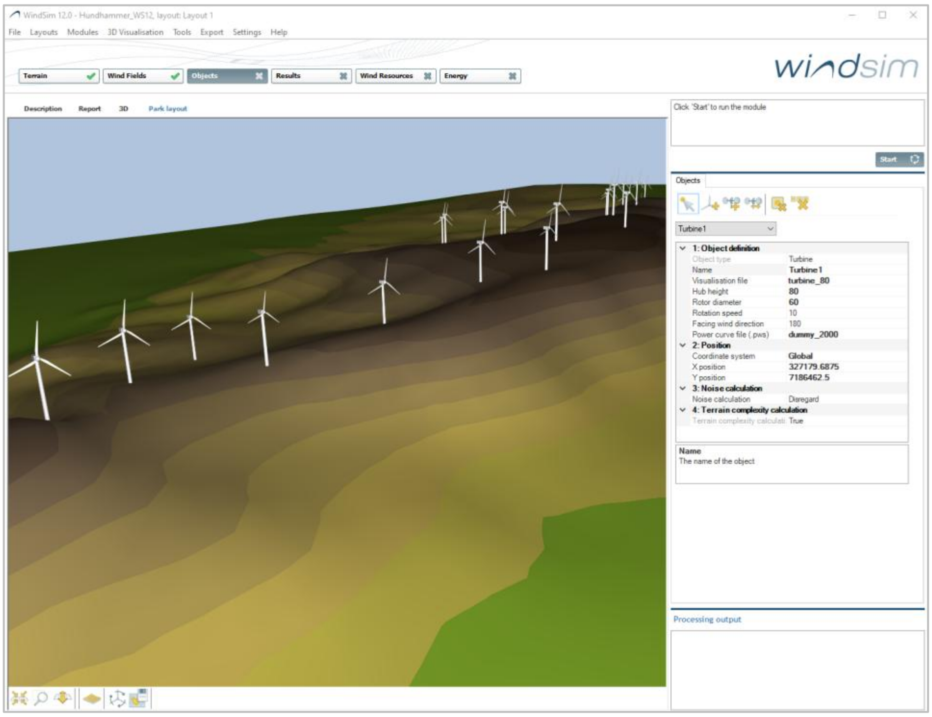


Figure 18 - Toolbox for interactive manipulation of objects.

with the following tools:

 Select mode

 Add turbine mode

 Add climatology from file

 Add transferred climatology mode

 Remove selected object

 Remove all

With the Park Layout it is possible to add, drag and drop wind turbines in a graphical window. This helps the user to design the park layout and make modifications to it. After adding a turbine, it could be moved interactively. Change the position of an object by first activating the "*Select mode*" in the Toolbox. Next, select an object by the clicking the left mouse button, the objects are moved with the mouse while pressing the Shift button.

The user still needs to load the climatology files (.wws or .tws), before running the module. Load a climatology using the "*Add climatology from file*" from the Toolbox. Two climatologies have been prepared for the site at Hundhammer namely: **0150-Tommerhol-30m_1year.wws** and **0801-Hundhammer-30m_1year.wws** located under the folder: **C:\Program Files\WindSim\WindSim 12.0.0\Data\Objects\Climatology.** Both sets of data are for anemometers at 30 meters height and referred to the same measuring period of 1 year. By using the "*Add climatology from file*" these climatologies are copied into the project folder structure.

Remark that climatology objects can't be moved, their position is given in the .wws or .tws files. The only way to change the location for a climatology object is to change the coordinates in the files.

The Objects module is now ready to run, click on the **Start** button. The layout of the wind farm can be visualized by selecting the **Report > Object representation** menu. The report for the Objects module will appear as in Figure 19 and Figure 20. The main features of the climatologies and turbines within the wind farm are listed in the tables 1 and 2 of the report.

Further information on climatologies and turbines are given if the user clicks on the links (blue bold type) in the tables. An animation is available which is activated by **Report > Animation** and **3D** to open **GLview Pro**. An animation is also available directly in the Park layout.

Object representation Animation
Texture Name 3D objects

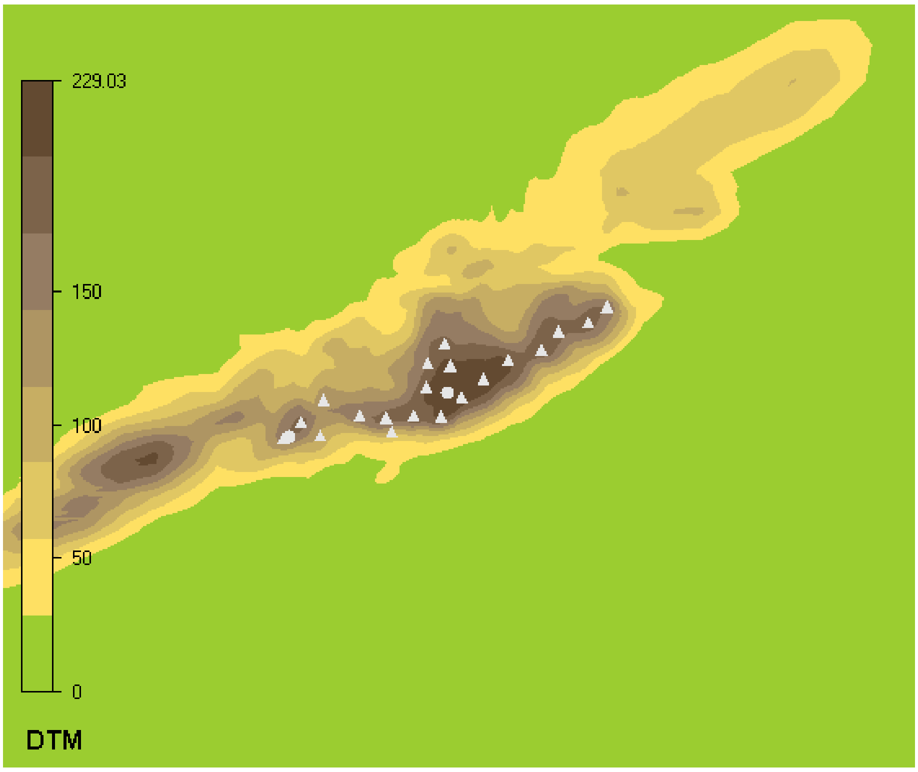


Fig 1. Digital terrain model with objects ▲ *Wind turbine* ● *Climatology station*

Figure 19 - Objects > Report > Object representation.

name	x	y	z	z (agl)	Reliability	climatology file
0150-Tommerhol..	325917.0	7185790.0	169.2	30.0	1.0	**0150-Tommerhol..**
0801-Hundhamme..	327362.0	7186191.0	222.9	30.0	1.0	**0801-Hundhamme..**

Table 1. Climatology objects.

The wind turbine characteristics combined with the calculated speed-ups constitute the basis for the energy calculations.

name	x	y	z	hub height	turbine type
Turbine1	327179.7	7186462.5	172.9	80.0	**windsim ws2000**
Turbine2	326850.1	7185836.5	114.0	80.0	**windsim ws2000**
Turbine3	326194.0	7185800.5	107.2	80.0	**windsim ws2000**
wecs1	325869.0	7185782.0	166.8	80.0	**Vestas V90 mod..**
wecs2	326031.0	7185924.0	174.0	80.0	**Vestas V90 mod..**
wecs3	326230.0	7186122.0	137.5	80.0	**Vestas V90 mod..**
wecs4	326564.0	7185983.0	167.5	80.0	**Vestas V90 mod..**
wecs5	326803.0	7185965.0	183.9	80.0	**Vestas V90 mod..**
wecs6	327057.0	7185983.0	197.8	80.0	**Vestas V90 mod..**
wecs7	327305.0	7185982.0	205.5	80.0	**Vestas V90 mod..**
wecs8	327505.0	7186159.0	221.7	80.0	**Vestas V90 mod..**
wecs9	327705.0	7186322.0	227.5	80.0	**Vestas V90 mod..**
wecs10	327932.0	7186491.0	202.7	80.0	**Vestas V90 mod..**
wecs11	328232.0	7186583.0	192.4	80.0	**Vestas V90 mod..**
wecs12	328387.0	7186747.0	183.7	80.0	**Vestas V90 mod..**
wecs13	328659.0	7186834.0	185.8	80.0	**Vestas V90 mod..**
wecs14	328833.0	7186977.0	167.0	80.0	**Vestas V90 mod..**
wecs15	327171.0	7186249.0	201.5	80.0	**Vestas V90 mod..**
wecs16	327391.0	7186440.0	210.6	80.0	**Vestas V90 mod..**
wecs17	327336.0	7186641.0	180.4	80.0	**Vestas V90 mod..**

Table 2. Turbine objects.

Figure 20 - Objects > Report > Object representation continued.

A photo or texture can be pasted on the terrain to provide a more realistic view of the wind farm area. Load the file **hundhammerfjellet.bmp** by opening ***Layouts > Terrain texture file > Open ….*** The file is located under the folder***:*** **C:\Program Files\WindSim\WindSim 12.0.0\Data\Texture**. Run the object module again in order to apply the texture. Activation of Texture in the Objects module report gives a plot with the photo draped over the terrain as seen in Figure 21.

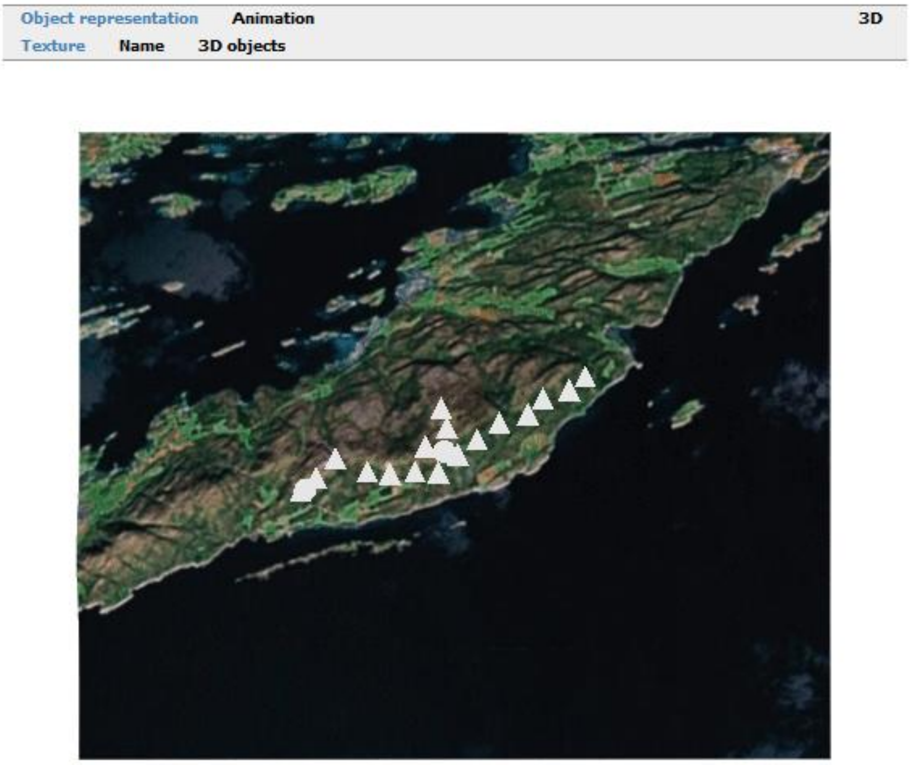


Figure 21 - Objects > Report > Object representation with Texture.

Click on **3D objects** and open this graphic in GLview by clicking the **3D** button. This allows various perspective views of the wind farm. At this stage fly-troughs could be generated, check out the 3D Samples at the WindSim web site.

