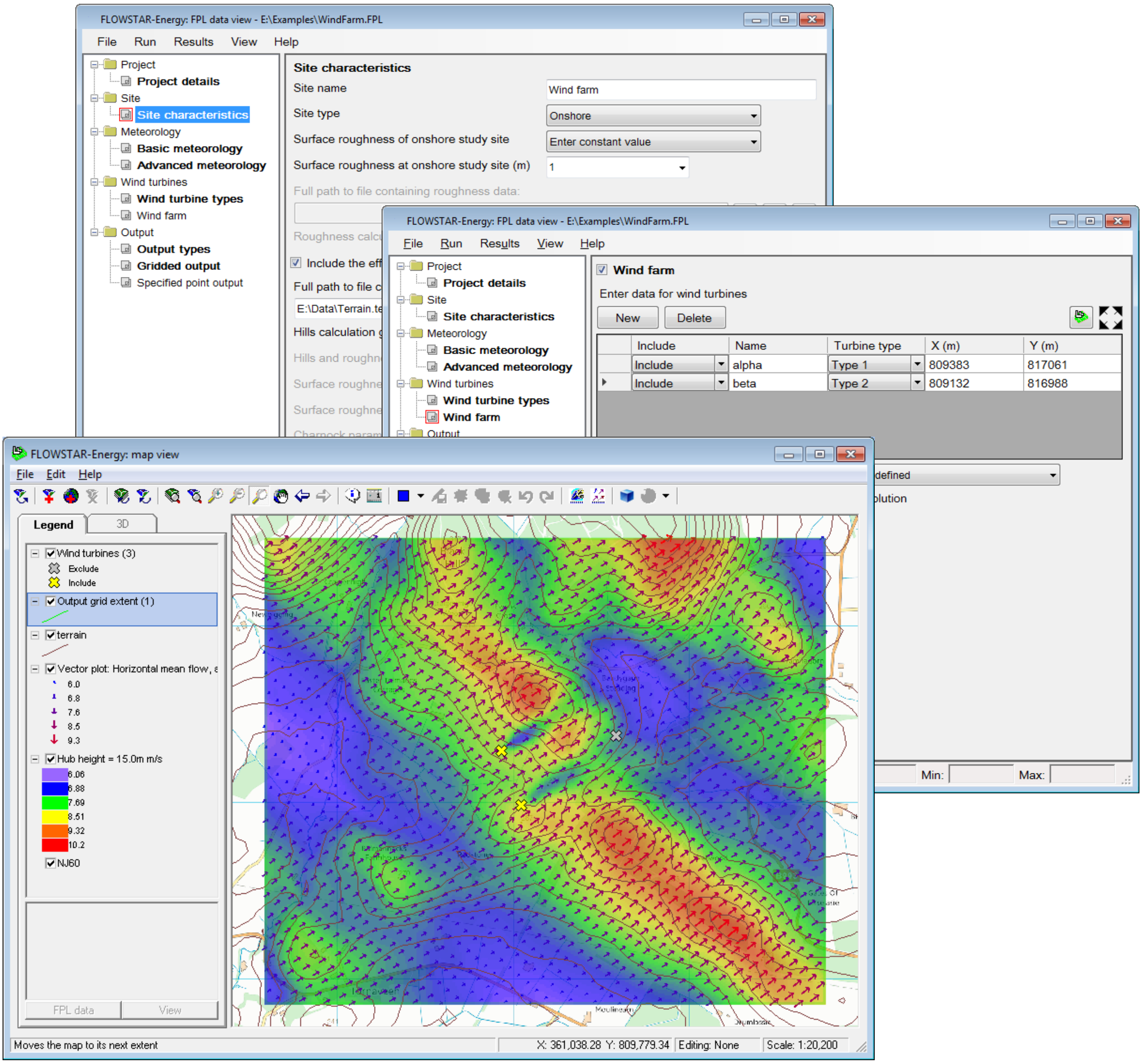
# FLOWSTAR-Energy




**User Guide** **CERC**

# FLOWSTAR-Energy

# User Guide

**Version 5.1**

December 2015

Cambridge Environmental Research Consultants Ltd.
3, King's Parade
Cambridge
CB2 1SJ
UK

| | |
|---|---|
| Telephone: | +44 (0)1223 357 773 |
| Facsimile: | +44 (0)1223 357 492 |
| Email: | help@cerc.co.uk |
| Website: | www.cerc.co.uk |

# Contents

# SECTION 1 Introduction

## 1.1 About FLOWSTAR-Energy

***What is FLOWSTAR-Energy?***

FLOWSTAR-Energy is a practical, short-range model that simulates air flow and turbulence over complex terrain, including the effects of stratification, variable surface roughness and wind turbines in operation. FLOWSTAR-Energy is an extended version of the FLOWSTAR model of flow over complex terrain and provides a range of useful outputs for wind farm planning, including estimates of free stream, gross and net energy yields. The model simulates individual wind turbine wakes and includes their effects in flow field output and wind energy predictions.

The FLOWSTAR-Energy map view can be used to visualise, add and edit wind turbines and output points and to view topographical data and model output including contour plots. FLOWSTAR-Energy links to Golden Software's Surfer software, a third party contour plotting package which can be used as an alternative to FLOWSTAR-Energy's own contour plotting.

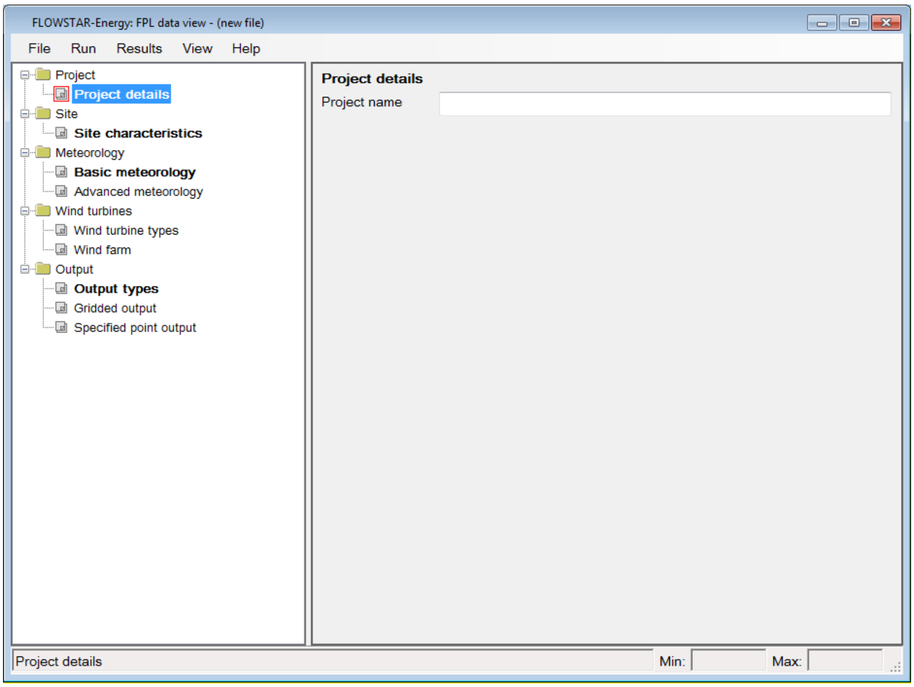



**Figure 1.1** – The FLOWSTAR-Energy **Data view** window.

### What are the main applications of FLOWSTAR-Energy?

Applications of the model include:

(a) Predictions of the wind field and wind turbine energy yields for wind farm planning.

(b) Wind engineering – for example flow in the proximity of a proposed bridge or flyover.

(c) Forestry – assessing where forests are most exposed to destructive winds.

(d) Pollution dispersion – FLOWSTAR is used in ADMS, CERC's air dispersion model, for calculating plume trajectory and spread in complex terrain.

### Model Theory

The FLOWSTAR-Energy model of wind turbine wake development exploits similarities between the decay of the wake behind a wind turbine (characterised by a region of reduced wind speed) and the dispersion of a plume of passive gas emitted from an elevated source.

FLOWSTAR-Energy is based upon several modules that are also used in ADMS, CERC's air dispersion model; in particular FLOWSTAR is used in ADMS for calculating plume trajectory and spread in complex terrain.

The description of the atmospheric boundary layer in FLOWSTAR is that of 'new generation' models, i.e. the atmospheric boundary layer is described not in terms of the single parameter Pasquill Class but in terms of two parameters, namely the boundary layer depth, $h$, and the Monin-Obukhov length, $L_{MO}$.

FLOWSTAR is derived from the theoretical work of Jackson and Hunt (1975), and Hunt *et al.* (1981 and 1988b). The model is based on the premise that different processes dominate the flow dynamics in layers at different heights above the ground; thus in the inner layer shear stress perturbations are locally important and are described by a mixing length closure whilst the flow is also impacted upon by pressure gradients. These can be determined from the outer layer flow where stratification plays an important role but where shear stress perturbations have little influence. There is a transitional or middle layer between the inner and outer layer.

Validation of FLOWSTAR (e.g. Carruthers *et al.* 1982) shows that it models the flow well typically for slopes up to 1 in 2 (upwind slopes and hill summits) and up to 1 in 3 locally in hill wakes. The model is also valid for larger slopes, though it may locally underestimate flow perturbations. Spatial scales treated by the model range from tens of metres up to several kilometres.

## 1.2 Model features

FLOWSTAR-Energy calculates wind speed, turbulence and wind energy at required locations for both onshore and offshore sites from input topographical, meteorological, surface roughness and wind turbine data. Results can be obtained for individual meteorological conditions or as a summary over all modelled meteorological conditions.

A meteorological pre-processor developed by the UK Met Office is part of the model and calculates values of the meteorological parameters in the atmospheric boundary layer required by FLOWSTAR-Energy from the input meteorological data.

As well as a map view for graphical data input and visualisation, FLOWSTAR-Energy includes a number of other helpful features:

- Meteorological data can be input in one of two widely-used formats: CERC's *.met* file format (sequential or statistical data) or *.tab* file format (statistical data);
- A wind rose plotter is included for easy visualisation of input wind speed and direction data;
- The model can run in verification mode to check model input and warn of user input errors before full calculations are performed.

## 1.3 About this User Guide

This *FLOWSTAR-Energy User Guide* is both a manual and a technical summary of the model.

### *Conventions*

To make this manual simpler to use, certain conventions have been followed with regard to layout and style.

- FLOWSTAR-Energy interface controls are shown in **Arial** font, e.g. the **Wind farm** screen, click on the **Plot** button.
- Keyboard keys are shown in **bold**, e.g. press **Enter**.
- Directory and file names are shown in *italics*, e.g. *oneday.met*, *<install_path>\Data*.
- Tips and other notes are shown thus:

---

*Think about the area you want to include in the calculation before specifying the output grid.*

---

- Table and figure references are shown in **bold**, e.g. see **Table 3.2**, **Figure 2.1**.

# SECTION 2 Getting Started

In this section the system requirements and the installation of FLOWSTAR-Energy are discussed in Sections 2.1 and 2.2 respectively. FLOWSTAR-Energy consists of two main windows or 'views': the **FPL data view**, introduced in Section 2.3; and the **map view**, introduced in Section 2.4. Section 2.5 gives a brief overview of setting up and running the model.

## 2.1 System requirements

FLOWSTAR-Energy will run in Windows 7, Windows 8 and Windows 10 environments. The minimum memory specification is 1 GB of RAM and 10 GB of hard disk space.

However, it is recommended that the highest specification machine available should be used for running simulations. Higher processor speeds and larger amounts of RAM will be particularly beneficial. As FLOWSTAR-Energy does not currently take advantage of multiple processing capabilities, there is no benefit in using a multi-processor PC for a single run. However, multiple runs can be run simultaneously on a single, multi-processor PC, providing the computer has sufficient RAM.

## 2.2 Installation

The installation of FLOWSTAR-Energy is straightforward. It uses an Installation Wizard, which guides the user through a short series of screens, collecting information on the user and installation parameters, before installing the software.

---

*Please check with your own IT personnel for company procedures for installing software.*

---

### 2.2.1 Installing FLOWSTAR-Energy

The following steps lead you through the FLOWSTAR-Energy installation process.

**Step 1** Log on as Local Administrator for the PC.

**Step 2** FLOWSTAR-Energy will either have been supplied by download link or on CD. Follow the appropriate instructions:

**Download:** Unzip the downloaded *.zip* file to a local directory. In Explorer, browse to this directory and double-click on the file '*setup.exe*'.

**CD:** Insert the installation CD and the install program should automatically start. If it does not, browse to locate the CD in Explorer and double-click on the file '*setup.exe*'.

In both cases, the screen shown in **Figure 2.1** will be launched.

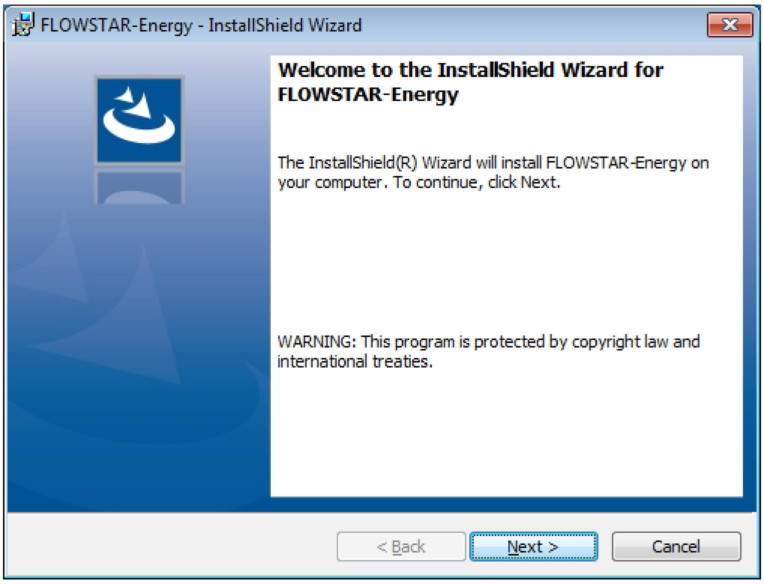



**Figure 2.1** – The FLOWSTAR-Energy installation **Welcome** screen.

**Step 3** Click **Next >** on the **Welcome** screen and then select **I accept the terms of the licence agreement**, and click **Next >** in the **Licence Agreement** screen if you accept the licence terms. The **Customer Information** screen is then displayed, as shown in **Figure 2.2**. If you do not accept the licence terms select **I do not accept the terms of the licence agreement** and click **Next >** to cancel the installation process.