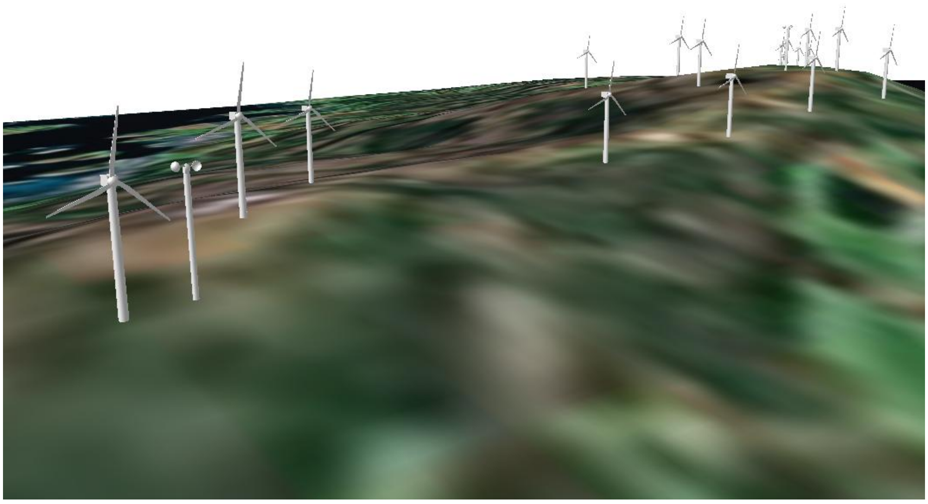


Figure 22 - Perspective view of the Hundhammer wind farm.

Noise calculations

Noise calculations are performed in the *Objects* module. First, select a wind turbine or a climatology and then select Noise **calculation>Based on broadband** at the “Objects frame”, as shown in Figure 23, in order to carry out a noise analysis, which can be output at any height, for any wind direction and for any wind speed. The default settings produce a noise map in 2 meters' height, for a northern wind of 5 m/s. The wind direction and wind speed are by default defined to be the conditions at the Hundhammer climatology.

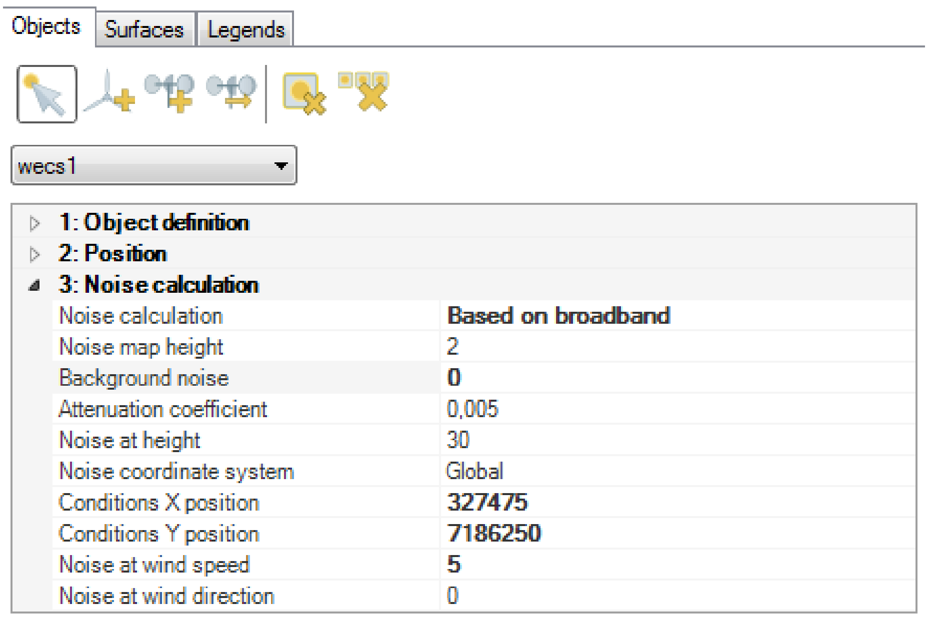


Figure 23 - Example Properties for the CFD noise calculations in the Objects Module

Background noise level is set to 0 db so that only the noise contribution from the wind turbines is estimated. Local standards or project requirements may require the background noise to be included. The attenuation coefficient represents the industry standard for broadband noise. More experienced users might wish to adjust this constant to account for site specific atmospheric conditions and/or to account for an octave band analysis.
Run the *Object module* again by clicking the **Start** button.

A new set of reports is generated with a separate noise map found under **Report > Object representation > Noise**. The map represents the noise pressure level in dB for all locations in the project.

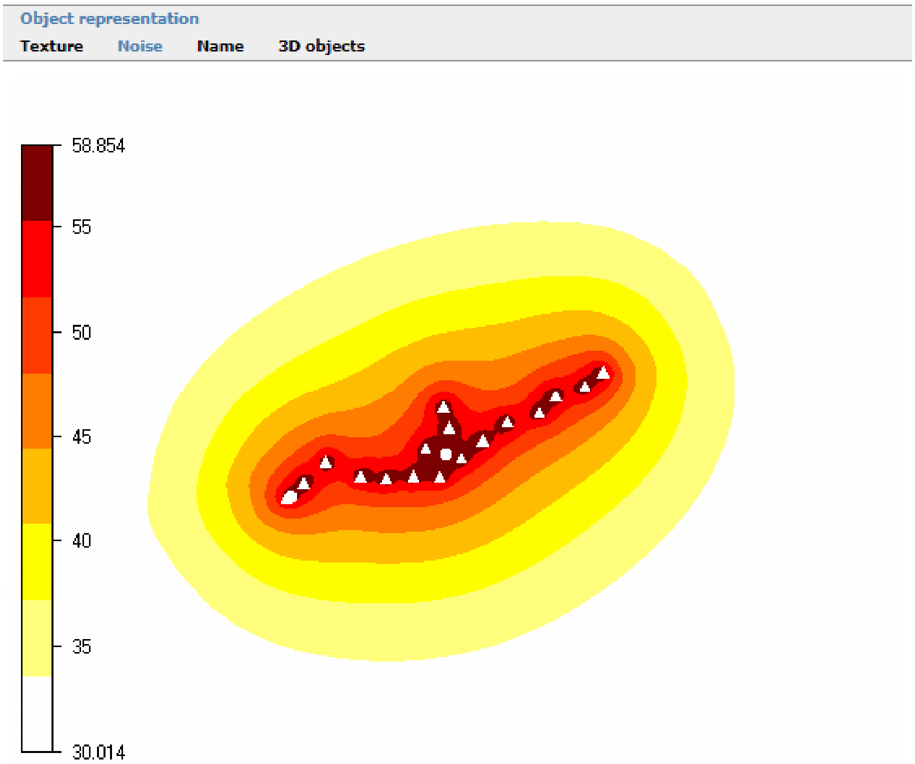


Figure 24 - Noise Calculation report

Results

In the *Results* module you can extract 2D horizontal planes with variables stored in the wind database. The 2D plane is a so-called “terrain plane”, that is a plane following the terrain but at a given height above the terrain. The “terrain planes” can be extracted from ground level up to the "Height of reduced wind database" as specified in the *Wind Fields* module.

In the below “CRT” is used as an abbreviation for “Cartesian”, hence “UCRT”, “VCRT” and “WCRT” means the velocity components along the X-axis (West-East), Y-axis (South-North) and Z-axis (Vertical). The following variables and derived variables are available in the wind database:

Speed scalar X	Wind speed scalar in the West-East direction, UCRT [m/s]
Speed scalar Y	Wind speed scalar in the South-North direction, VCRT [m/s]
Speed scalar Z	Wind speed scalar in the vertical direction, WCRT [m/s]
Speed scalar XY	Wind speed scalar in 2D, $\sqrt{(UCRT^2 + VCRT^2)}$ [m/s]
Speed scalar XYZ	Wind speed scalar in 3D, $\sqrt{(UCRT^2 + VCRT^2 + WCRT^2)}$ [m/s]
Velocity vector XY	Wind speed vector in the horizontal plane, (UCRT,VCRT,0)
Velocity vector XYZ	Wind speed vector in 3D space, (UCRT,VCRT,WCRT)
Direction scalar	Wind direction in the horizontal plane [deg]
Direction scalar relative	Wind direction in the horizontal plane relative to the incoming wind direction [deg]
Turbulent kinetic energy	Turbulent kinetic energy, KE [m²/s²]
Turbulent intensity	Turbulent intensity assuming isotropic turbulence, $100 * \sqrt{\frac{\frac{4}{3} * KE}{\sqrt{(UCRT^2 + VCRT^2)}}}$ [%]
Turbulent dissipation rate	Turbulent dissipation rate, EP [m²/s³]
Pressure	Relative pressure minus hydrostatic term [Pa], the relative pressure has a value equal to zero at the outlet of the domain
Inflow angle	Angle between the wind vector and the horizontal plane [deg]
Wind shear exponent	Exponent alpha of the equation $\frac{speed1}{speed2} = \left(\frac{height1}{height2}\right)^{alpha}$ [-]

As an example the Velocity vector XYZ and the Turbulent kinetic energy at 50 m height is extracted. Remember to press **New** every time a new variable should be activated:

1. Click **New** at the “Properties” window.
2. Choose “Velocity vector XYZ” at the “Normalisation variable” menu.
3. Click **New** at the “Properties” window once more.
4. Choose “Turbulent kinetic energy” at the “Normalisation variable” menu.

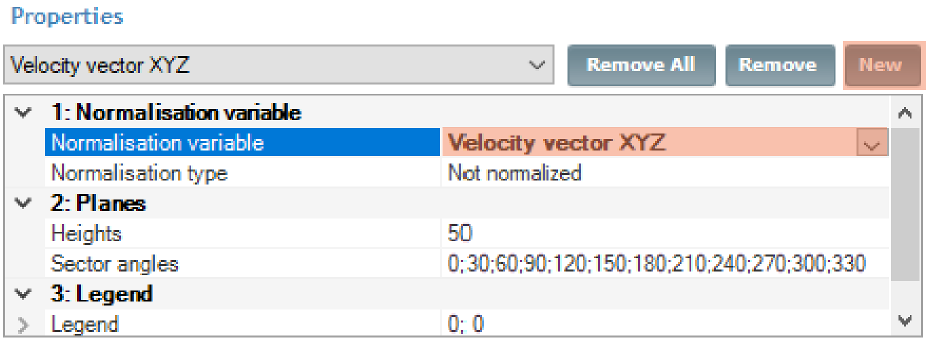


Figure 25 - Properties panel, Results module, extraction of Velocity vectors

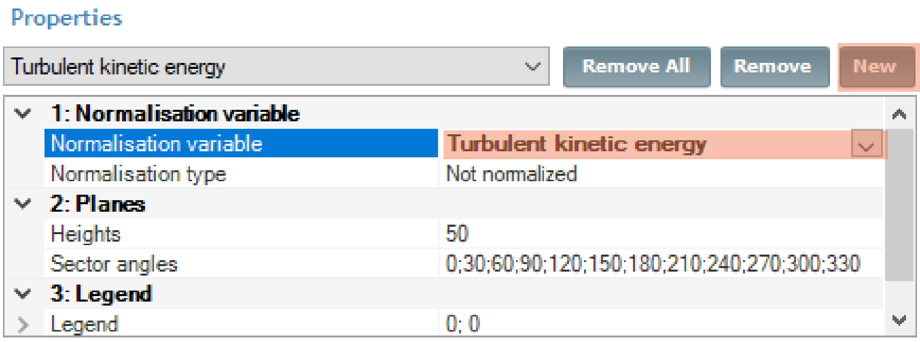


Figure 26 - Properties panel, Results module, extraction of Turbulent kinetic energy

NOTE: *The height for the extracted results has been set to the default value of 50 m above ground level. Typical heights of interest are the turbine hub height and the height of the wind measurements.*

NOTE: *That the default legend setting of 0;0 sets the minimum and maximum values found in each extracted dataset as legend limit.*

NOTE: *The default number of Sector angles is the same as the set sectors in the module Wind Fields.*

Run the module by clicking the **Start** button. The report will display plots of the “Velocity vector XYZ” and “Turbulent kinetic energy” for all 12 sectors as shown in Figure 27 and Figure 28. In total the report will consist of 24 plots, that is: 2 variables x 12 sectors x 1 height = 24 plots. There is a limit of 200 plots that could be generated during one run. Re-run the module if the 200-plot limit impose a restriction on your exploration of the various datasets.

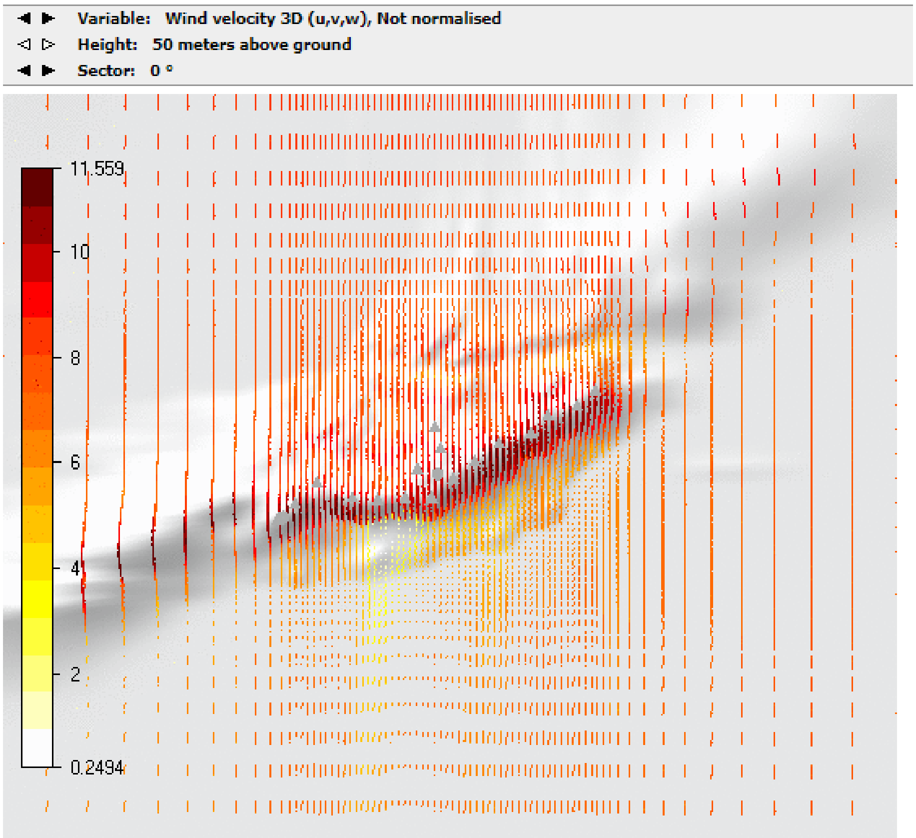


Figure 27 - Results > Report. Wind velocity 3D (m/s) for sector 0° at 50 m a.g.l.

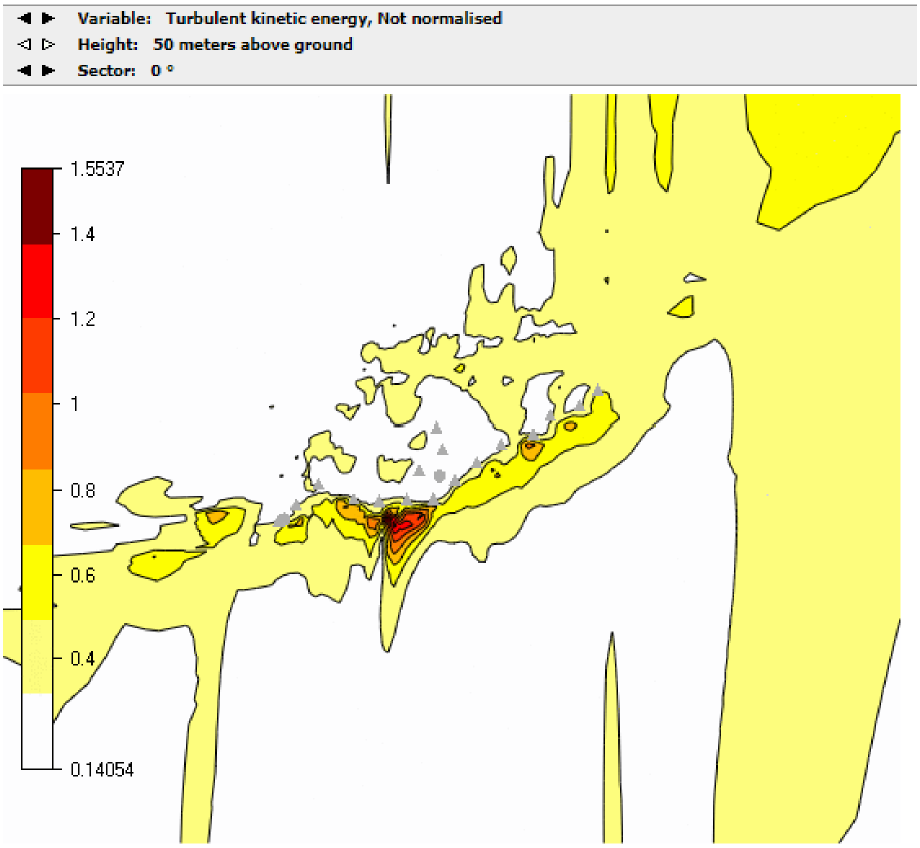


Figure 28 - Results > Report. Turbulent Kinetic energy (m2/s2) for sector 0° at 50 m a.g.l.

If a figure does not reveal sufficient details, then further details can be inspected in the 3D viewer by clicking the information page button 3D. The wind field displayed in Figure 27 indicates a recirculation zone in the mid part of the model, more specifically on the lee side of the ridge when wind is coming in from north. Below in Figure 29, the wind field is inspected in detail by zooming in on the recirculation in the 3D viewer. Notice that the recirculation zone is the reason for the high level of turbulent kinetic energy at the same location, seen in Figure 40.

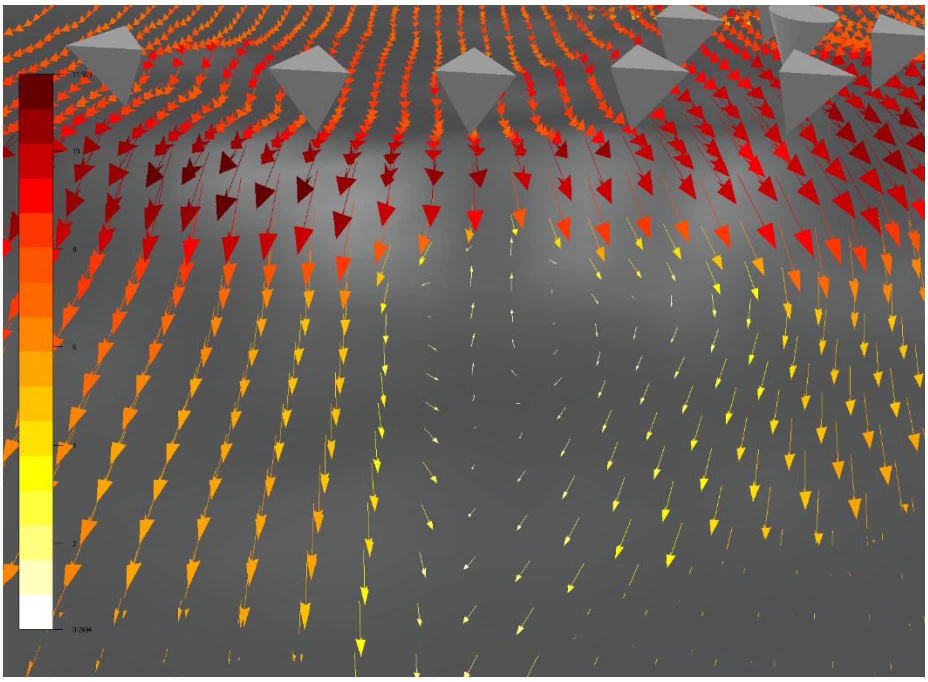


Figure 29 – Inspection of the recirculation zone for sector 0° at 50 m a.g.l.

It is important to note that the *Results* module is a tool for visualization of the wind field database. To observe how the wind fields are affected by shifting terrain elevation and roughness. In the module *Results* the wind fields are not calibrated against climatology data, as such the resulting wind fields reflect the boundary conditions set in the module *Wind Fields*. In the next two modules *Wind Resources* and *Energy,* then the wind database will be calibrated against climatology data.

Wind Resources

The *Wind Resources* module is used to create wind resource maps at chosen heights.

NOTE: *The Results, Wind Resources and Energy modules are independent from each other, it is not necessary to run them sequentially. If, for example, you are only interested in the* *energy production, you can just run the Energy module and it is not needed to run Results and Wind Resources first.*

At least **one** visible climatology must exist in the current layout before running the *Wind Resource* module. All sectors defined in a given climatology must exist in the wind database. The wind resource map is established by weighting the wind database against the climatology. If several climatology objects are available, the wind resource map will be weighted against all of them. This is done by an inverse distance interpolation of the climatology objects. A wind resource map is also established for each climatology separately.

Run the *Wind Resources* module to create a wind resource map for the Hundhammer project at 50- and 80-meters height with the properties given in Figure 30.

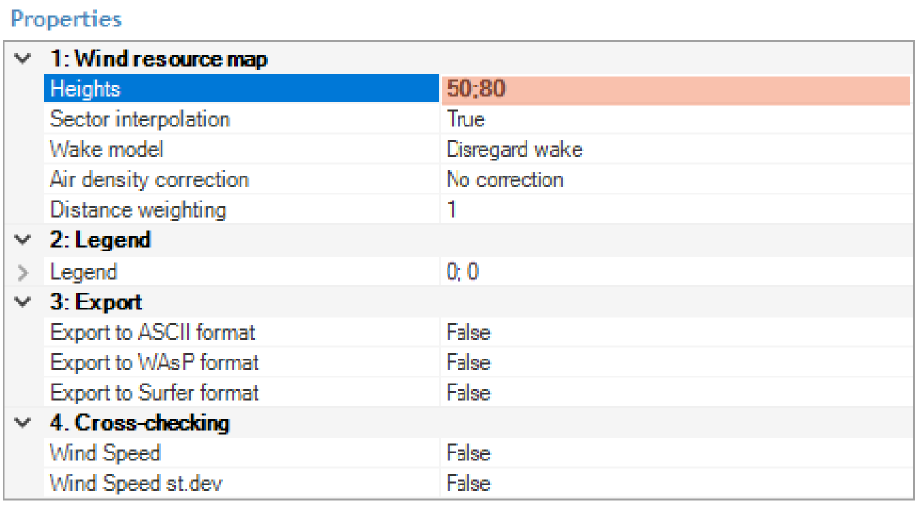


Figure 30 - Properties panel of the Wind Resources module.

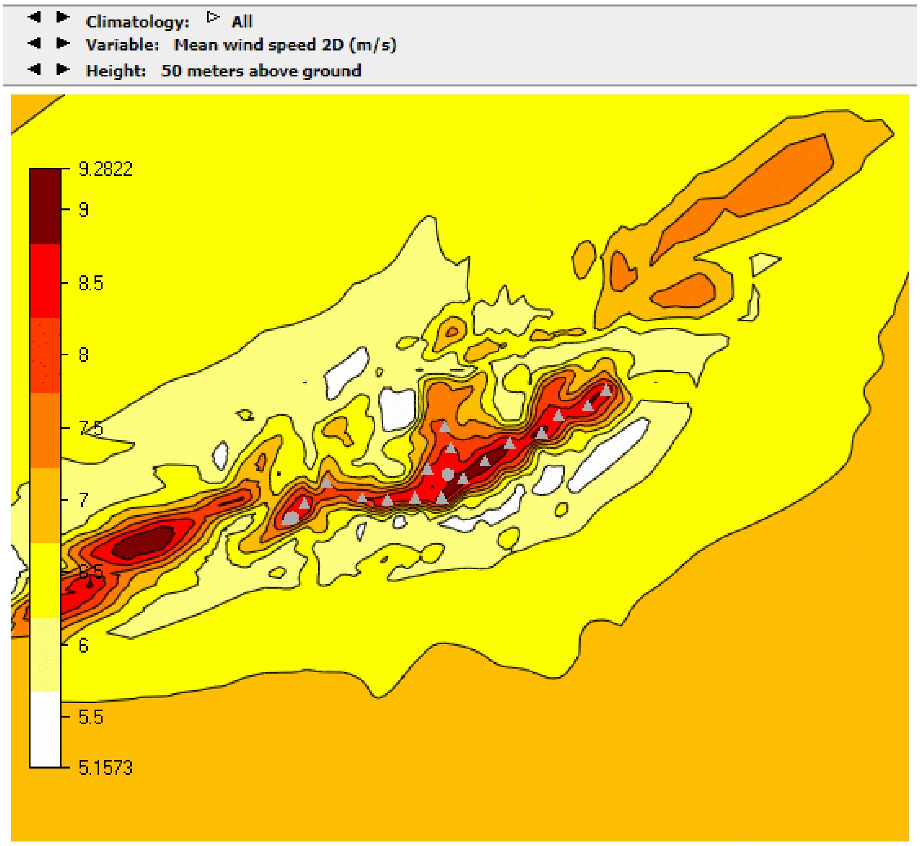


Figure 31 - The wind resource map which is the 2D mean wind speed at 50 m a.g.l. weighted against all climatologies.