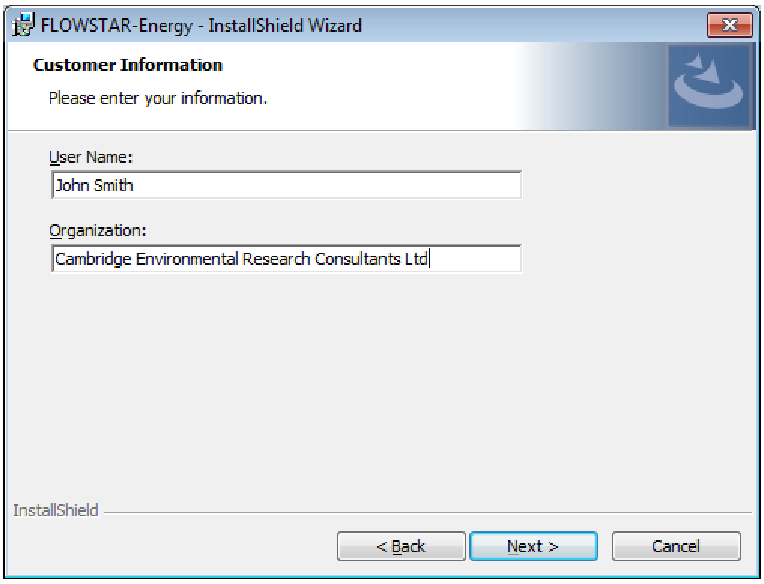



**Figure 2.2** – The FLOWSTAR-Energy **Customer Information** screen.

**Step 4** Enter your user name and organisation in the designated places. Click **Next >** to proceed to the **Destination Folder** screen, as shown in **Figure 2.3**.

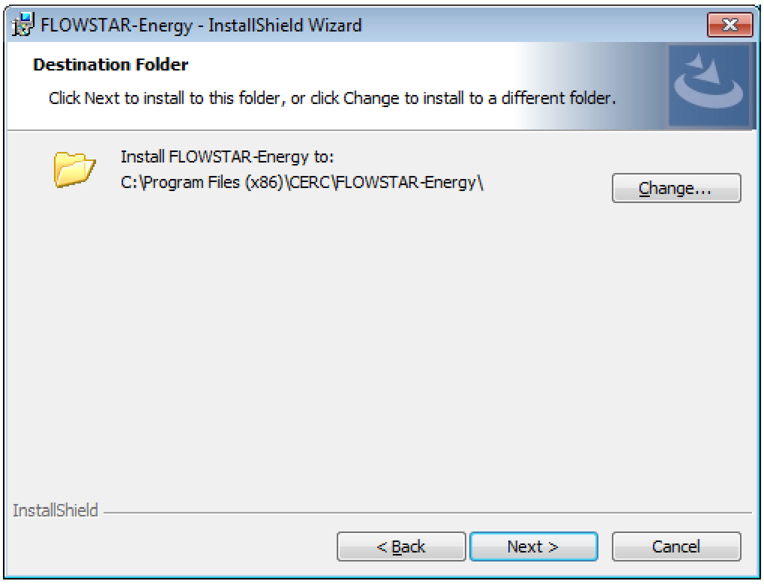



**Figure 2.3 –** The FLOWSTAR-Energy **Destination Folder** screen.

**Step 5** You should select a drive with at least 1 GB of available disk space. The default installation directory is *C:\Program Files (x86)\CERC\FLOWSTAR-Energy*. If required, use the **Change...** button to select your own installation directory. Click **OK** to return to the **Destination Folder** screen.

---

*The abbreviation* <install_path> *will be used in the rest of the User Guide to denote the installation directory you have chosen, for example* C:\Program Files (x86)\CERC\ FLOWSTAR-Energy.

---

Click **Next >** to choose who should be able to use FLOWSTAR-Energy, as shown in **Figure 2.4**.

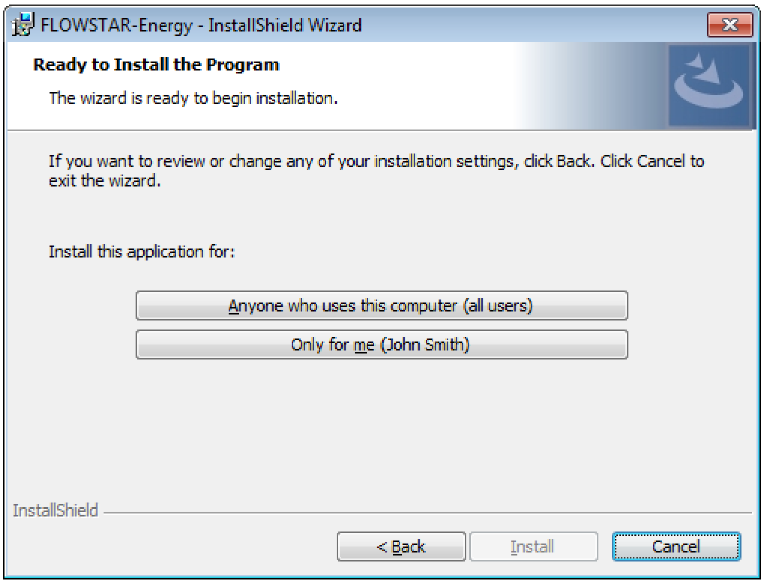



**Figure 2.4 –** The FLOWSTAR-Energy **Ready to Install** screen.

**Step 6** Chose whether to install for all users or just the current user.

*If the local administrator profile is not the usual user profile, choose to install for all users.*

If you wish to amend any details, press the **< Back** and **Next >** buttons as appropriate. Once the FLOWSTAR-Energy files have been successfully installed, the final screen will appear, as shown in **Figure 2.5**.

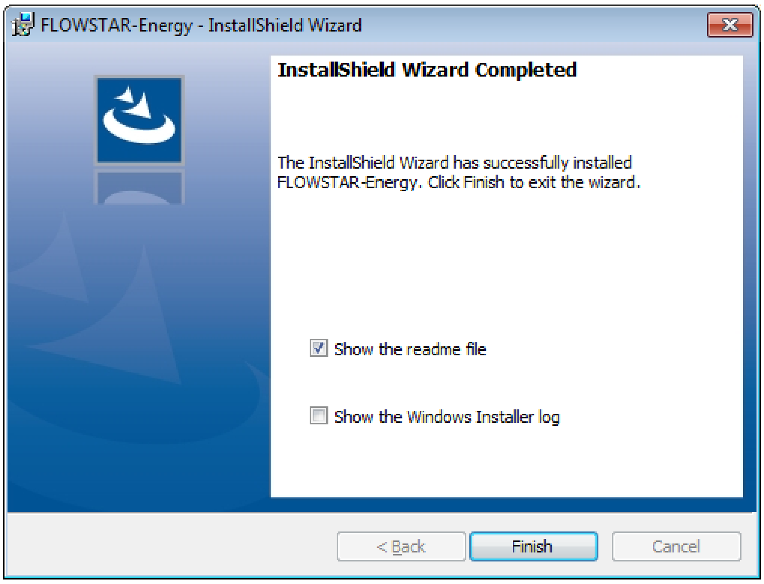



**Figure 2.5** – The FLOWSTAR-Energy **InstallShield Wizard Completed** screen.

**Step 7** Click **Finish** to close the installation screen.

You have been provided with a unique licence file by email, which is required in order to run the model. It is important that you install this new licence file as instructed.

**Step 8** To install the FLOWSTAR-Energy licence, copy the file *FLOWSTAR.lic* to the <*install_path*> directory.

**Step 9** Restart your computer: you are now ready to use the model. The installation procedure automatically puts a shortcut to FLOWSTAR-Energy on your Windows desktop.

## 2.3 FLOWSTAR-Energy: FPL data view

This section gives an overview of the functions of the **FLOWSTAR-Energy: FPL data view** window. For more information about each component, with advice on suitable inputs and settings, please refer to Section 2.3.4 for menu items and Section 3 for modelling parameters.

### 2.3.1 Mouse buttons

Unless otherwise stated, mouse instructions refer to the left button. If the mouse options have been used to reverse the mapping (e.g. because you are left-handed), the right mouse button should be used instead.

### 2.3.2 Keyboard access

Most of the mouse instructions in this manual can be reproduced using keystrokes. A brief guide to these keystrokes is given in **Table 2.1**.

Also known as shortcut keys, there are combinations of keys that perform some of the main commands. For example, menu commands that have one letter underlined are accessible by holding down the **ALT** key and then typing the underlined letter. For example, the menu command **Open...** located on the **File** menu, may be executed by typing **ALT + F** and then **ALT + O.** Some frequently-used menu commands can be executed by a simpler combination. For example the **File Open...** menu command can also be executed by typing **Ctrl + O**.

| Key | Description |
|---|---|
| ***Moving the cursor between data entry boxes*** | |
| **TAB** | Move the cursor forwards through data entry boxes or buttons |
| **SHIFT + TAB** | Move the cursor backwards through data entry boxes and buttons |
| **RETURN** | ‘Enter’ or accept the current data page or execute the action of a highlighted button |
| **SPACEBAR** | Select or deselect the highlighted option |
| ***Entering data in a box*** | |
| **DELETE** | Delete the character immediately to the right of the cursor |
| **BACKSPACE** | Delete the character immediately to the left of the cursor |
| **← arrow** | Move the cursor one space to the left in the current box |
| **→ arrow** | Move the cursor one space to the right in the current box |
| **SHIFT + arrow** | Begin highlighting characters in the direction of the arrow (see above) |
| ***Highlighted text*** | |
| **DELETE** | Delete all highlighted characters |
| **(Type)** | Typing text replaces the highlighted text with new text |

**Table 2.1 –** Keystrokes to enable you to move through the **FPL data view** screen.

### 2.3.3 FPL data view buttons

There are several buttons throughout the **FPL data view** window. The buttons and their actions are given in **Table 2.2**.

| Button | Description |
|---|---|
|  | Browse for a particular file |
|  | View or edit the input data in the **map view** |
|  | View the input file in the application set under preferences |
|  | View wind rose |
|  | Full screen editing |

**Table 2.2** – Buttons available in the **FPL data view** screen.

### 2.3.4 Main menu options

The **FPL data view** menu bar has five headings: **File**, **Run**, **Results**, **View** and **Help**. All menu headings have drop-down lists of options (see for example the **File** menu options shown in **Figure 2.6**). **Table 2.3** gives the list of options and roles of each menu item.

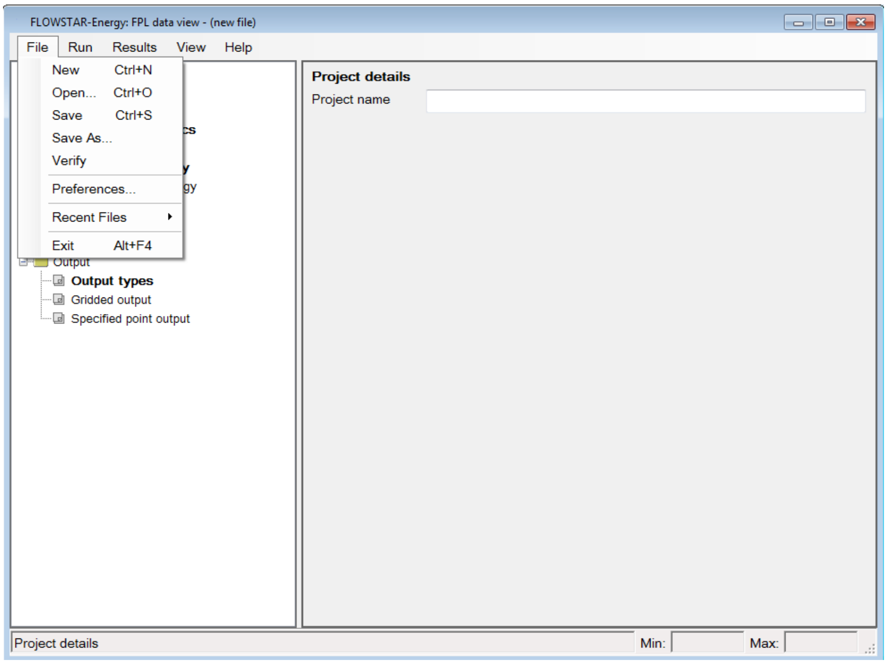



**Figure 2.6** – The **File** menu from the menu bar in the **FPL data view** window.

| **Menu** | | **Role** |
|---|---|---|
| **File** | **New** | Reset the parameters in the *.fpl* file to their default values |
| | **Open…** | Open a previously saved parameter file |
| | **Save** | Save the current parameters under the current file name |
| | **Save As…** | Save the current parameters with a user-specified file name |
| | **Verify** | Run verification checks on the *.fpl* file |
| | **Preferences…** | • **Viewing** tab: choose the application for viewing numerical input files or results files. The default application is Notepad.<br>• **Template** tab: A previously saved *.fpl* file can be specified as the template, to be loaded on selecting **File New** |
| | **Recent Files** | On holding the mouse over **Recent Files**, a list of recently-used files is shown, and can be selected for opening |
| | **Exit** | Quit FLOWSTAR-Energy |
| **Run** | **Model** | Run the FLOWSTAR-Energy model using the current *.fpl* file |
| **Results** | **Plot in map view…** | Opens the **map view** window so that the **map view** toolbar buttons can be used to plot model output. |
| | **Plot in Surfer…** | Launch the 2D plotting facility (if Surfer is installed) |
| | **View short-term flow fields at specified points** | View the relevant numerical output from the current *.fpl* file. The menu option will be grey (disabled) if there is no numerical output of the relevant type. The viewing application can be chosen under **File Preferences… Viewing** |
| | **View short-term potential wind power at specified points** | |
| | **View short-term wind farm results** | |
| | **View short-term free stream boundary layer profiles** | |
| | **View long-term flow field at specified points** | |
| | **View long-term potential wind energy at specified points** | |
| | **View long-term wind farm results** | |
| | **View processed met data** | View processed met data from the current *.fpl* file. The viewing application can be chosen under **Preferences … Viewing** |
| | **Plot wind rose of processed met data** | Launch the wind rose viewer with the processed met data from the current *.fpl* file |
| | **View report** | View the report file for the current *.fpl* file |
| | **View log** | View the log file for the current *.fpl* file |
| | **View warnings** | View any warnings for the current *.fpl* file |
| | **View errors** | View any errors for the current *.fpl* file |
| | **Open results folder** | Opens Microsoft Windows Explorer at the directory containing the current *.fpl* file |
| **View** | **Open map view…** | Open the **map view** |
| **Help** | **User guide** | Open the User Guide in a PDF viewer |
| | **Email CERC** | Opens the user's default email client and auto-addresses a new email to the FLOWSTAR-Energy helpdesk |
| | **About FLOWSTAR-Energy** | Show product version number |

**Table 2.3** – Options and roles of menu items from the **FPL data view** screen

## 2.4 FLOWSTAR-Energy: map view

The **FLOWSTAR-Energy: map view** window is an interactive map which can be used to edit and view model input data and plot model results.