Wake Modelling

If the user is concerned about wake losses in the wind farm (also known as wind park effect), one of the 3 wake models can be activated. These wake models can be used both in the module *Wind Resources*, and *Energy*. A complete description of the models is provided in the description page of the *Wind Resources* module.

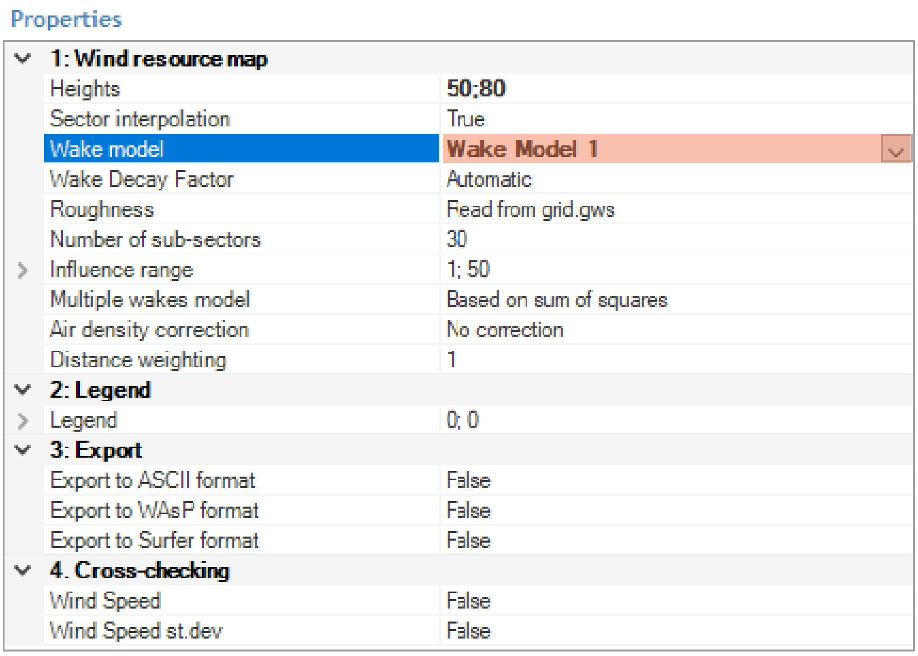


Figure 32 – Activation of a wake model in the Properties panel of the Wind Resources module.

Activate “Wake Model 1” and re-run the module. The Report now contains new pages, displaying the reduced wind speed or the so-called wake deficit, see Figure 33.

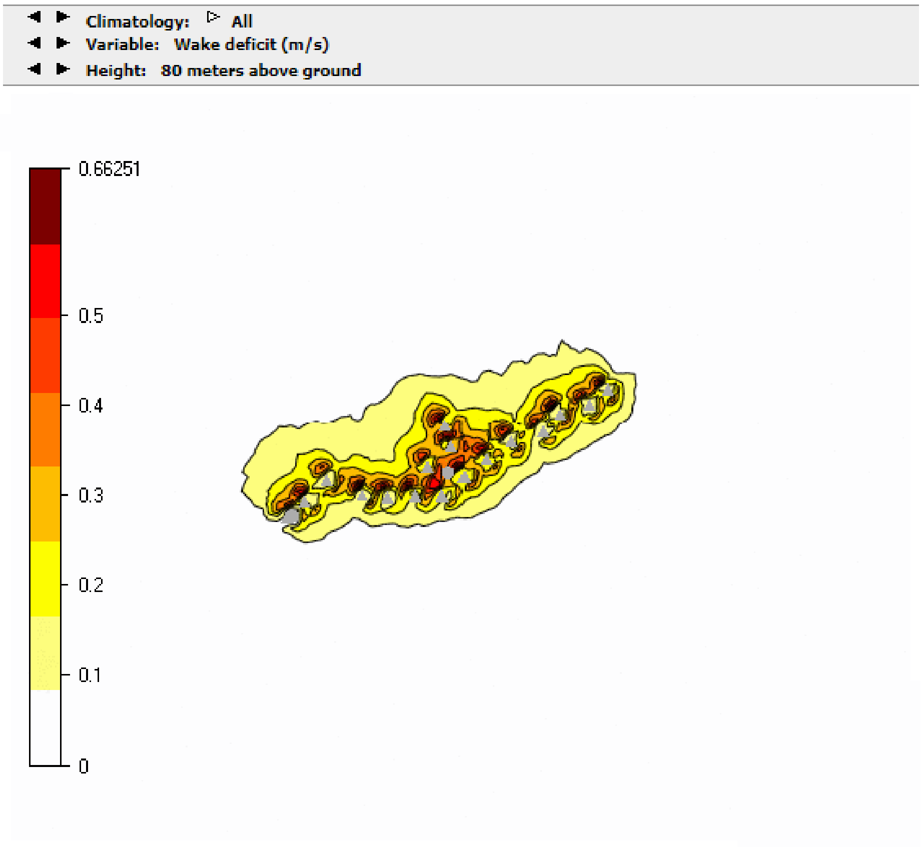


Figure 33 - Wake deficit at hub height, wake model 1.

Energy

The Annual Energy Production (AEP) is the most important parameter to be estimated in most micrositing projects. For a given wind condition, the available power is proportional to the third cube of the wind speed. This means that an uncertainty in wind prediction is largely increased when computing the available power; so high-quality wind modelling is particularly appreciated within micrositing.

Run the *Energy* module with the default settings except for the activation of a wake model as seen in Figure 34.

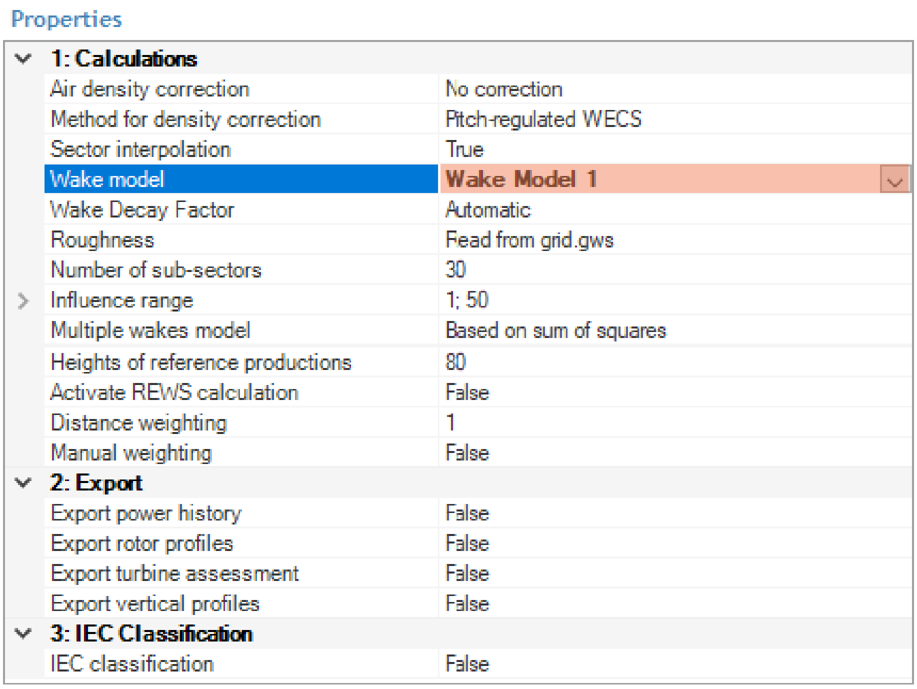


Figure 34 - Properties panel for the Energy module.

The *Energy* report gives several tables summarizing the AEP including wake losses on the wind farm level, see Figure 35, and on a per turbine level. For each climatology object, two AEPs are given in the report.

- The first one is obtained from the frequency table of the climatology files.
- The second is obtained by Weibull fitting the frequency table.

An estimate of the AEP is given per climatology and for all available climatologies, using the same interpolation technique as in the module *Wind Resources*.

Climatology	Distribution	AEP with wake losses	Wake loss %
0150-Tommerhol..	Frequency table	138.4683	5.43
0150-Tommerhol..	Weibull distribution	138.5328	5.45
0801-Hundhamme..	Frequency table	140.8233	4.98
0801-Hundhamme..	Weibull distribution	140.8934	5.00
All	Frequency table	139.9848	5.11
All	Weibull distribution	140.0542	5.13

Table 1. Energy production in GWh/y based on climatology represented as frequency table, Weibull distribution and time series (time series are calculated only if power history and IEC classification are active, note that missing values in the time series are treated as 0 speed values in the production calculation).

Air density (kg/m^3)	Wake model	Multi-wakes model	Roughness (m)	Amb. Turb. Int. (%)	Sub-sectors	Influence range (Rotor diameter)
No correction	1	2	Variable	-	30	1.0 - 50.0

Table 2. Site and wake characteristics.

Figure 35 - Energy report on the wind farm level including wake losses. For more information on a per turbine level click on any number in the column "AEP with wake losses".

There is a 1.67% difference in the estimated AEP when using the two different climatologies, ((140.8233 – 138.4683)/140.8233)*100 = 1.67%. Ideally there should be no difference in the estimated AEPs, but since both the wind measurements and the numerical simulations contains errors there will always be discrepancies.

One of the errors associated with numerical simulations stems from the discrete nature of the numerical model. In particular if the numerical model has a too low resolution the discretization error can be significant. Ideally the numerical model should have a sufficiently high resolution, such that a further increase of the resolution would not change the numerical results. In this case the numerical model is said to be grid independent. See next section where grid independency with respect to AEP is investigated for the Hundhammer project.

A Grid Sensitivity Study

In this section we investigate the effect of discretization errors on the numerical results. We use the AEP for the whole wind farm as a metric for the numerical results, comparing the AEP obtained for five different numerical models with the number of cells given in Figure 36.

N	Nx	Ny	Nz
4 370	23	19	10
24 320	38	32	20
90 280	74	61	20
375 000	150	125	20
750 480	212	177	20

Figure 36 - Number of cells in models used for grid independency tests

Ideally a so-called grid independent solution is obtained, giving the minimum discretization error, when a further increase in the number of cells doesn't change the AEP.

Climatology	Distribution	Gross AEP	Wake loss %
Climatology1	Frequency table	159.2292	-
Climatology1	Weibull distribution	159.4079	-
Climatology2	Frequency table	140.0524	-
Climatology2	Weibull distribution	140.1058	-
All	Frequency table	146.7715	-
All	Weibull distribution	146.8683	-

Table 1. Energy production in GWh/y based on climatology represented by the frequency table and by the Weibull distribution.

Air density (kg/m^3)	Wake model	Multi-wakes model	Roughness (m)	Amb. Turb. Int. (%)	Sub-sectors	Influence range (Rotor diameter)
No correction	0	2	-	Variable	5	1.0 - 50.0

Table 2. Site and wake characteristics.

Figure 37 - Report panel for the Energy module, Hundhammer project (~5 000 cells).

Climatology	Distribution	Gross AEP	Wake loss %
Climatology1	Frequency table	152.9213	-
Climatology1	Weibull distribution	153.0664	-
Climatology2	Frequency table	140.9978	-
Climatology2	Weibull distribution	141.0464	-
All	Frequency table	145.1288	-
All	Weibull distribution	145.2123	-

Table 1. Energy production in GWh/y based on climatology represented by the frequency table and by the Weibull distribution.

Figure 38 - Report panel for the Energy module, Hundhammer project (~25 000 cells).

Climatology	Distribution	Gross AEP	Wake loss %
Climatology1	Frequency table	146.8367	-
Climatology1	Weibull distribution	146.9455	-
Climatology2	Frequency table	146.6540	-
Climatology2	Weibull distribution	146.7458	-
All	Frequency table	146.6858	-
All	Weibull distribution	146.7840	-

Table 1. Energy production in GWh/y based on climatology represented by the frequency table and by the Weibull distribution.

Figure 39 - Report panel for the Energy module, Hundhammer project (~100 000 cells).

Climatology	Distribution	Gross AEP	Wake loss %
Climatology1	Frequency table	149.0998	-
Climatology1	Weibull distribution	149.2445	-
Climatology2	Frequency table	147.3412	-
Climatology2	Weibull distribution	147.4403	-
All	Frequency table	147.9218	-
All	Weibull distribution	148.0365	-

Table 1. Energy production in GWh/y based on climatology represented by the frequency table and by the Weibull distribution.

Figure 40 - Report panel for the Energy module, Hundhammer project (~400 000 cells).

Climatology	Distribution	Gross AEP	Wake loss %
Climatology1	Frequency table	149.5149	-
Climatology1	Weibull distribution	149.6543	-
Climatology2	Frequency table	147.0314	-
Climatology2	Weibull distribution	147.1283	-
All	Frequency table	147.8624	-
All	Weibull distribution	147.9743	-

Table 1. Energy production in GWh/y based on climatology represented by the frequency table and by the Weibull distribution.

Figure 41 - Report panel for the Energy module, Hundhammer project (~800 000 cells)

The AEP calculated with the five different models is summarized in Figure 54. Important discretization errors are present for the two models with ~5000 and ~25000 cells. Considering a grid independency reached with a model of 800 000 cells, a model comprising just 5 000 cells gave errors of 20 % for one of the climatology data. The model of size 100 000 cells provided a good estimation of the AEP, the errors could be considered within 3%; considering a grid independency reached with 800 000 cells.

It´s important to stress that the characteristics of a grid which is leading to grid independency is its resolution. So, for wider areas, a higher number of cells are required to minimize the discretization errors.

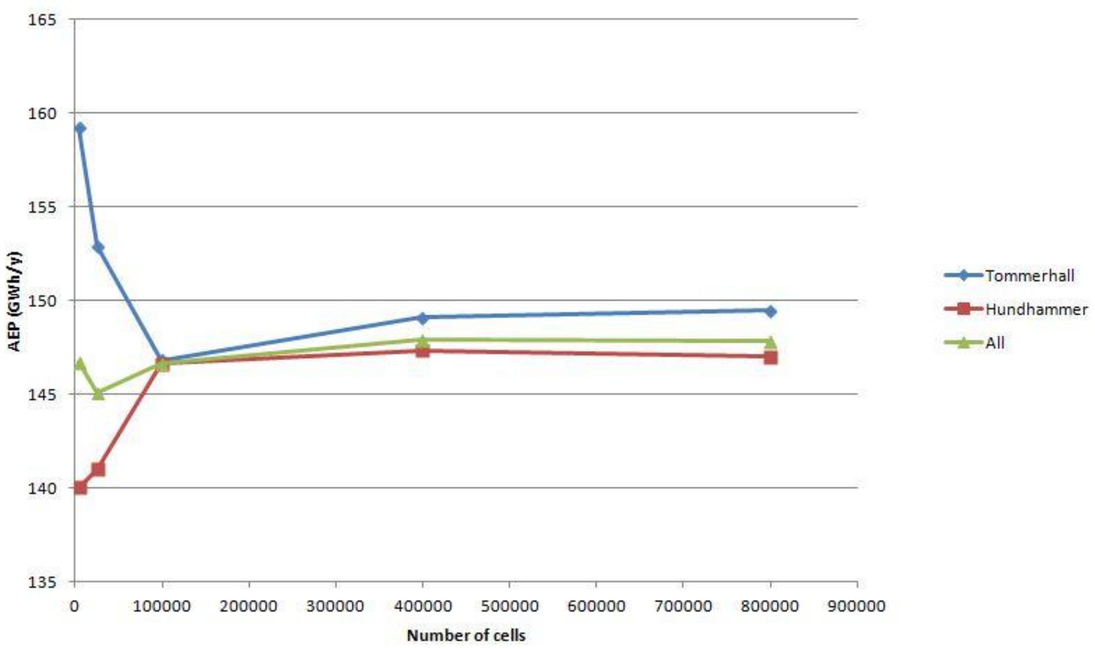


Figure 42 - AEP for the two climatologies against number of cells used in the 3D models.

Configure Particle Traces in GLview Pro

Generation of the Wind Visualization File

How to generate a 3D wind visualization file (.vtf) in the menu”3D Visualization” is shown below:

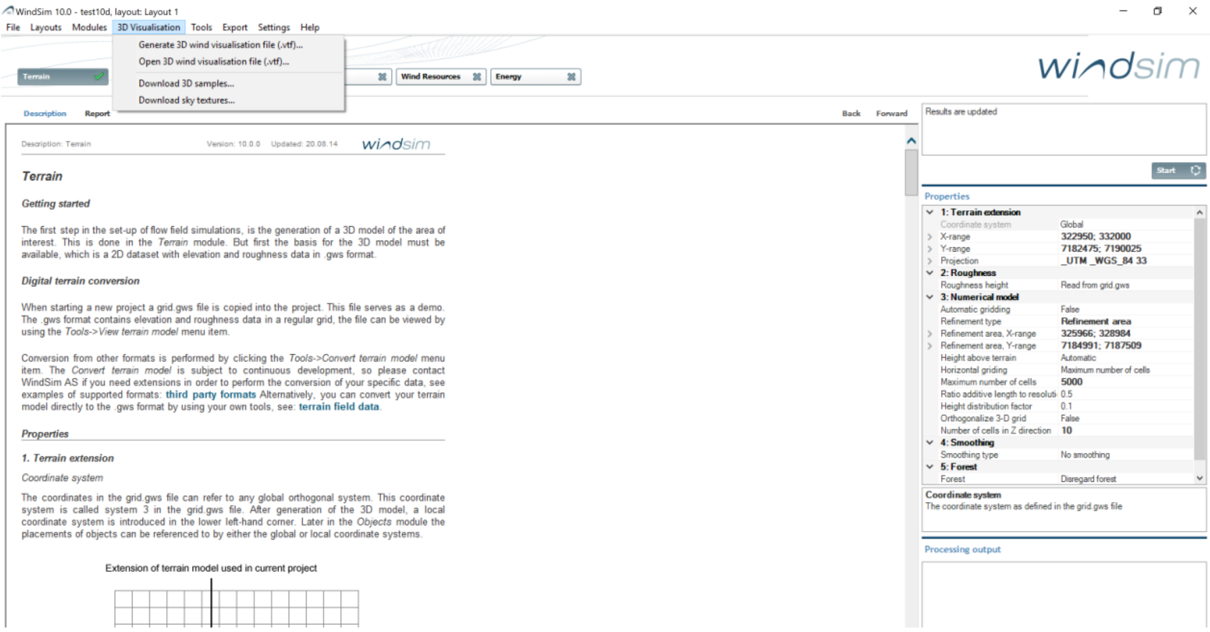


Figure 43 - The .vtf file is generated from the 3D Visualization menu on the toolbar

The generated .vtf file must contain the 3D velocity vector, which is the vector field used to establish the particle traces.

1. Click **3D Visualization > Generate 3D wind visualisation file (.vtf)**. A DOS pop-up window will appear.

2. Type “l” to choose from the list of sectors.

3. Choose the sector 000 by simply pressing “000”.

4. Type “1” to transfer the default variables to the .vtf file.

5. Open the 3D Visualization file by clicking **3D Visualisation>Open 3D wind visualization file (.vtf)…** and then selecting the 000.vtf file.

Activate the VELOCITY_3D as vector by pressing the “Apply” button as shown in Figure 44. The max and min values of the chosen scalar and vector fields will appear in the blue info window in the lower right corner:

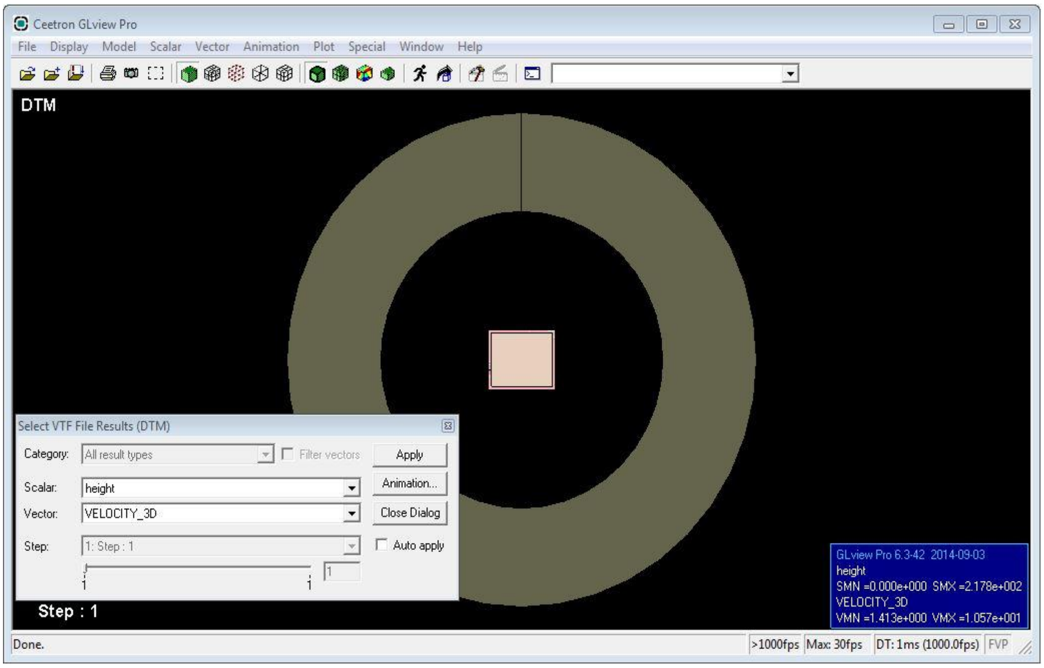


Figure 44 - Set Vector to VELOCITY_3D and hit Apply in the dialogue box to the bottom left in the screen above

Setting the Attributes

Zoom in towards the terrain of the model, as shown below. Rotate the model by using the right mouse button. Then set the attributes of each part in the “Change Part Attributes…” window found under “Model”

Then the grid has to bet set invisible first by choosing “grid” and then unselecting the attribute “Visible”. Finish the process by pressing the “Apply” button.

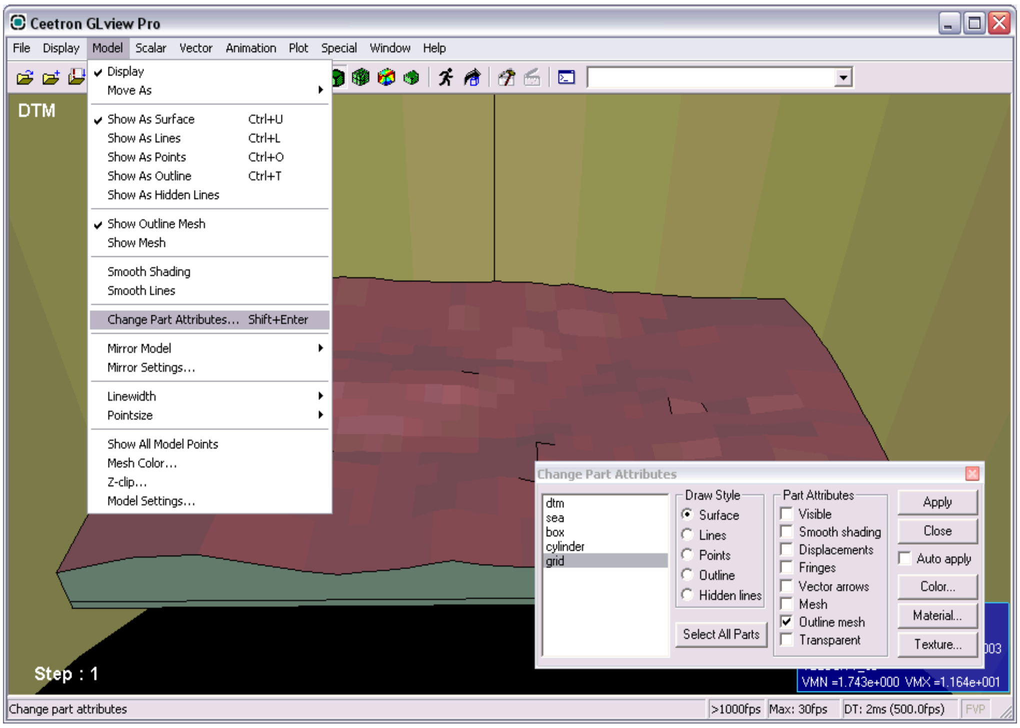


Figure 45 - Configure the view to display the relevant parts

In the **Change Part Attributes** the default setting of all parts which is “Visible” and “Outline mesh” could be reset. Selecting all parts is done by clicking on the first part, while holding down the shift key clicking on the last part.