### 2.4.1 Map view menu options

The **map view** menu bar has three headings: **File**, **Edit** and **Help**. All menu headings have drop-down lists of options (see for example the **File** menu options shown in **Figure 2.7**). **Table 2.4** gives the list of options and roles of each menu item.

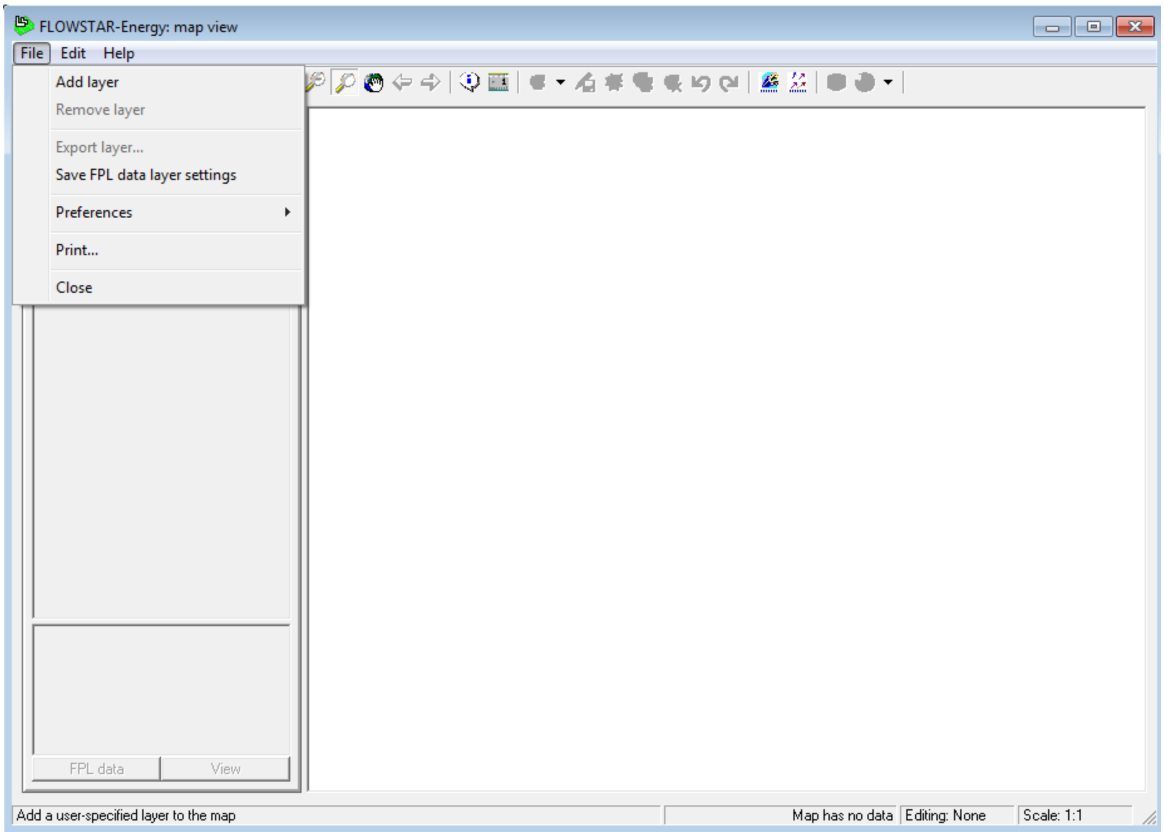



**Figure 2.7** – The **File** menu from the menu bar in the **map view**.

<table>
<tr><th colspan="3">Menu</th><th>Role</th></tr>
<tr><td rowspan="8">File</td><td colspan="2">Add layer</td><td>Allows a layer to be added, e.g. a background map.</td></tr>
<tr><td colspan="2">Remove layer</td><td>Removes a layer from the map view.</td></tr>
<tr><td colspan="2">Export layer...</td><td>Allows the selected layer to be exported in various formats.</td></tr>
<tr><td colspan="2">Save FPL data layer settings</td><td>Saves the current appearance of the FPL data layers (colour, symbol etc.) as the default appearance.</td></tr>
<tr><td>Preferences</td><td>Viewing options</td><td>Allows option to show/hide the north arrow.</td></tr>
<tr><td>Preferences</td><td>Gridding option</td><td>Allows selection of gridding method.</td></tr>
<tr><td colspan="2">Print…</td><td>Brings up a print preview showing a printable version of the map.</td></tr>
<tr><td colspan="2">Close</td><td>Closes the FLOWSTAR-Energy map view.</td></tr>
<tr><td rowspan="6">Edit</td><td colspan="2">Save edits</td><td>Saves the edits made to the model.</td></tr>
<tr><td colspan="2">Copy map to clipboard</td><td>Copies the current view in the map window to the clipboard.</td></tr>
<tr><td colspan="2">Copy legend to clipboard</td><td>Copies the Legend tab layer panel to the clipboard.</td></tr>
<tr><td colspan="2">Copy scalebar to clipboard</td><td>Copies the scalebar to the clipboard.</td></tr>
<tr><td colspan="2">Use the FPL data coordinate system</td><td>Sets the coordinate system used in the map view to the same as that defined in the FPL data view: i.e. Cartesian non-specified coordinate system.</td></tr>
<tr><td colspan="2">Set map coordinate system</td><td>This allows you to set the coordinate system to a projected or a geographic system, or to turn off the coordinate system.</td></tr>
<tr><td rowspan="2">Help</td><td colspan="2">User guide</td><td>Opens the FLOWSTAR-Energy User Guide.</td></tr>
<tr><td colspan="2">About FLOWSTAR-Energy: map view</td><td>Displays information about FLOWSTAR-Energy: map view, e.g. version number and CERC contact details.</td></tr>
</table>

**Table 2.4** − Options and roles of menu items from the **map view**.

### 2.4.2 Map view toolbar buttons

The **map view** toolbar contains buttons that allow layers and features in the FLOWSTAR-Energy layers, and the **map view** to be edited, as well as providing access to the 2D output plotter. The purpose of each of the buttons is summarised in **Table 2.5** and **Table 2.6**.

As the cursor is moved across the toolbar, explanations appear in the status bar at the bottom of the **map view** window explaining the function of each button. When using a tool that requires the user to specify a location in the **map view** window, the appearance of the cursor will change.

| Button | Name | Cursor | Function |
|---|---|---|---|
| | **Export layer** | N/A | Export the current layer to a file. |
| | **Add layer** | N/A | Add a new layer from a file. |
| | **Add WMS layer** | N/A | Add a Web Map Service (WMS) layer. |
| | **Remove layer** | N/A | Remove the selected layer from the layer panel and **map view**. This is only available for user created layers. |
| | **Refresh layers** | N/A | Obtain the latest data from the **FPL data view** for all the layers and update the **map view**. |
| | **Refresh layer** | N/A | Obtain the latest data from the **FPL data view** for the selected layer and updates the **map view**. |
| | **Zoom to layers** | N/A | Set the **map view** to show all the data in all the layers. |
| | **Zoom to layer** | N/A | Set the **map view** to the extent of the selected layer. |
| | **Zoom in** | N/A | Zoom in. |
| | **Zoom out** | N/A | Zoom out. |
| | **Zoom** | | Zoom to display an area defined by the user by clicking and dragging the cursor. |
| | **Pan** | | Move the **map view** without altering the scale. |
| | **Previous extent** | N/A | Return to the previously displayed extent in the **map view**. |
| | **Next extent** | N/A | Go to the next extent in the **map view** window. |
| | **Information** | | View information about a model feature in the **map view** window. |
| | **Measure** | | Measure the distance between two or more points in the **map view** window. |
| | **Add feature** | | Add a feature to the selected layer, e.g. a wind turbine or a specified point. |
| | **Save edits** | N/A | Save the edits made during the current session to the **FPL data view**. |
| | **Edit feature** | | Edit the geometry of a feature in the selected layer. |
| | **Shift feature** | | Move a feature in the selected layer. |
| | **Delete feature** | N/A | Delete the selected feature. |
| | **Undo** | N/A | Undo a previous action while editing. |
| | **Redo** | N/A | Redo an action while editing. |
| | **Wind Energy** | N/A | Launch the Contour Plotter for wind energy plots. |
| | **Flow field** | N/A | Launch the Output Plotter for flow field plots. |
| | **Change view** | N/A | This icon is shown when the current view is 2D and changes the view to a 3D visualisation of the input. When the display is in 3D mode additional buttons are available as shown in **Table 2.6**. |

**Table 2.5** – Map view toolbar buttons

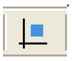
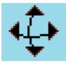
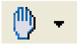
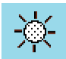
| Button | Name | Cursor | Function |
|---|---|---|---|
|  | **Change view** | N/A | This icon is shown when the current view is 3D and changes the view to a 2D visualisation of the input. |
|  | **Camera position** |  | Changes the camera pan mode so that the camera position is changed by yaw and pitch rotations. Use the drop down arrow to change the mode. |
|  | **Camera XYZ** |  | Changes the camera pan mode so that the camera position is changed by vertical translations and left-to-right horizontal translations. |
|  | **Camera XY** |  | Changes the camera pan mode so that the camera position is changed by left-to-right and forward-and-backward translations in the horizontal plane. |
|  | **Camera rotation** |  | Changes the camera pan mode so that the camera position is changed by point-of-view rotation. |
|  | **Sun position** |  | Changes the light and shade of the display. |
|  | **Zoom** |  | Zoom in and out. |

**Table 2.6** – Map view toolbar buttons for 3D display

Some of the buttons are only available under certain conditions; for instance, the **Add Feature**, **Edit Feature** and **Shift Feature** buttons are only available if the current selected layer can be edited from within the **map view**. If a button is not currently available it will be greyed out.

### 2.4.3 The layer panel

There are two tabs in the layer panel: **Legend** and **3D**. The former shows the data layers and the latter shows the additional options for manipulating the 3D views.

The **Legend** shows the **FPL data view** layers along with any user defined layers, such as contour plots or background images. The layers that are present when the **map view** is first opened are the **FPL data view** layers. These layers contain all the turbines and output locations for the *.fpl* currently open in the **FPL data view**. To update the display to reflect any changes that have been made in the **FPL data view**, e.g. through opening a new *.fpl*, or adding a new wind turbine, click on the **Refresh All Layers** button on the toolbar. 

An information panel can also be displayed beneath the layer panel. The information panel appears when you select the **Information** tool and click on a feature; refer to Section 5.6 for full details. The splitting and the sizes of the information panel and layer panel can be adjusted by clicking on the horizontal splitter bar and holding and dragging. Similarly, the vertical splitter bar between the whole layer panel and the map panel can also be adjusted.

For each of the layers that can contain multiple features the **Legend** shows the name of the layer, the number of features in that layer (e.g. the number of wind turbines) and the symbol used to represent that layer. The user can change the symbol used to represent a layer and details of how to do this are given in Section 5.1. The **Output grid extent** layer only shows the name of the layer and the symbol used to represent the layer.

User-defined layers can be contours generated using the **Wind Energy** button, vector or flow field plots generated using the **Flow field** button, or web map service or background images added using the **Add Layer** button. Any user-defined layers can be removed by first selecting the layer and then clicking on the **Remove Layer** button; alternatively they can be removed by selecting the layer in the layer panel, right-clicking on the layer and selecting **Remove layer** from the menu list. Details about creating wind energy contour layers can be found in Section 4 and instructions for adding a background map as a layer can be found in Section 2.5.2.

## 2.5 Creating a model file and running the model

To generate results using FLOWSTAR-Energy, there are several steps to complete:

- create a new model file,
- enter data to define the problem,
- save the model file,
- verify the model file,
- run the model,
- display the output.

The first four of these steps are described in Sections 2.5.1 to 2.5.5. Entering model data is described in general terms here, and in full detail in Section 3. Displaying model output is covered in detail in Section 4.

### 2.5.1 Creating a model file

When FLOWSTAR-Energy is opened or when you select the **New** command from the **FPL data view File** menu, a new model file, or scenario, is created and default values are loaded into the screens for you to edit.