While all parts are selected, uncheck the “outline mesh” box and check “smooth shading” to give a better visual appearance (see Figure 45).

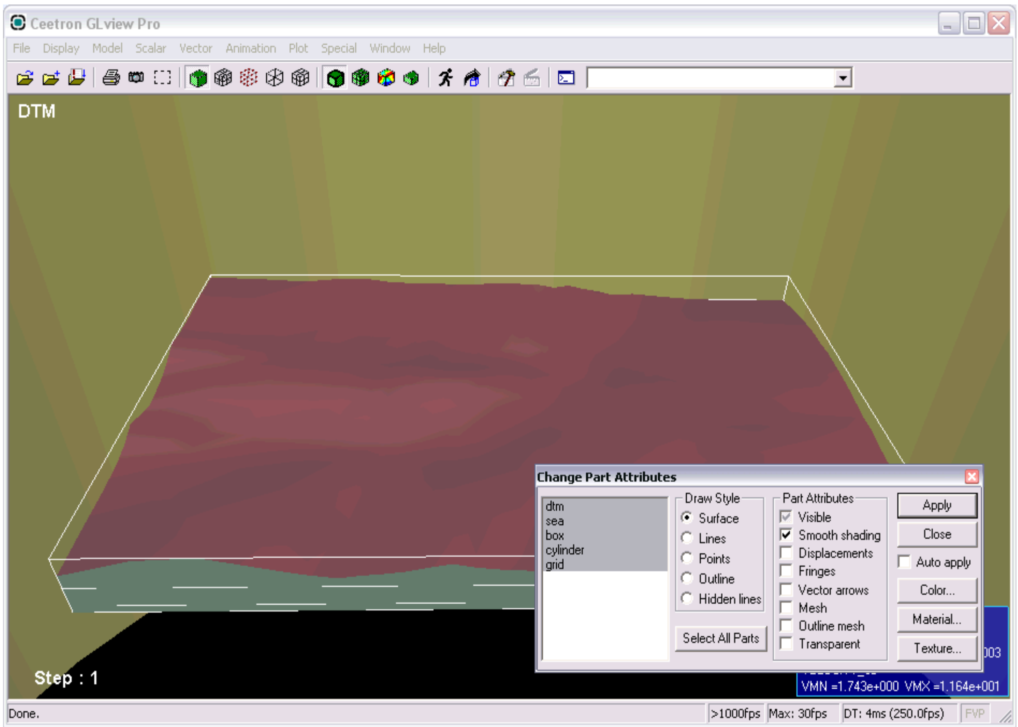


Figure 46 - Perfecting the visualization to your projects specific requirements is easy in WindSim!

Creating the Particle Traces

Before calculating particle traces you need to define the start positions for the traces. Each trace represents the path of an artificial particle without mass following the wind field in a passive manner. In order to get a good representation of the wind field it is convenient to specify many traces. This is done by specifying the start positions within a box, with a given number of start positions in x, y, and z direction. To open the window “Particle Traces” click **Vector > Particle Tracing…** (see Figure 47).The dimension of the box will be model dependent. In the model below the box is positioned towards the North border as the wind field with incoming wind from North has been loaded (000.vtf).

The coordinates (X1,Y1,Z1) and (X2,Y2,Z2) represent the front left bottom point and the far upper right point respectively. In order to specify the dimensions of the box, you need to set these coordinates and then click “Apply” (see Figure 48).

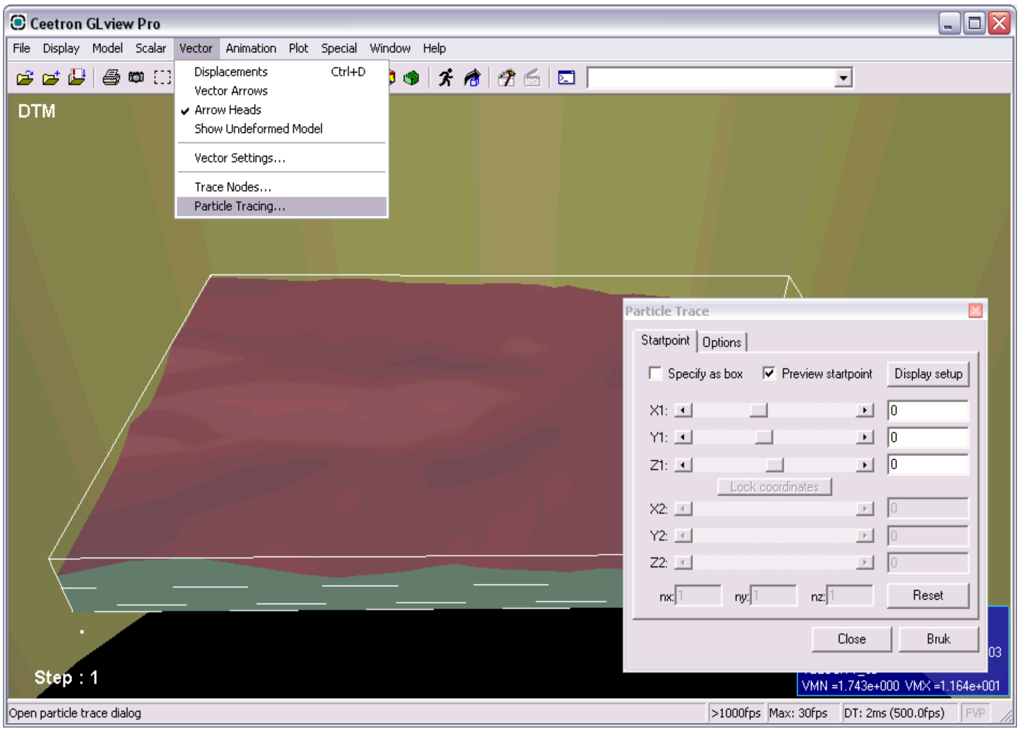


Figure 47 - Open the dialogue box from the Vector menu on the toolbar

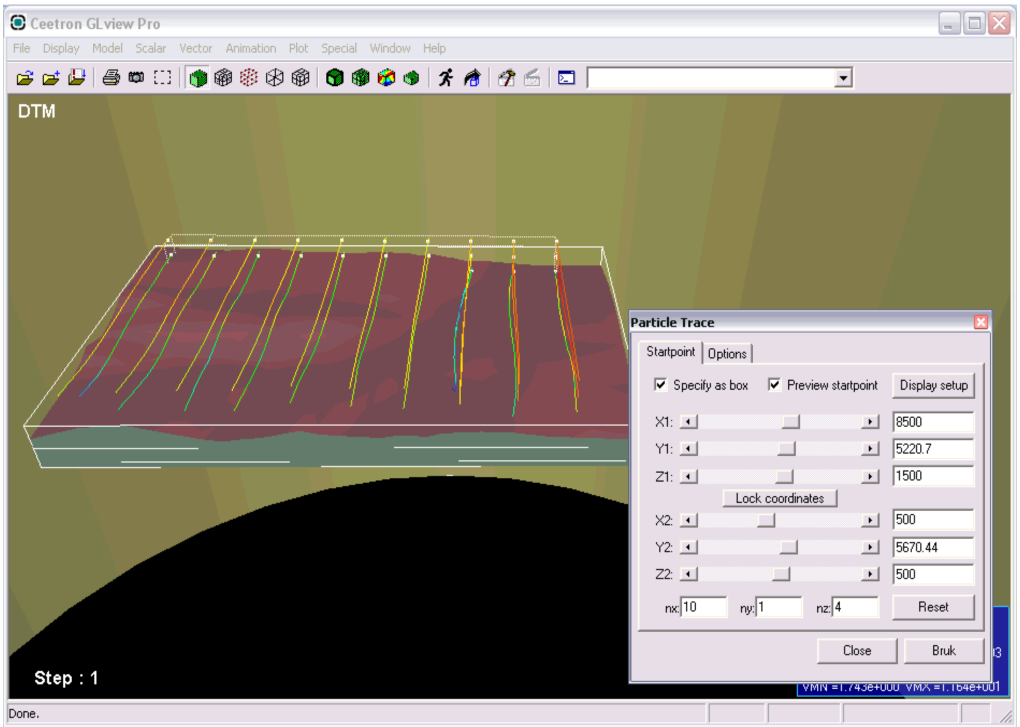


Figure 48 - Replicate the settings above to achieve the same particle traces

Under the folder "Options" the user can set the direction for the calculation of the traces. In the given case the calculation is only done forward, as the box with starting point is put near the inlet border therefore the traces will pass through most of the model.

In some cases, the default method for integrating the paths, the Euler method, will fail. If the traces do not follow the terrain, but appears like straight lines pointing towards the sky, then it is recommended to switch to the linear integration method.

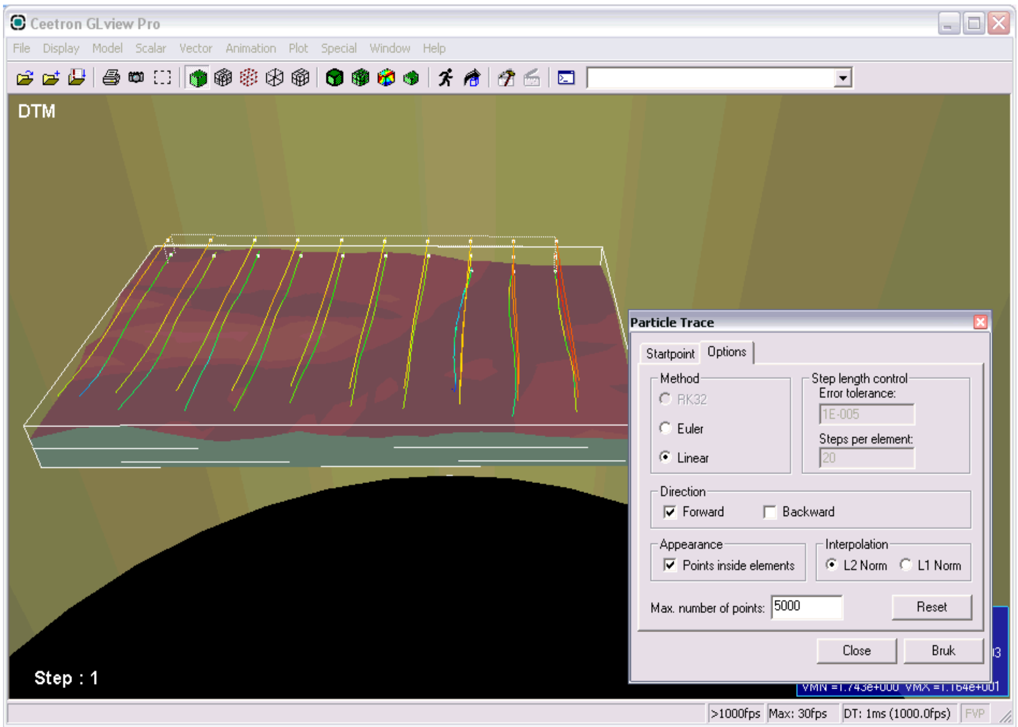


Figure 49 - Set the direction of your particle traces

Animating the Traces

From the window “Particle Trace” the folder “Startpoint” should be chosen. Then by choosing the “Display Setup” button in the upper right corner the window “Particle Trace Display Setup” will open.

The setting for animating the particles is set in this window.