To open an existing model file for editing or running, choose **Open...** from the **File** menu. FLOWSTAR-Energy input files have the extension *.fpl*.

### 2.5.2 Entering information

***FPL data view window layout***

The **FPL data view** window has two main panels. The left hand side of the window shows a series of folders, each folder containing one or more screens. Clicking on a screen name in the left hand side takes you to the screen on the right hand side. The icon for the current screen is indicated with a red outline in the left hand side, as shown in **Figure 2.8**. Some screens must be completed for every *.fpl* file, and some are optional. Optional screens can be enabled/disabled by checking/unchecking the box at the top of the screen or by double-clicking the screen name in the left hand side. Screen names are highlighted in bold in the left hand side if they are enabled.

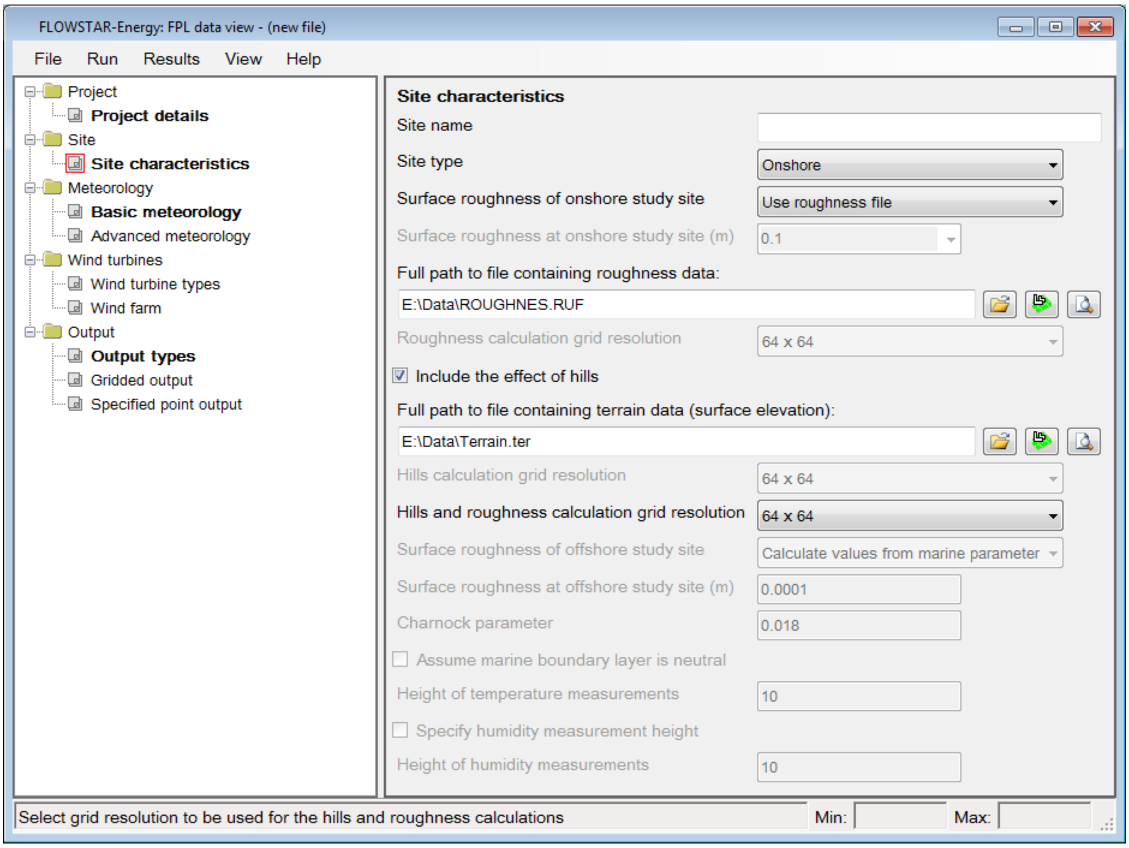



**Figure 2.8** – The **FPL data view** window with the **Site characteristics** screen of the **Site** section selected for editing.

### *Changing values in the input screens*

To change a parameter value in an input screen, move the pointer until it is over the appropriate text box and click. Alternatively, use the **TAB** or **arrow keys** to move systematically through the sections contained in each screen. The selected area will be highlighted. Now type the new value, which will automatically replace what was highlighted. Alternatively, use **DELETE** and/or **BACKSPACE** to remove unwanted characters before typing in the new value.

---

*Note that a blank cell does not denote a value of zero.*

---

### *The helpline*

This is a single line of text that appears at the bottom of the active screen. The information in the helpline changes when different controls on the screen are selected. It gives a brief description of the selected control's function. Where you are prompted for a numerical value, the helpline will give the minimum and maximum values allowed.

For example, when the **Latitude (°)** box in the **Meteorology** screen is selected, the help bar will display the line:

*Approximate latitude of the site (°), Min: -90, Max: 90*

### *Data validity and integrity checking*

As you enter data, the **FPL data view** window performs checks to ensure that all user-entered data are consistent with the model's logic and that minimum and maximum values are satisfied.

### *The map view window layout*

**Figure 2.9** shows the **map view** window with the main features labelled. The options available from the menu are described in Section 2.4.1, the buttons available on the toolbar are discussed in Section 2.4.2, and the **Legend** is discussed in Section 2.4.3.

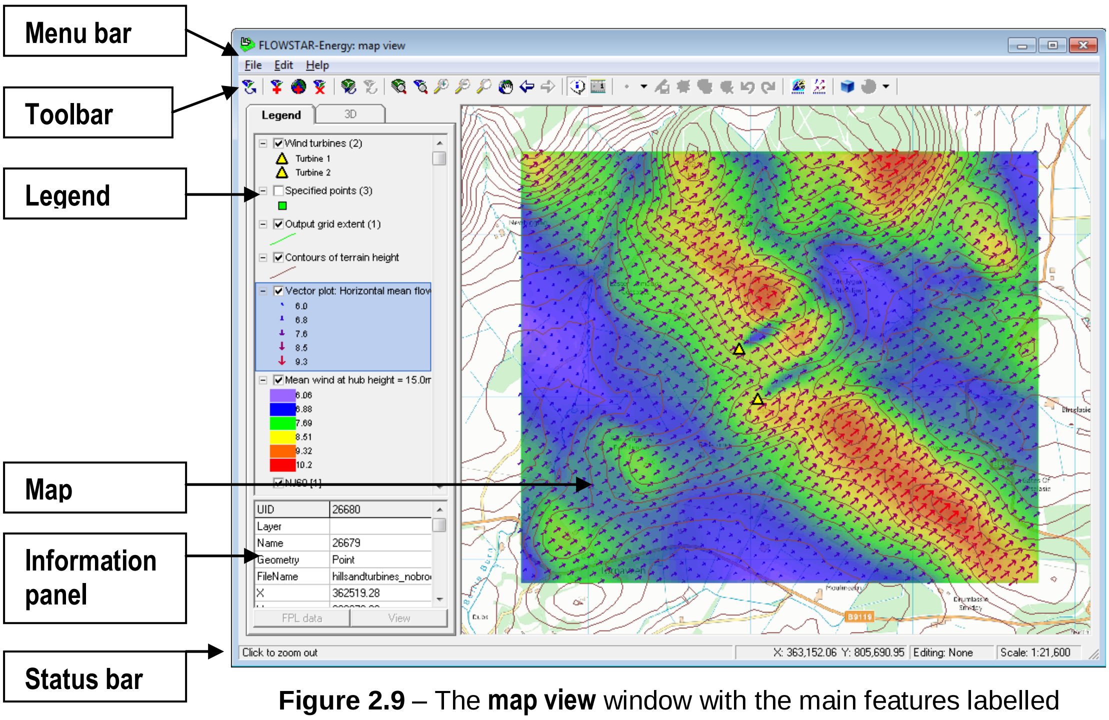



**Figure 2.9** – The **map view** window with the main features labelled

### *Displaying background maps*

Background images in a wide variety of image file formats can be added to the **map view** window, to aid in visualising and manipulating the model input and output. This is done using the **Add Layer** button on the toolbar.

To load an image file as a background map, the image must be correctly georeferenced, which can be done using world files or MapInfo *.tab* files – not to be confused with the observed climate meteorological data files which also use the file extension *.tab*. For British OS Map data, world files are available from the Ordnance Survey website. These instructions below explain how to load an OS map tile as a background map using the corresponding world file to georeference it.

**Step 1** Ensure the map tile image file (*.tif*) and world file (*.tfw*) have the same name and are in the same directory.

**Step 2** You may wish to set a coordinate system for the background map as described in Section 5.3.1. This may be useful if you are using multiple coordinate systems in the **map view**.

**Step 3** In the **map view** click on the **Add Layer** button on the toolbar.

**Step 4** The **Add Layer** screen is then displayed, as shown in **Figure 2.10**. Browse to find the *.tif* file for the map tile, then click on the **Open** button.

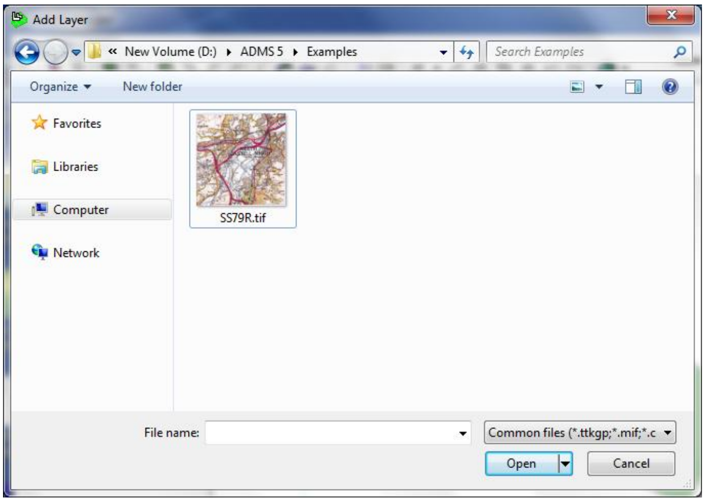



**Figure 2.10** – The **Add Layer** screen

### 2.5.3 Saving input data to a model file

When you are ready to run the model, choose **Save** from the **FPL data view File** menu. If the current scenario has not been saved before, you will be prompted to choose a directory and file name. FLOWSTAR-Energy model files are always saved with the extension *.fpl*.

### 2.5.4 Additional model input files

There are a number of additional model input files that may be required by the model in order to run. These are most often comma-delimited text files in a particular format. The most common example of a file used by the model is the meteorological data file required by the FLOWSTAR-Energy met pre-processor, usually with the file extension *.tab* or *.met*. Other model input files include the *.wtd* file that contains power and thrust data for wind turbines, the *.ter* and *.ruf* files that contain variable terrain height and surface roughness values respectively, and the *.prf* file that contains vertical profile data. Please refer to Section 3 for details about these additional model input files.

The user can browse to locate these additional model input files associated with a model run, and the absolute paths of the files are automatically written to the FLOWSTAR-Energy model *.fpl* file.

---

*If you use Excel to save the additional model input files in comma-delimited format, it is important to open them in a text editor such as Notepad, remove any trailing commas, and re-save them, prior to use by the model.*

---

### 2.5.5 Verifying the model file

An option exists to verify the contents of an *.fpl* file. This carries out all of the model data checks, runs the meteorological pre-processor, and produces a report file, but stops before calculating flow fields or potential wind energy outputs. It is therefore quick to run, providing a useful check of the modelling input data before the main run.

While running *.fpl* verification, the model will carry on past as many errors as it can so as to produce as comprehensive a list of errors as possible. Select **File**, **Verify** from the **FPL data view** menu bar to run *.fpl* verification. Once verification is complete the output can be viewed using the **FPL data view Results** menu. Select **Results**, **View report** to view the verification report. Select **Results**, **View errors** to view a list of any errors found during verification. This menu option will be disabled, with a grey appearance, if no errors were found. Similarly select **Results**, **View warnings** to display a list of warnings found. This menu option will be disabled if no warnings were found.

### 2.5.6 Running FLOWSTAR-Energy

Save the current scenario as an *.fpl* file, then select **Run, Model** from the **FPL data view** menu bar to run the model. If you have never saved the current scenario or if you have changed anything in the **FPL data view** since the scenario was last saved, then you will be prompted to save the modifications.

While the model is running, information is displayed in a progress window (see **Figure 2.11**) to show the status of the model run in terms of the number of lines of meteorological data completed. When all the calculations have successfully completed, the progress window displays "`***RUN SUCCESSFULLY COMPLETED***`".

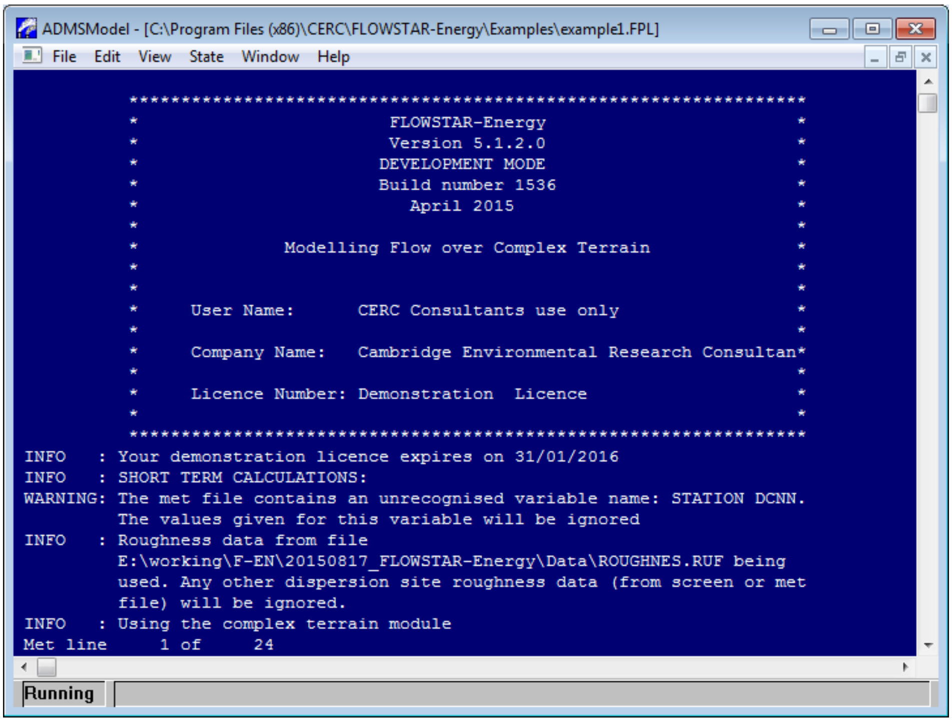



**Figure 2.11 −** Progress window of a FLOWSTAR-Energy run.

### 2.5.7 Displaying model output

Please refer to Section 6 for details about displaying output from FLOWSTAR-Energy model runs.

# SECTION 3 Model Input

Setting up a modelling problem in FLOWSTAR-Energy requires the user to input information specifying the site characteristics, meteorological conditions, wind turbines (if any) and the output required. This section provides a guide to providing this information through the **FPL data view** and, where relevant, the **map view**. The opening screen of the **FPL data view** window is shown in **Figure 3.1**.

Individual screens of the **FPL data view** can be opened from the folder structure on the left, while data and modelling options are entered on the right.

There are nine input screens under five distinct folders associated with a FLOWSTAR-Energy model run. The folders are:

- **Project**: project name,
- **Site**: site characteristics, including terrain and roughness details,
- **Meteorology**: meteorological conditions,
- **Wind Turbines**: wind turbine specifications and wind farm details,
- **Output**: output required from the calculations.

It is advisable to work through each screen of the **FPL data view** window in turn, starting at the top of the folder structure.

---

*Note that incomplete model files can be saved for later completion, or for use as templates.*

---

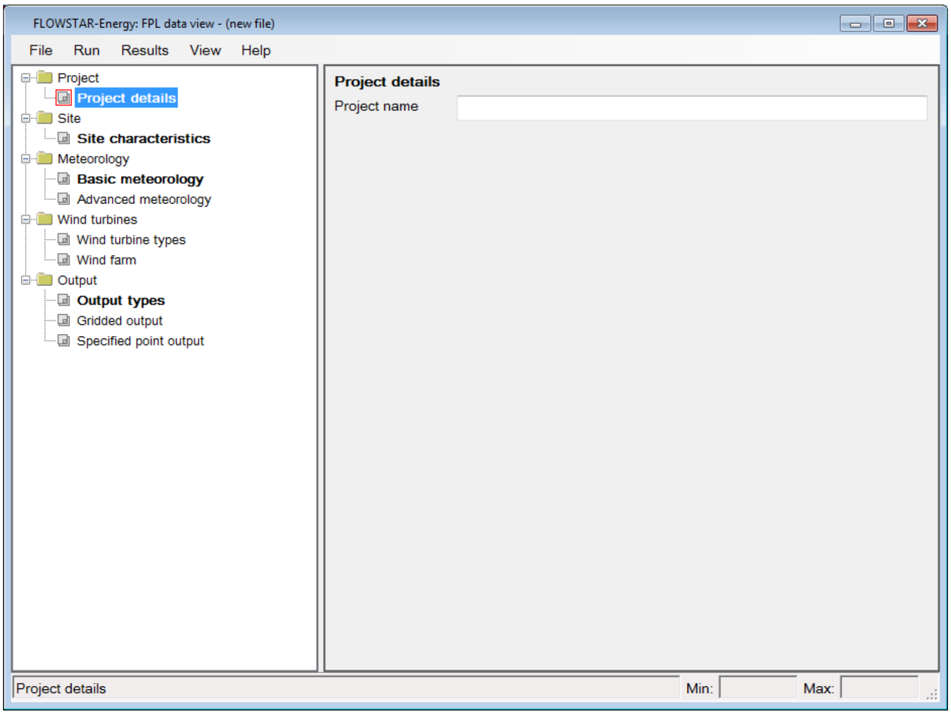



**Figure 3.1** – The FLOWSTAR-Energy: **FPL data view** window.

# 3.1 Project

There is only one screen in the Project folder; this is named **Project details** and is shown in **Figure 3.1**.

### 3.1.1 Project details

***Project name***

Up to 80 characters may be entered, or the box may be left empty. Text entered here will appear in the output "report" file, so it is recommended that meaningful names are included each time.

# 3.2 Site

There is only one screen in the Site folder; this is named **Site characteristics** and is shown in **Figure 3.2** as it appears for a new model run. All parameters describing the site characteristics are entered here: site type, surface roughness, terrain, etc.

### 3.2.1 Site characteristics

***Site name***

Up to 80 characters may be entered, or the box may be left empty. Text entered here will appear in the output "report" file, so it is recommended that meaningful names are included each time.

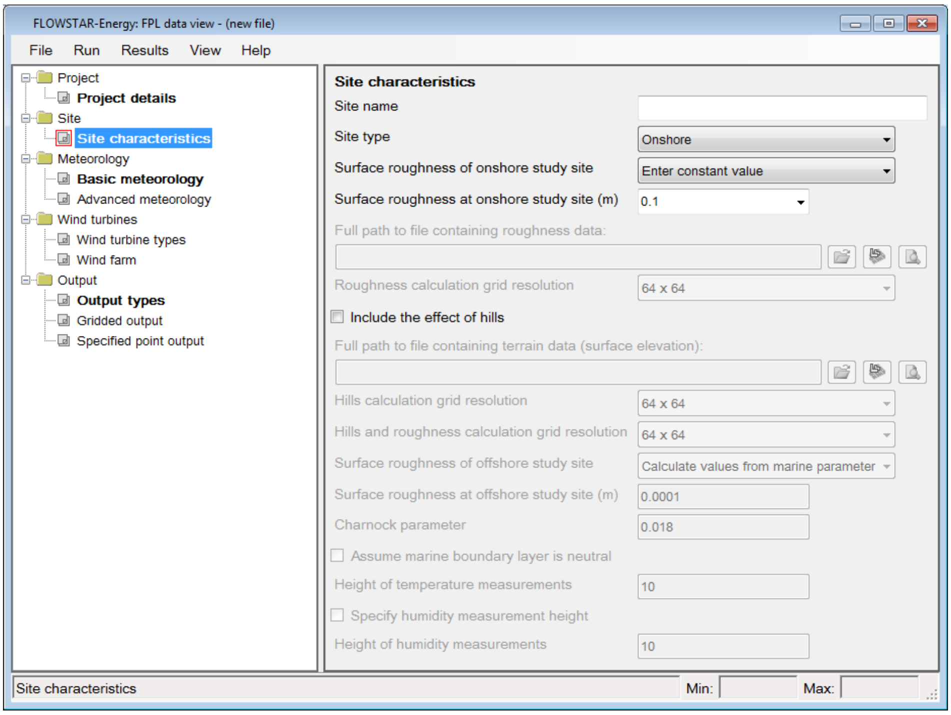



**Figure 3.2** – The **Site characteristics** screen.

### *Site type*

Two choices are available from a drop down list: **Onshore** (for a site on land) or **Offshore** (for a site at sea). Depending upon this selection various options become available below. When **Onshore** is selected the parameters for surface roughness and terrain become available. When **Offshore** is selected the marine parameters at the bottom of the screen are available to edit.

### *Surface roughness of onshore study site*

When modelling an onshore scenario, the user has three options for entering surface roughness; these are available from the drop down list: **Enter constant value**; **Use roughness file** or **Use values from .MET file.**

When **Enter constant value** is chosen from the drop down list, the option **Surface roughness at onshore study site (m)** below the drop down list becomes available. In the box, enter the surface roughness at the site (in metres) based on land use. Either enter a value directly in the box, subject to the model limits (minimum = $10^{-7}$ m, maximum = 10 m), or alternatively use the drop-down box to select a value based on the land use − **Table 3.1** shows the land use types and corresponding values of surface roughness. The default value of surface roughness is 0.1 m, corresponding to a region of root crops.

| Land use | Surface roughness (m) |
|---|---|
| Large urban areas | 1.5 |
| Cities, woodlands | 1 |
| Parkland, open suburbia | 0.5 |
| Agricultural areas (max) | 0.3 |
| Agricultural areas (min) | 0.2 |
| Root crops | 0.1 |
| Open grassland | 0.02 |
| Short grass | 0.005 |

**Table 3.1** − The surface roughness values for the different land uses available on the **Site Characteristics** screen.

When **Use roughness file** is chosen from the drop down list, the option **Full path to file containing roughness data:** becomes available to select a file containing spatially varying roughness data: the *.ruf* file. This file format is described below. The browse button can be used to select a file.

Once the path to the roughness file has been entered, the two buttons beside the browse button are activated. The map view button displays the roughness data file in the **map view**. The view button loads the roughness data file into the numerical data viewing application. The **Roughness calculation grid resolution** also needs to be selected when modelling this option. The available choices are shown in **Table 3.2**.

*For most calculations the 32 x 32 or 64 x 64 grids are sufficient.*

| Grid resolution | Notes |
|---|---|
| **256 x 256** | This option should only be used if the input data are highly resolved, and cover large or complicated areas; run times may be prohibitive. |
| **128 x 128** | |
| **64 x 64** | This is the default. |
| **32 x 32** | |
| **16 x 16** | For testing only |

**Table 3.2** – The choices of internal calculation grid resolution available on the **Site Characteristics** screen.

Note that the grid resolution here refers to the internal grid used to calculate the flow field and is different to that in the **Gridded output** screen, which refers to the grid of points on which output is given.

When **Use values from .MET file** is chosen from the drop down list, the user can enter hourly varying surface roughness values for the site via the meteorological data file, using the variable name '`Z0 (D)`'. This option can only be used with a meteorological data file in the *.met* format.

### *Include the effect of hills*

Check the **Include the effect of hills** box if the effect of terrain height is to be included in the model run. An additional data file is required which contains the spatially varying terrain height data: the *.ter* file. The **Full path to file containing terrain data (surface elevation):** can be entered in the box provided. The browse button  can be used to enter the full path to the file. The file format for a *.ter* file is described below.

Once the path to the file has been entered, the two buttons beside the browse button  are activated. The map view button  automatically loads the terrain data file into the **map view**. The view button  automatically loads the terrain data file into the numerical data viewing application.

The resolution of the hills calculation grid is then entered (see the description of grid resolution in **Table 3.2** in the surface roughness section above).

---

*Note that if the surface roughness uses a roughness file and the hills option is being used, the surface roughness resolution box is greyed out, and the grid resolution is selected for both hills and roughness in the same selection box, and the boxes to select either roughness or hills calculation grid resolution individually become greyed out.*

---

### ***File format of the terrain height (.*ter*) and surface roughness (.*ruf*) files***

The file format for spatially varying roughness (*.ruf*) is identical to the file format for spatially varying terrain height (*.ter*).

These files should consist of a series of lines with

```
N, X, Y, Z
```

entered on each line, where `N` is an incrementing counter for each line, `X` is the X coordinate of the data point, `Y` is the Y coordinate of the data point, and `Z` is:

- the height of the terrain (in metres) at that data point for the *.ter* file (see example in **Figure 3.3**), and
- the surface roughness value (in metres) at that data point for the *.ruf* file.

---

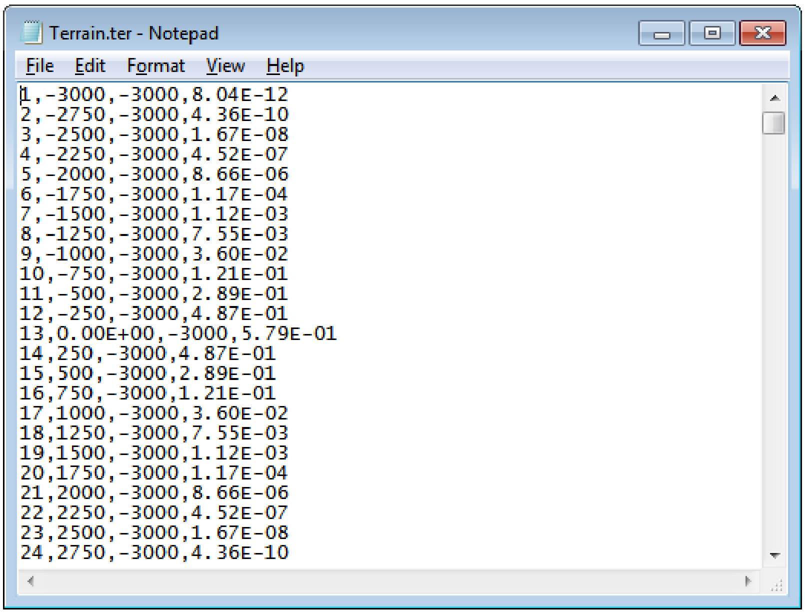
*The* .ter/.ruf *files should be comma-separated and the maximum number of data points that may be included in each file is 66,000.*

---

The data points do not need to be regularly spaced, but the area described should be rectangular. Note that the coordinate system used should be the same as that in which the turbine locations, output points, etc., are defined. An example *Terrain.ter* file describing a bell-shaped hill is supplied in *<install_path>\Data* and shown in **Figure 3.3**. An example *Roughnes.ruf* file is also supplied in the *<install_path>\Data*.

The example files may be edited to create new *.ter* or *.ruf* files using a spreadsheet program or text editor. The CERC website has information about obtaining terrain data from third party suppliers and converting to *.ter* format. Visit www.cerc.co.uk/UserArea for this information for the UK, France, Northern Ireland, Republic of Ireland and the US.