- Check the box “Animate” to unlock the animation options.
- Check the box “Incremental trace”
- Uncheck the box “Complete trace”
- Press the “Apply” button.

Try to animate using line (worm) by check the box “Tail”. Play with the settings and see what works for your model.

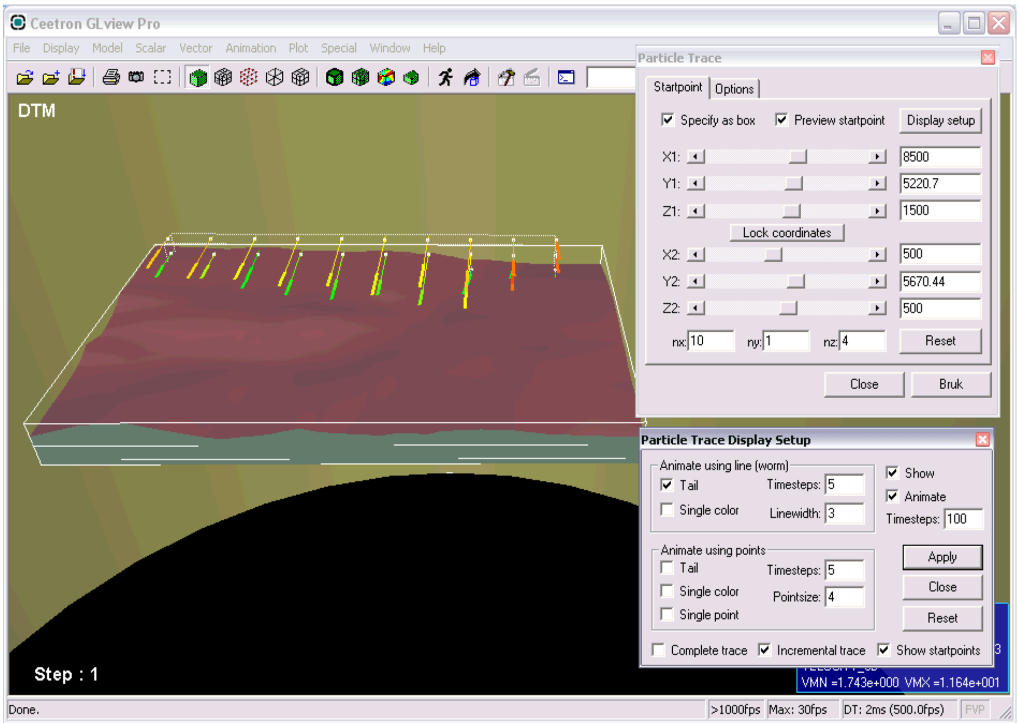


Figure 50 - The user has full control over the simulation parameters to design an animation to suit any project requirements

Textures

The final touch for your presentations is obtained by adding textures. Textures could be added to the different parts. For example, adding a satellite photo or a scanned map to the DTM (Digital Terrain Model) would make it easier to recognize locations in the model. The only purpose of the cylinder surrounding the model is to add a “sky texture”.

A texture is added to a part using the “Change Part Attributes” window introduced above. Select the part and then click the button “Texture…”. In the “Texture Settings For Part(s)” window, a picture must first be loaded and then this picture must be pasted over the part from a given direction, defined by the “Plane”. The picture is stretched over the part if the box “Clamp” is checked. Otherwise, a tile pattern is generated.

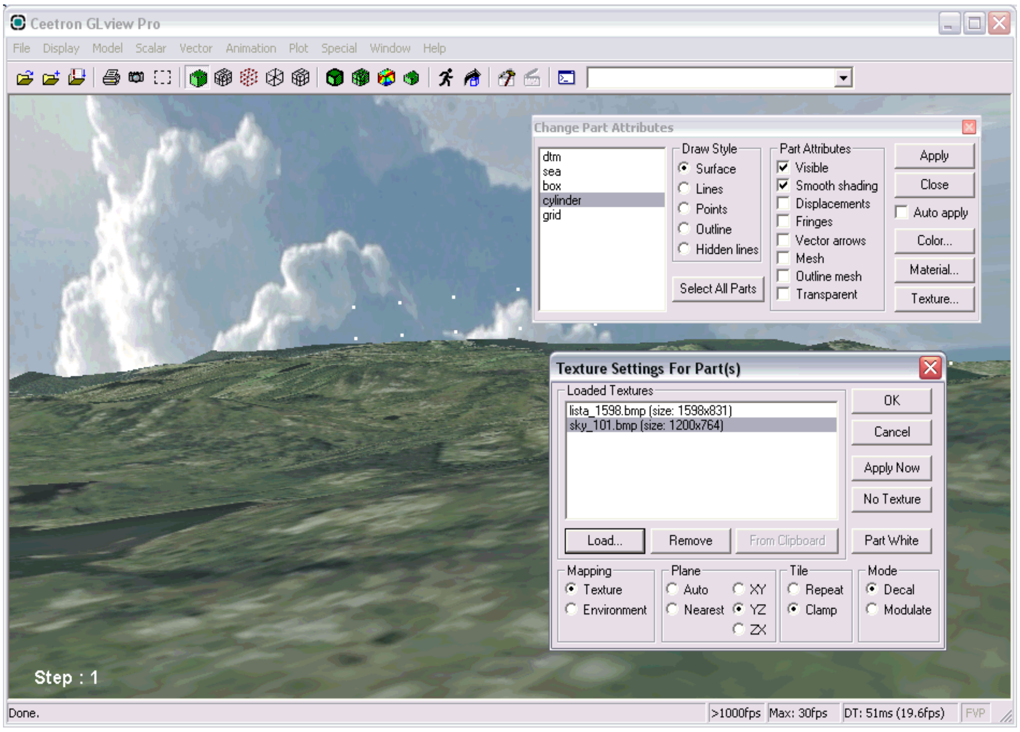


Figure 51- Adding a sky texture greatly enhances the quality of the visualization

Save and Share

When you are satisfied with the setting, you can save all the settings to a new .vtf file in the menu “File” – “Export to File” – “VTF File…”. Send us your .vtf file and we will publish it at windsim.com under: [3D Visualizations (windsim.com)]

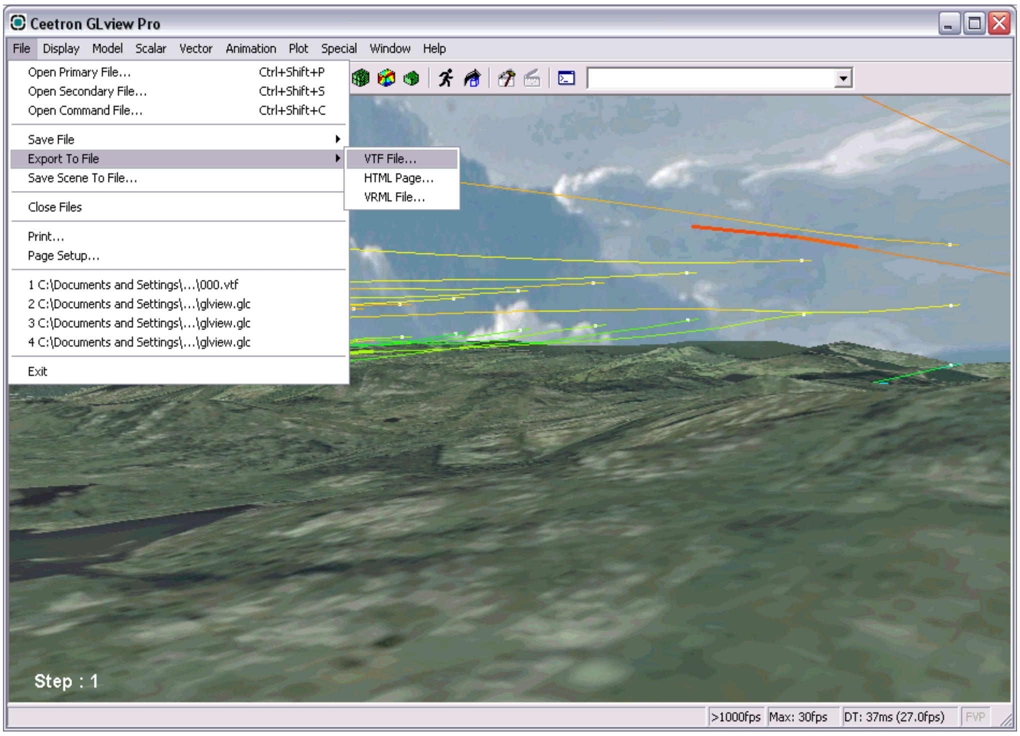


Figure 52 - Just one example of the infinite customizability of WindSim visualization

Configure Isosurfaces in GLview Pro

Generation of the Wind Visualization File

Generate a 3D wind visualization file (.vtf) in the menu ”3D Visualization” following the same procedure as in paragraph “Configure Particle traces in GLview Pro”.

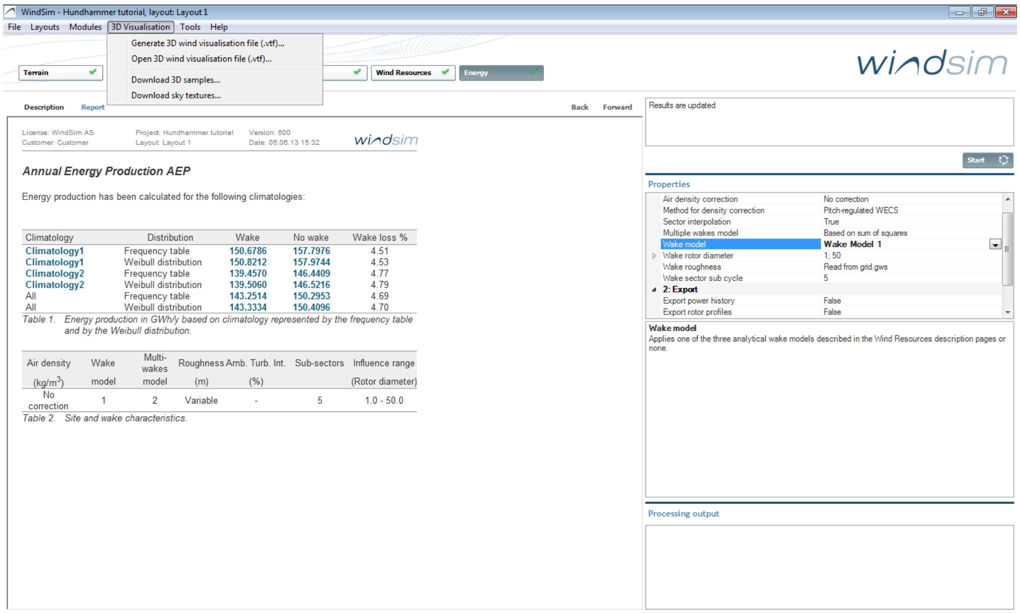


Figure 53 - Locate the menu show above

The generated .vtf file must contain the 3D velocity vector, which is the vector field used to establish the particle traces. Moreover, the .vtf has to contain the scalar variable that has to be plotted as isosurface.

Open 3D Visualization file

Open the 3D wind visualization file (.vtf) in the menu *3D Visualization*. Activate SPEED_2D as Scalar by pressing the button “Apply”, then the min and max values of the chosen scalar fields will appear in the blue info window in the lower right corner (see Figure 54).