An easy way to create a roughness file is to edit the Z values of the terrain file being used so that the same domain, with the same X and Y coordinates, is used for both files. This can be carried out using a spreadsheet package such as Excel.




```
1,-3000,-3000,8.04E-12
2,-2750,-3000,4.36E-10
3,-2500,-3000,1.67E-08
4,-2250,-3000,4.52E-07
5,-2000,-3000,8.66E-06
6,-1750,-3000,1.17E-04
7,-1500,-3000,1.12E-03
8,-1250,-3000,7.55E-03
9,-1000,-3000,3.60E-02
10,-750,-3000,1.21E-01
11,-500,-3000,2.89E-01
12,-250,-3000,4.87E-01
13,0.00E+00,-3000,5.79E-01
14,250,-3000,4.87E-01
15,500,-3000,2.89E-01
16,750,-3000,1.21E-01
17,1000,-3000,3.60E-02
18,1250,-3000,7.55E-03
19,1500,-3000,1.12E-03
20,1750,-3000,1.17E-04
21,2000,-3000,8.66E-06
22,2250,-3000,4.52E-07
23,2500,-3000,1.67E-08
24,2750,-3000,4.36E-10
```

**Figure 3.3** – The start of the terrain file *Terrain.ter*

When both a terrain file and a surface roughness file are selected, they must both cover the same area.

---

*The values in the terrain height file should not all be equal.*

---

### *Surface roughness of offshore study site*

When an offshore site is modelled there are three methods to select the surface roughness. Two of these options are similar to those for onshore study sites: **Enter constant value**, and **Use values from .MET file**. See the corresponding descriptions for the onshore study site above. The default surface roughness value, 0.0001m, is a typical value for the sea.

The third option **Calculate values from marine parameters** is only available for offshore sites; please refer to Section 7.3 for further details about the calculation methodology. Selecting this option causes the following to become available:

* **Charnock parameter** - This is a constant used in the heat flux calculations. Typical values range from 0.018 to 0.08, but in the marine boundary layer this may depend on, for example, distance from the coast. Validation suggests that a value of 0.08 is most appropriate for regions not altogether remote from land (e.g. North Sea).
* **Assume marine boundary layer is neutral**, this option should be checked if no temperature or humidity measurements are available in the meteorological data file, for instance if the file is in *.tab* file format.
* **Height of temperature measurements** (in metres above sea level). This should be entered if temperature data are available and the marine boundary layer is not assumed to be neutral.
* **Specify humidity measurement height**, if humidity data are available in the meteorological data file this option should be checked and the height entered below. This option should not be checked if the meteorological data file is in *.tab* format.
* **Height of humidity measurements** (in metres above sea level). This should be entered if humidity data are available and the marine boundary layer is not assumed to be neutral.

## 3.3 Meteorology

The **Meteorology** folder consists of two screens, the **Basic meteorology** and **Advanced meteorology** screens. The **Basic meteorology** screen is shown in **Figure 3.4** and should be used to provide details about the met site and meteorological conditions. The **Advanced meteorology** screen is optional and can be used to provide further details if required.

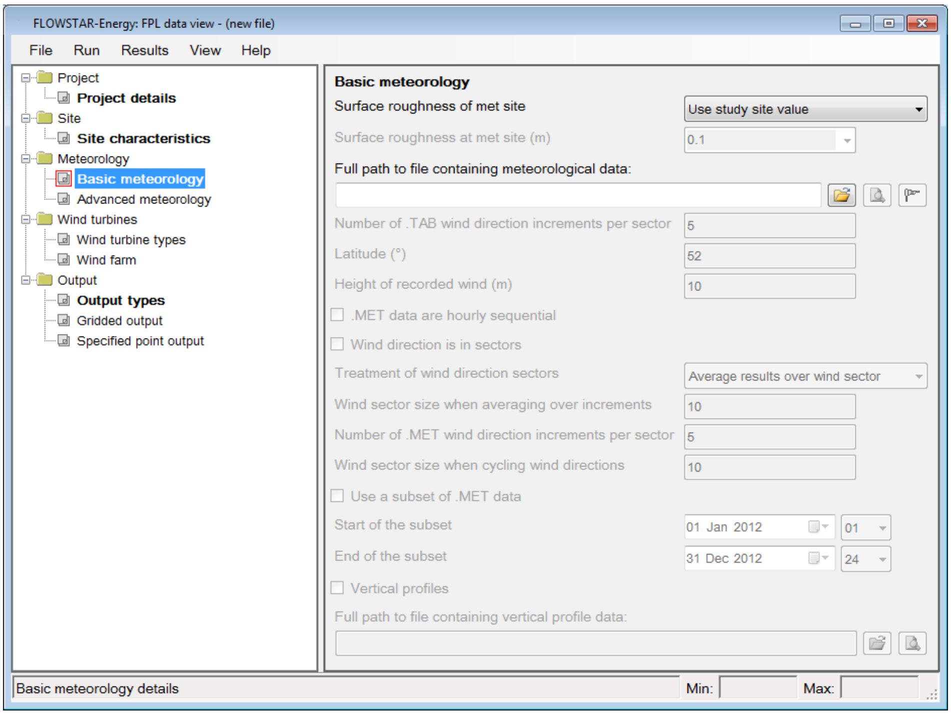



**Figure 3.4** – The **Basic meteorology** screen.

### 3.3.1 Basic meteorology

#### *Surface roughness of met site*

There are three options available for describing the surface roughness at the met site, one of which should be chosen from the drop down list. The options are:

* **Enter constant value** – When this option is chosen, the option **Surface roughness at met site (m)** below the drop down list becomes available. In the box, enter the surface roughness at the site (in metres) based on land use. See **Table 3.1** for land use types and corresponding values of surface roughness. The default value of surface roughness is 0.1 m, corresponding to a region of root crops.
* **Use values from .MET file** - To enter hourly varying surface roughness values for the met site via the meteorological data file, select this option and add the surface roughness values to the file using the variable name '`Z0 (M)`'. This option can only be used with a meteorological data file in the *.met* format.
* **Use study site value** – Using this option sets the surface roughness at the met site equal to that entered for the study site.

### ***Full path to file containing meteorological data:***

The meteorological data is entered through a prepared meteorological data file. This file may be an example file supplied in the *<install path>\Data* directory, a file prepared by the user from their own available data (for example from an on-site meteorological station) or a pre-formatted file obtained from a supplier. In order to select the meteorological file click on the browse button and search to find the file. The selected file can then be viewed using the view button .

There are two different formats of meteorological data that can be entered:

- a *.met* file, which can contain statistical data; a series of unrelated meteorological conditions; or a series of hourly sequential data covering a certain period, for instance a year.
- a *.tab* file, which is an observed wind climate file, and contains statistical meteorological data.

Examples of each of these can be found in the *<install path>\Data* directory. Some of the files in this directory are for use in the worked examples and do not contain meaningful data to be used in actual calculations. If the format of the meteorological file is a *.met* file, the wind rose button can be used to plot a wind rose in the **Wind rose viewer**. This option is not available for a *.tab* file; however, once a run has been completed a wind rose can be plotted for the *.tab* data using the **FPL data view** menu option **Results - Plot wind rose of processed met data**.

### **.met** ***file format***

The *.met* meteorological file format is a comma-separated format and uses a *.met* extension by convention. The data file should be as follows. The first line should contain the keyword "`VARIABLES:`". The second line should contain the number of parameters to be entered in the *.met* file. Then follows a list of the parameters by name (one per line) for which data are entered in the *.met* file (details of the names to be used for each parameter can be found in Section 7.1. The data section starts with the keyword "`DATA:`" followed by the actual data. For each line of meteorological data, the parameters should be entered in a row in the order that they are listed in the "`VARIABLES:`" section. Comments can be added before the "`VARIABLES:`" and between the "`VARIABLES:`" and "`DATA:`" sections, no blank lines should appear in the file except as part of the comments. Further details about the format of the *.met* file can also be found by looking at *MetDemo.met* which can be found in the *<install path>\Data* directory.

There is a minimum amount of data that must be entered. These are the wind speed (`U`) and wind direction (`PHI`) and either

- The Julian day number, time of day (local solar time) and cloud cover (`TDAY`, `THOUR` and `CL`),

or

- Sensible surface heat flux (`FTHETA0`),

or

- Reciprocal of Monin-Obukhov length (`RECIPLMO`).

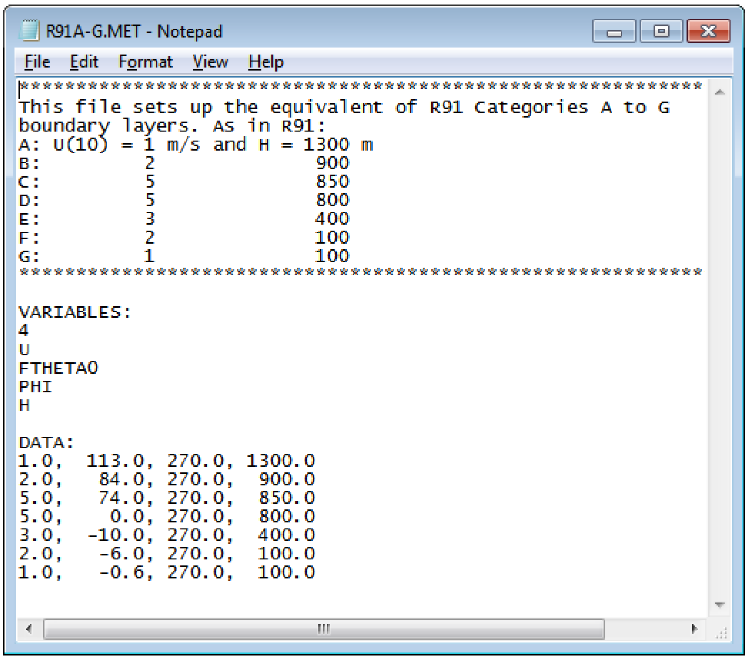
Additional meteorological parameters can be added to the meteorological data file, and should be included if known. Some model options require further parameters to be entered in this input file. Full details of the parameters that can be included can be found in Section 7.

---

*Any missing data should be identified using the value '-999'.*

---

There are a number of example *.met* files provided with FLOWSTAR-Energy. They are installed in the *<install_path>\Data* directory of your computer. A particularly useful file is *R91A-G.met*, which contains seven lines of meteorological data that correspond approximately to stability classes A-G. This file can be viewed in an editor such as Notepad or WordPad and is shown in **Figure 3.5**.



```
R91A-G.MET - Notepad
File  Edit  Format  View  Help
***************************************************************
This file sets up the equivalent of R91 Categories A to G
boundary layers. As in R91:
A: U(10) = 1 m/s and H = 1300 m
B:         2               900
C:         5               850
D:         5               800
E:         3               400
F:         2               100
G:         1               100
***************************************************************

VARIABLES:
4
U
FTHETA0
PHI
H

DATA:
1.0,  113.0, 270.0, 1300.0
2.0,   84.0, 270.0,  900.0
5.0,   74.0, 270.0,  850.0
5.0,    0.0, 270.0,  800.0
3.0,  -10.0, 270.0,  400.0
2.0,   -6.0, 270.0,  100.0
1.0,   -0.6, 270.0,  100.0
```

**Figure 3.5 –** The *.met* file *R91A-G.met*.

### .tab *file format*

The observed wind climate file format (*.tab*) contains the frequencies of the wind in a number of sectors and wind speed bins. It further contains the height of observation above ground level and the geographical coordinates (latitude and longitude) of the wind mast.

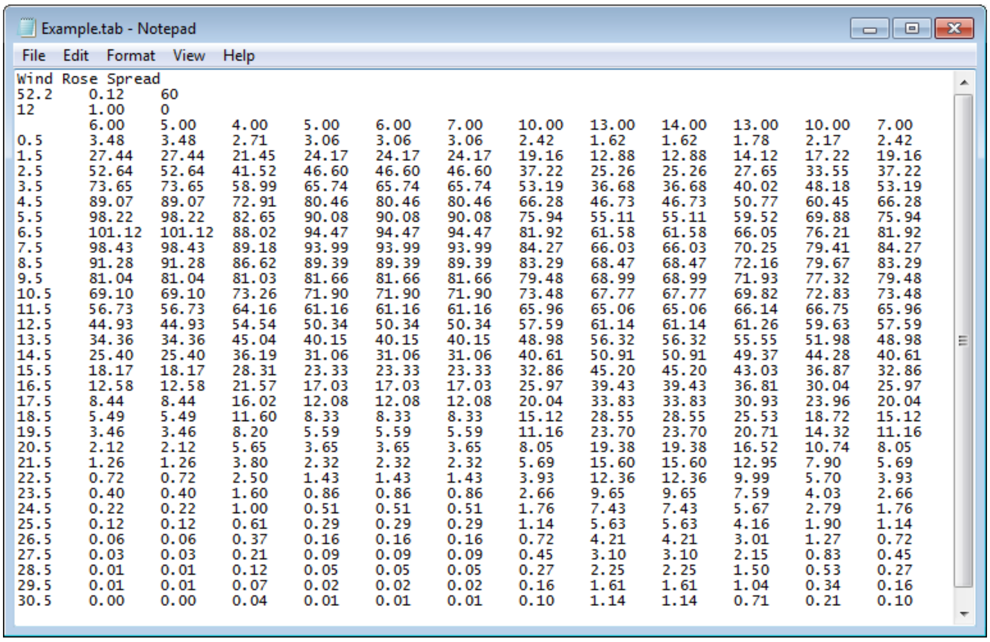
Data are stored in an ASCII (text) file. FLOWSTAR-Energy requires a *.tab* file to have the file name extension *.tab*. The general format of a *.tab* file is shown in **Table 3.3**.

| Line | Contents |
|---|---|
| 1 | Text string identifying the observed wind climate/anemometer |
| 2 | Latitude ($^0$), Longitude ($^0$) and height above ground level of anemometer (m) |
| 3 | Number of sectors, speed factor $a_u$ and direction offset $b_d$ ($^0$)<br>Wind speed bin limits (m/s) = $a_u$ x{column 1}<br>Wind rose rotated by $b_d$ |
| 4 | Sector-wise frequencies of occurrence (%) |
| 5 | Upper limit for speed class 1, sector-wise frequencies (per mille) in class 1 |
| 6 | Upper limit for speed class 2, sector-wise frequencies (per mille) in class 2 |
| 7-n | Same as line 5 and 6, but for speed class 3-n |

**Table 3.3** – format of *.tab* files

The speed distributions may be described by a maximum of 50 wind speed bins and 36 sectors. The wind speed bins need not have the same width and the bin limits need not be integer values. The frequencies of occurrence of wind speed are given in per mille, i.e. they will add up to 1000 for each sector. You may also give the frequency as an absolute number of hours of observation.

An example of an observed wind climate file is shown in **Figure 3.6**.




```
Wind Rose Spread
52.2    0.12    60
12      1.00    0
        6.00    5.00    4.00    5.00    6.00    7.00    10.00   13.00   14.00   13.00   10.00   7.00
0.5     3.48    3.48    2.71    3.06    3.06    3.06    2.42    1.62    1.62    1.78    2.17    2.42
1.5     27.44   27.44   21.45   24.17   24.17   24.17   19.16   12.88   12.88   14.12   17.22   19.16
2.5     52.64   52.64   41.52   46.60   46.60   46.60   37.22   25.26   25.26   27.65   33.55   37.22
3.5     73.65   73.65   58.99   65.74   65.74   65.74   53.19   36.68   36.68   40.02   48.18   53.19
4.5     89.07   89.07   72.91   80.46   80.46   80.46   66.28   46.73   46.73   50.77   60.45   66.28
5.5     98.22   98.22   82.65   90.08   90.08   90.08   75.94   55.11   55.11   59.52   69.88   75.94
6.5     101.12  101.12  88.02   94.47   94.47   94.47   81.92   61.58   61.58   66.05   76.21   81.92
7.5     98.43   98.43   89.18   93.99   93.99   93.99   84.27   66.03   66.03   70.25   79.41   84.27
8.5     91.28   91.28   86.62   89.39   89.39   89.39   83.29   68.47   68.47   72.16   79.67   83.29
9.5     81.04   81.04   81.03   81.66   81.66   81.66   79.48   68.99   68.99   71.93   77.32   79.48
10.5    69.10   69.10   73.26   71.90   71.90   71.90   73.48   67.77   67.77   69.82   72.83   73.48
11.5    56.73   56.73   64.16   61.16   61.16   61.16   65.96   65.06   65.06   66.14   66.75   65.96
12.5    44.93   44.93   54.54   50.34   50.34   50.34   57.59   61.14   61.14   61.26   59.63   57.59
13.5    34.36   34.36   45.04   40.15   40.15   40.15   48.98   56.32   56.32   55.55   51.98   48.98
14.5    25.40   25.40   36.19   31.06   31.06   31.06   40.61   50.91   50.91   49.37   44.28   40.61
15.5    18.17   18.17   28.31   23.33   23.33   23.33   32.86   45.20   45.20   43.03   36.87   32.86
16.5    12.58   12.58   21.57   17.03   17.03   17.03   25.97   39.43   39.43   36.81   30.04   25.97
17.5    8.44    8.44    16.02   12.08   12.08   12.08   20.04   33.83   33.83   30.93   23.96   20.04
18.5    5.49    5.49    11.60   8.33    8.33    8.33    15.12   28.55   28.55   25.53   18.72   15.12
19.5    3.46    3.46    8.20    5.59    5.59    5.59    11.16   23.70   23.70   20.71   14.32   11.16
20.5    2.12    2.12    5.65    3.65    3.65    3.65    8.05    19.38   19.38   16.52   10.74   8.05
21.5    1.26    1.26    3.80    2.32    2.32    2.32    5.69    15.60   15.60   12.95   7.90    5.69
22.5    0.72    0.72    2.50    1.43    1.43    1.43    3.93    12.36   12.36   9.99    5.70    3.93
23.5    0.40    0.40    1.60    0.86    0.86    0.86    2.66    9.65    9.65    7.59    4.03    2.66
24.5    0.22    0.22    1.00    0.51    0.51    0.51    1.76    7.43    7.43    5.67    2.79    1.76
25.5    0.12    0.12    0.61    0.29    0.29    0.29    1.14    5.63    5.63    4.16    1.90    1.14
26.5    0.06    0.06    0.37    0.16    0.16    0.16    0.72    4.21    4.21    3.01    1.27    0.72
27.5    0.03    0.03    0.21    0.09    0.09    0.09    0.45    3.10    3.10    2.15    0.83    0.45
28.5    0.01    0.01    0.12    0.05    0.05    0.05    0.27    2.25    2.25    1.50    0.53    0.27
29.5    0.01    0.01    0.07    0.02    0.02    0.02    0.16    1.61    1.61    1.04    0.34    0.16
30.5    0.00    0.00    0.04    0.01    0.01    0.01    0.10    1.14    1.14    0.71    0.21    0.10
```

**Figure 3.6** – Example *.tab* file

*.tab* files are available in four other variant file formats. However only the default format described above is supported by FLOWSTAR-Energy.

### *Other Basic Meteorology parameters*

The parameters on the remainder of the **Basic Meteorology** screen are greyed out until the meteorological file has been selected. If a *.tab* file has been entered the **Number of .TAB wind direction increments per sector** should be entered. The default value of 5 is suitable for small wind sector sizes, for instance up to 10 degrees, but for larger wind sector sizes it may be necessary to increase the number of increments; this will increase model run time. No other data is required as all other parameters, such as latitude, anemometer height and wind sector size, are contained within the *.tab* file.

The other parameters on the screen are only used if a *.met* file has been selected, and are discussed below.

- **Latitude**: The **Latitude** of the site must be entered by the user and must lie between -90° and 90°. The default is 52°. UK users should note that this represents a site on a line roughly between Fishguard and Harwich, and would therefore be appropriate for southern to mid-England and for South Wales.
- **Height of recorded wind (m)**: height of the wind measurements (default = 10 m).
- **.MET data are hourly sequential**: check the box if the meteorological data are hourly sequential or are to be interpolated to become hourly sequential. If statistical data or a series of unrelated meteorological data are entered, for instance to examine what happens during specific meteorological conditions, this box should not be checked. Hourly sequential data can either be entered using files prepared by the user, say from an on-site meteorological station or using files obtained from a supplier. Hourly sequential files are normally available for periods of one year. Each line of data represents one hour of measurements, and therefore a data file for one year contains 8760 lines of data.

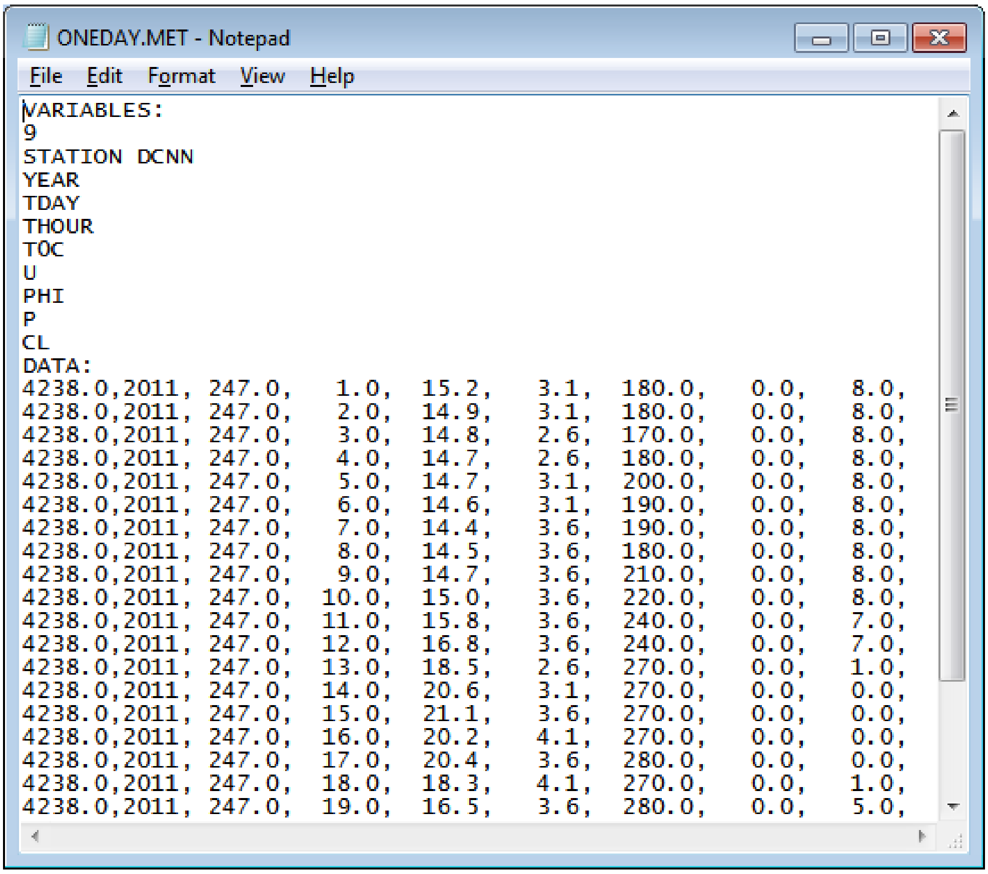


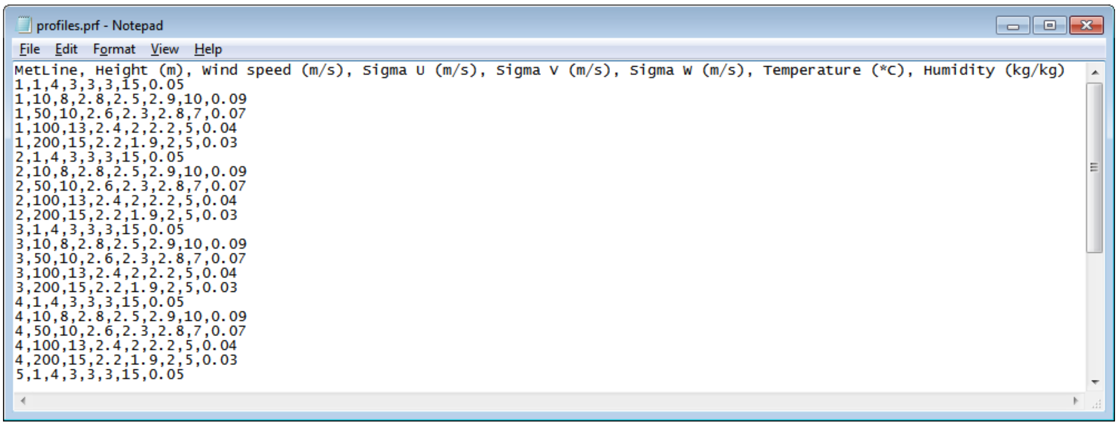

**Figure 3.7 –** Example of hourly sequential meteorological data file.

Hourly sequential files can be used for long-term averages. Since sequential data files consist of continuous hour-by-hour measurements, they are a better representation of actual meteorological conditions than statistically-averaged data files. However, it is important to note that year-to-year variations occur in meteorological conditions.

- **Wind direction is in sectors**: wind direction measurements (`PHI`) are often reported to the nearest 10 degrees, i.e. the wind data are in 10 degree sectors. If this is the case, check the box and select from the drop down list whether to: **Average results over wind sector**, or to use **Wind direction cycling within sector**. For both cases the wind sector size should be entered into the appropriate box. When using **Average results over wind sector** the **Number of .MET wind direction increments per sector** should also be entered. The default value of 5 is suitable for small wind sector sizes, for instance up to 10 degrees, but for larger wind sector sizes it may be necessary to increase the number of increments; this will increase model run time.
- **Use a subset of .MET data**: check this box to only use a subset of the meteorological data; the dates and times of the first and last met. lines required should be entered.
- **Vertical profiles**: enter user-defined vertical profile data for wind speed, turbulence parameters, temperature and specific humidity using a *.prf* file. The file format is described below.

### *User-input vertical profiles of meteorological data*

An example *.prf* file is shown in **Figure 3.8** to illustrate the format of the file.




```
MetLine, Height (m), Wind speed (m/s), Sigma U (m/s), Sigma V (m/s), Sigma W (m/s), Temperature (*C), Humidity (kg/kg)
1,1,4,3,3,3,15,0.05
1,10,8,2.8,2.5,2.9,10,0.09
1,50,10,2.6,2.3,2.8,7,0.07
1,100,13,2.4,2,2.2,5,0.04
1,200,15,2.2,1.9,2,5,0.03
2,1,4,3,3,3,15,0.05
2,10,8,2.8,2.5,2.9,10,0.09
2,50,10,2.6,2.3,2.8,7,0.07
2,100,13,2.4,2,2.2,5,0.04
2,200,15,2.2,1.9,2,5,0.03
3,1,4,3,3,3,15,0.05
3,10,8,2.8,2.5,2.9,10,0.09
3,50,10,2.6,2.3,2.8,7,0.07
3,100,13,2.4,2,2.2,5,0.04
3,200,15,2.2,1.9,2,5,0.03
4,1,4,3,3,3,15,0.05
4,10,8,2.8,2.5,2.9,10,0.09
4,50,10,2.6,2.3,2.8,7,0.07
4,100,13,2.4,2,2.2,5,0.04
4,200,15,2.2,1.9,2,5,0.03
5,1,4,3,3,3,15,0.05
```

**Figure 3.8** – Example *.prf* file

The *.prf* file should be comma separated. The first line of the file is a header line; the rest of the lines then contain data in the following format:

```
Met line, Height, Wind speed, Sigma U, Sigma V, Sigma W, Temp.,
Spec. humidity
```

Data should be entered for at least two heights for every line of data in the meteorological file. The lines should be in order with the heights in ascending order for each meteorological data line. All of the columns must be present for each line of data; if values are unknown for any parameter then a value of -999 should be entered.