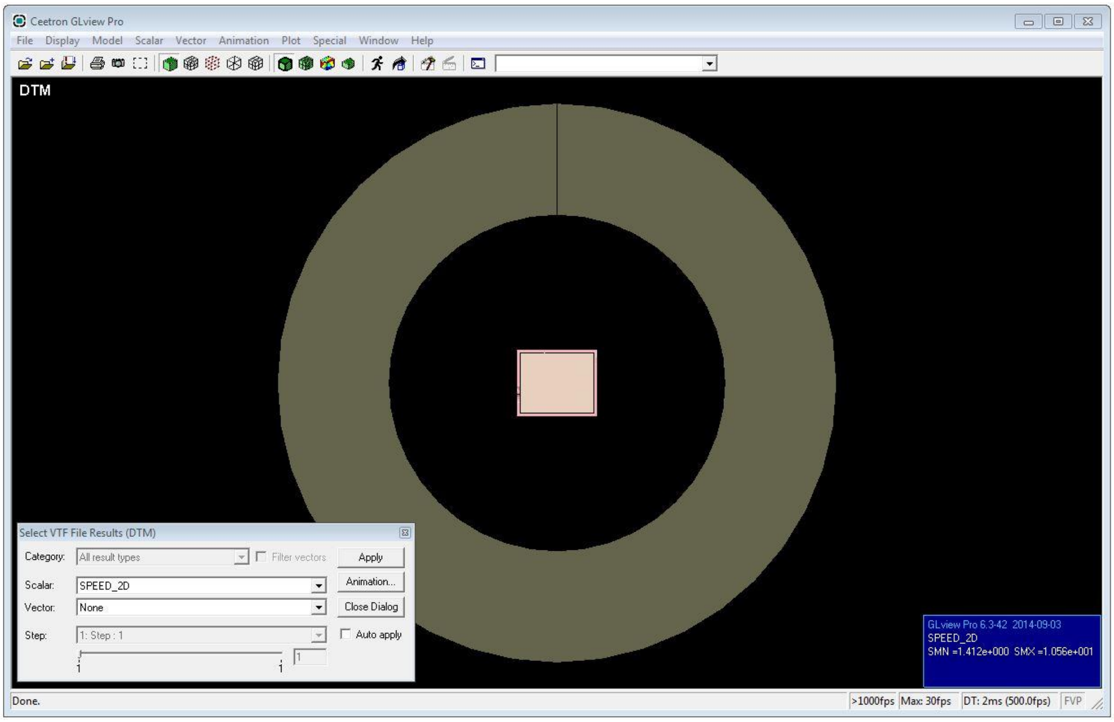


Figure 54 - Set the Scalar to SPEED_2D and hit Apply

Setting the Attributes

Zoom in towards the box and rotate the model by using the mouse buttons. Then set the attributes of each of the parts in the “Change Part Attributes” window found under “Model” – “Change Part Attributes…”. First the grid has to bet set transparent by selecting the grid and check the attribute “Transparent” then press the button “Apply” (see Figures 55 and 56); then the transparency level has to be put to the maximum value (0.0), click on the Material button to set the transparency to 0.0 as shown in Figure 57, then click on OK and Apply to confirm.

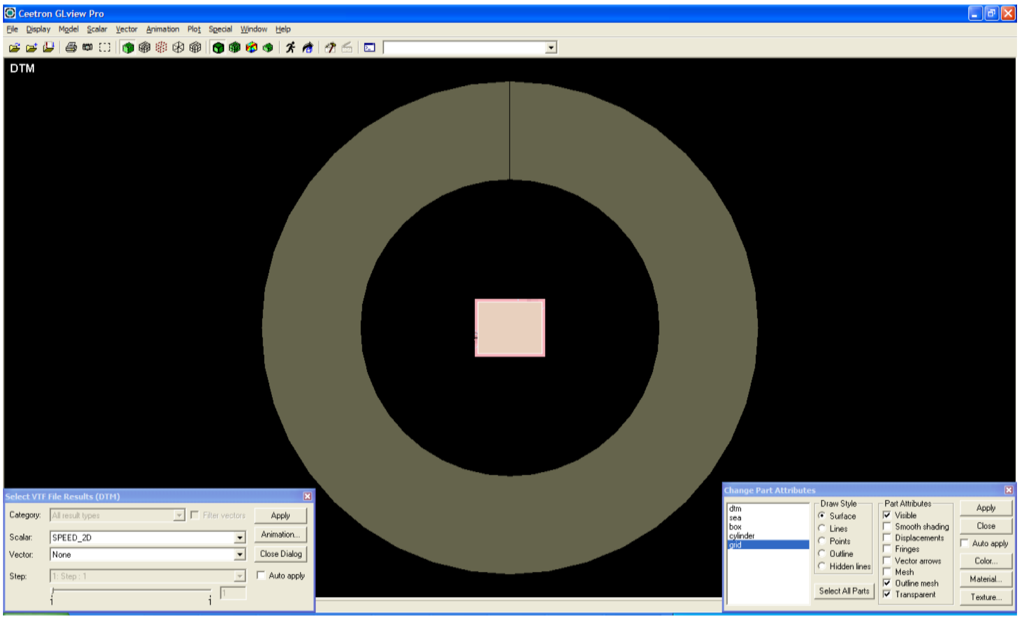


Figure 55 - Open the Change Part Attributes dialogue box

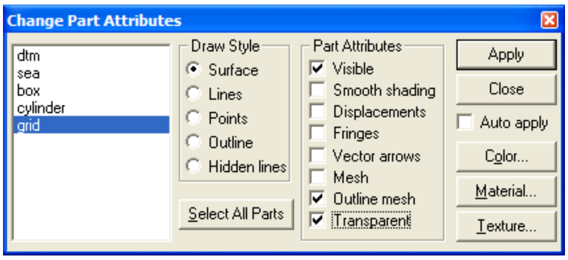


Figure 56 - Replicate the selections made above

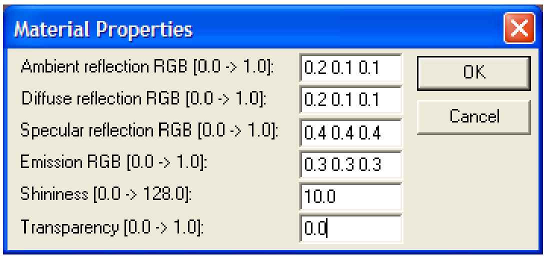


Figure 57 - Open the Material Properties box by clicking the Material... button

Creating the Isosurfaces

Select the “Isosurface...” option under the Scalar menu (see Figure 58). Select the value of the loaded scalar that you want to display and press Apply, as shown in Figure 59. Rotate and zoom with the mouse to obtain a good view of the flow field.

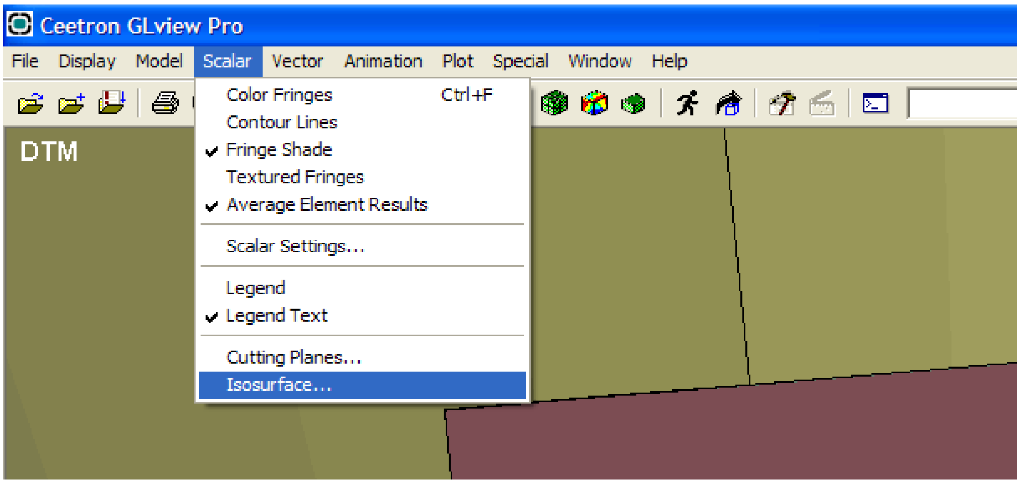


Figure 58 - Open the Isosurface dialogue box from the Scalar menu

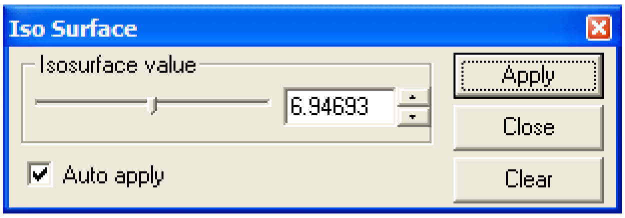


Figure 59 - The above value generates a nice plot for the Hundhammer project

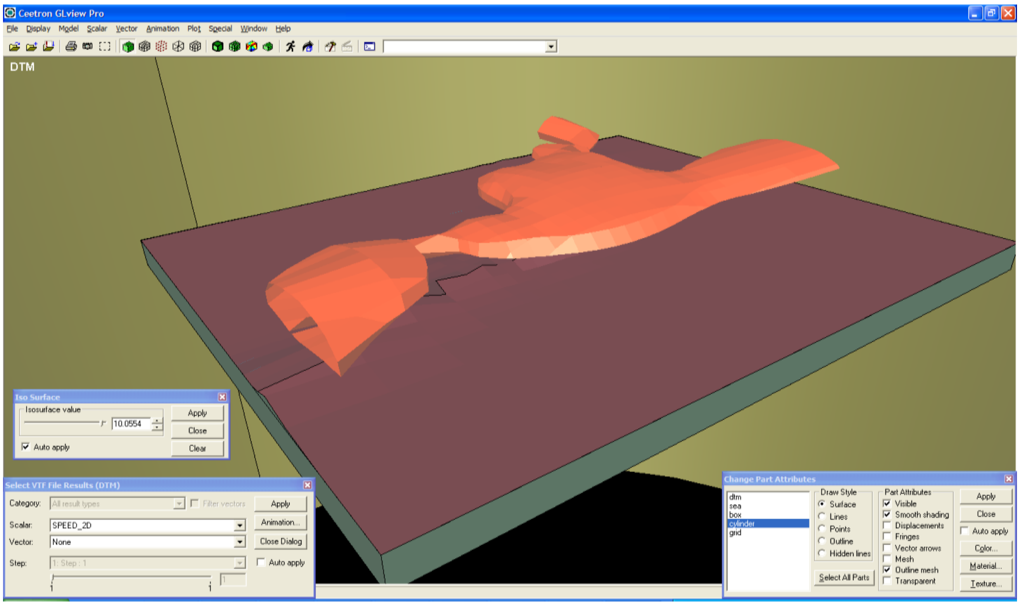


Figure 60 - The Isosurface is now displayed relative to the terrain to visualize the flow in the wind farm

Textures - Save and Share

For adding textures and sharing data with others see the procedures described in the above section “Configure Isosurfaces in GLview Pro”.

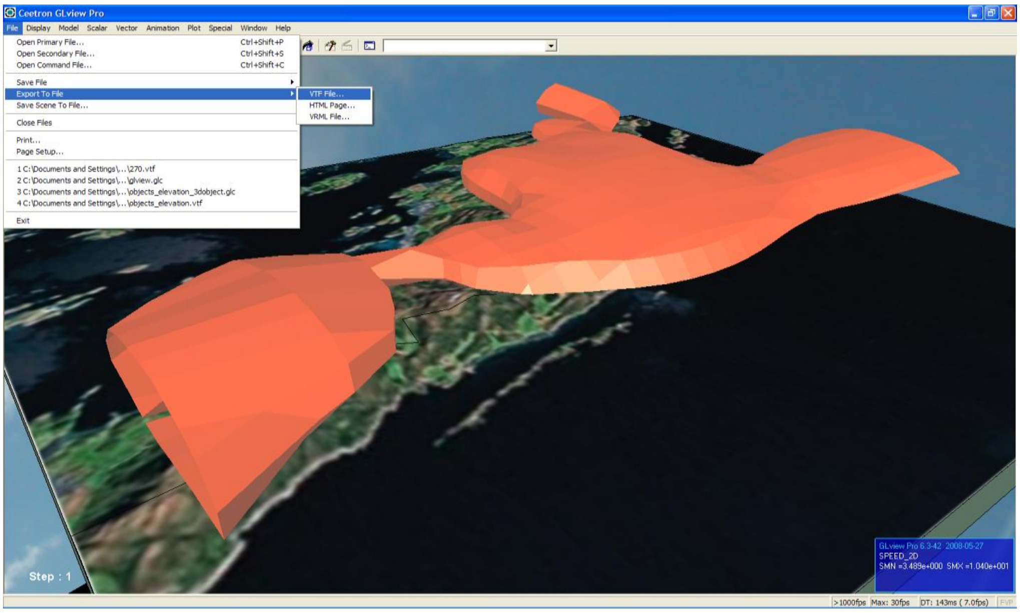


Figure 61 - A texture adds a nice touch to the graphics for use in presentation and reports!