The variables entered in the meteorological file are as follows:

1. `Met line`: the meteorological line number corresponding to this line of data

   ---

   *If the option to use a meteorological data subset is selected then the* .prf *file should contain only data for that subset with meteorological line numbering relative to the start of the subset.*

   ---

2. `Height`: the height in metres (must be greater than 0 m)
3. `Wind speed`: the wind speed in m/s

   The wind speeds entered should be consistent with the wind speeds in the meteorological file.

   Minimum > 0 m/s
   Maximum = 100 m/s

4. `Sigma U`: The along wind turbulence in m/s

   Minimum > 0 m/s
   Maximum = 1000 m/s

5. `Sigma V`: The cross wind turbulence in m/s

   Minimum > 0 m/s
   Maximum = 1000 m/s

6. `Sigma W`: The vertical turbulence in m/s

   Minimum > 0 m/s
   Maximum = 1000 m/s

7. `Temperature`: the temperature in °C

   Minimum =  100°C
   Maximum = 60°C

8. `Specific humidity`: the specific humidity in kg/kg

   Minimum = 0 kg/kg
   Maximum = 0.1 kg/kg

### *Interpolation of meteorological data*

If hourly sequential data are not available, but a chronological sequence of meteorological data are available then, providing certain constraints are met, the data will be interpolated to become hourly sequential. To enable this option, enter your meteorological data with a meteorological data file in the *.met* format and check the **Met. Data are hourly sequential** box on the **Basic meteorology** screen.

In order for the meteorological data to be interpolated, they must meet the following criteria:

- Contain year, day and hour;

- Be in chronological order, with gaps no larger than 9 hours;
- Contain wind speed, wind direction and cloud cover; and
- Not contain any variables that cannot be interpolated – please refer to Section 7.1 for details.

---

*If the meteorological data entered are hourly sequential and the* ***Met. Data are hourly sequential*** *box is checked, no interpolation will be carried out and the restrictions regarding which variables can be entered do not apply.*

---

If the chronological sequence of meteorological data entered can be interpolated by the model to a series of hourly sequential data covering the same period, the interpolated meteorological data will then be modelled.

### 3.3.2 Advanced meteorology

The **Advanced meteorology** screen is shown in **Figure 3.9**.

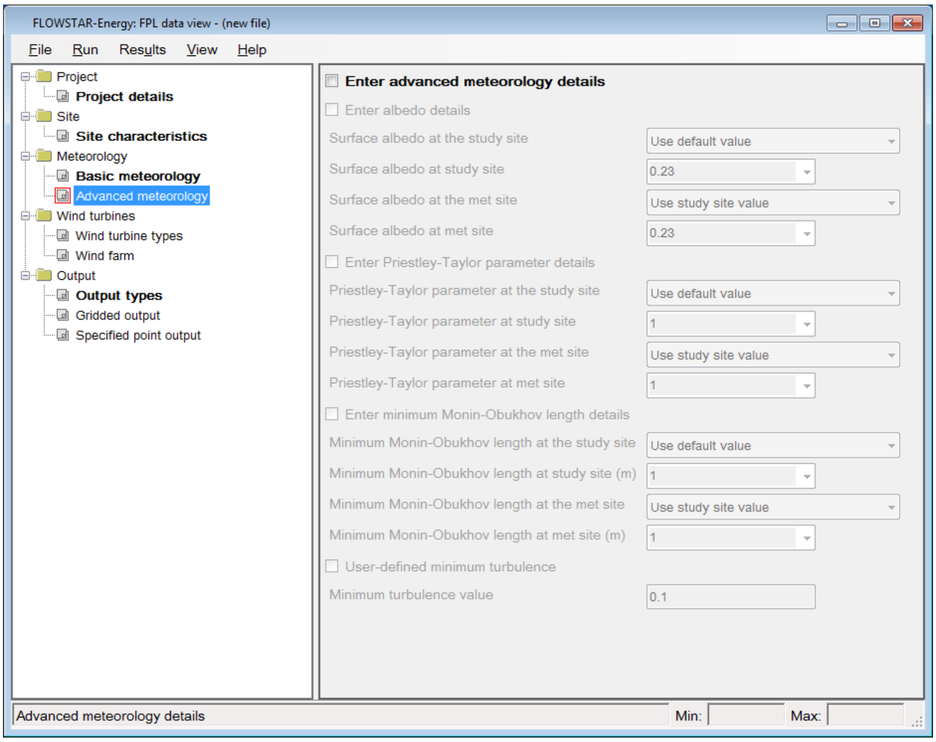



**Figure 3.9** – The **Advanced meteorology** screen

For both the study site and met. measurement site, **Enter advanced meteorology details** can be selected to allow the user to alter the values of surface albedo, Priestley-Taylor parameter and minimum Monin-Obukhov length between the study site and meteorological measurement site. The default values of these parameters are provided for a typical rural UK site. This screen allows the user to specify values of these parameters more suitable for the site being modelled. A user-defined minimum turbulence value can also be entered.

### *Enter albedo details*

The surface albedo is the ratio of reflected to incident shortwave solar radiation at the surface of the earth. It therefore lies in the range 0 to 1. In particular, it takes a high value (high proportion of incident radiation reflected) when the ground is snow-covered. The default value for surface radiation is 0.23, i.e. not snow covered (Oke, 1987). The following options are available for the surface albedo.

**Use default value**: use the default value (0.23); this is the default option for the dispersion site.

**Enter constant value**: Enter a user-defined constant value. Either enter a value directly into the box or select a value from the list using the drop down menu:

- **Snow-covered ground** = 0.6
- **Not snow-covered** = 0.23

**Use values from the .MET file**: Use hourly varying values of surface albedo from the meteorological file. Refer to Section 7.1 for details of the variable names to use in the meteorological file. This option can only be used with a *.met* format file.

At the meteorological measurement site there is an additional option of **Use study site value;** select this to ensure that the surface albedo values at the study site and the meteorological measurement site are the same. This is the default option for the meteorological measurement site.

### *Enter Priestley-Taylor parameter details*

The Priestley-Taylor parameter is a parameter representing the surface moisture available for evaporation. The Priestley-Taylor parameter must be between 0 and 3 and the default value is 1 corresponding to moist grassland (Holtslag and van Ulden, 1983). The following options are available for the Priestley-Taylor parameter.

**Use default value**: use the default value of 1.

**Enter constant value**: Enter a constant value. Either enter a value directly into the box or select a value from the list using the drop down menu:

- **Dry bare earth** = 0
- **Dry grassland** = 0.45
- **Moist grassland** = 1

**Use values from .MET file**: Use hourly varying values of Priestley-Taylor parameter from the meteorological file. See Section 7.1 for details of the variable names to use in the meteorological file. This option can only be used with a *.met* format file.

At the meteorological measurement site there is an additional option of **Use study site value**; select this option to ensure that the Priestley-Taylor parameter values at the dispersion site and the meteorological measurement site are the same. This is the default option for the meteorological measurement site.

### *Enter minimum Monin-Obukhov length details*

This is an option to specify the *minimum* value of the Monin-Obukhov length, $L_{MOmin}$. This allows for the effect of heat production in cities, which is not represented by the meteorological data. The Monin-Obukhov length provides a measure of the stability of the atmosphere (see Section 7.1). In very stable conditions in a rural area its value would typically be 2 to 20 m. In urban areas, there is a significant amount of heat generated from buildings and traffic, which warms the air above the town/city. For large urban areas this is known as the urban heat island. It has the effect of preventing the atmosphere from ever becoming very stable. In general, the larger the area, the more heat is generated and the stronger this effect becomes. This means that in stable conditions the Monin-Obukhov length will never fall below some minimum value; the larger the city, the larger the minimum value. The mimimum Monin-Obukhov length should be between 1 and 200 m; the default value is 1 m corresponding to a rural area.

The following options are available for the minimum Monin-Obukhov length:

**Use default value**: use the model default value; this is the default option for the dispersion site.

**Enter constant value**: Enter a value, either by typing directly into the box, or by selecting from a list using the drop down menu:

- **Large conurbations > 1 million** = 100 m
- **Cities and large towns** = 30 m
- **Mixed urban/industria**l = 30 m
- **Small towns < 50,000** = 10 m

At the meteorological measurement site there is an additional option of **Use study site value**; select this option to ensure the minimum Monin-Obukhov lengths at the dispersion site and the meteorological measurement site are the same. This is the default option for the meteorological measurement site.

### *User-defined minimum turbulence*

An option exists to specify the **Minimum turbulence value**, which is typed into the box available.

If a user-defined minimum turbulence value is *not* used, the model determines a minimum value for the turbulence parameters, $\sigma_{min}$, from the minimum value of the Monin-Obukhov length, $L_{MOmin}$, in the following way:

(3.1)
$$\sigma_{min} = \begin{cases} 0.01 & L_{MOmin} \le 10 \\ 0.01 + 0.0095(L_{MOmin} - 10) & 10 < L_{MOmin} < 30 \\ 0.2 & L_{MOmin} \ge 30 \end{cases}$$

In addition to this calculation, if complex terrain is modelled a further minimum of 0.1 m/s is applied.

# 3.4 Wind turbines

## 3.4.1 Wind turbine types

Details of any wind turbine types to be modelled should be entered on the **Wind turbine types** screen, shown in **Figure 3.10**.

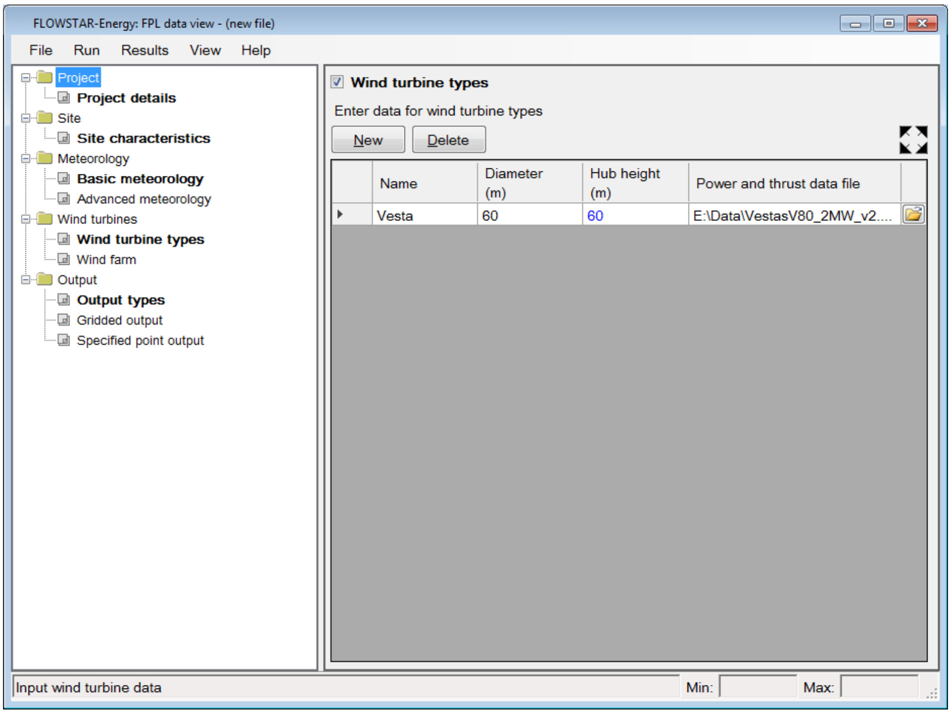



**Figure 3.10** – The **Wind turbine types** screen

In order to specify the wind turbine types, first check the box beside **Wind turbine types**. This enables the data to be entered in the table below. For each wind turbine type a separate row of data should be entered in the table. The required data are shown in **Table 3.4**.

| Column | Contents |
|---|---|
| Name | Name of the make, or type, of turbine. This name is referenced in the **Wind farm** and **Output types** screens. |
| Diameter (m) | Turbine rotor diameter in metres. |
| Hub height (m) | Turbine hub height in metres. The hub height is measured from ground level to the turbine hub. |
| Power and thrust data file | Full path to file containing power and thrust data, (*.wtd*) file. The browse button can be used to select this file. |

**Table 3.4** – columns in the **Wind turbine types** table

Wind turbine types may be added to or deleted from the table by clicking the **New** and **Delete** buttons. To copy data to another application such as Microsoft Excel, select the cells in the table and then right-click and choose **Copy**. To paste data from another application, select the first destination cell in the table, right-click and choose **Paste**. When editing the data for wind turbine types, **Full screen editing** can be selected by pressing the button in the top right hand side of the table: .

The wind turbine data (*.wtd*) file is used to provide the model with the power and thrust coefficients for a particular type of turbine as functions of wind speed. This data is usually available from the turbine manufacturer. The *.wtd* file must be created by the user, and this can be done in a text editor, such as Notepad, or in a spreadsheet package such as Excel. An example file illustrating the required format is shown in **Figure 3.11**.

Each line of data in the file includes:

- `U_HUB`: the wind speed (m/s) at the turbine hub height
- `POWER`: the power (kW) generated by the turbine at the corresponding wind speed
- `CT`: the thrust coefficient for the turbine at the corresponding wind speed

To create the file, follow these steps:

- Add the version string `WTDFileVersion2` as the first line in the file
- Create the "`VARIABLES:`" section

  This is used to list the variables that are included in the file.

  * On the second line type `VARIABLES:`
  * On the third line type `3`

    This indicates that there are values for three variables in the file.

  * On lines four to six list the variable names `U_HUB`, `POWER,CT`

    These can appear in any order, but the data in the "`DATA:`" section must match this order.

- Create the "`DATA:`" section.
  * On a new line type `DATA:`
  * After the "`DATA:`" keyword enter a line of data giving the wind speed and the corresponding power and thrust coefficient at that wind speed. The order of these variables must match the order specified in the "`VARIABLES:`" section.
  * Add an additional line of data for each wind speed.

---

*CT is only required in the* .wtd *file if a wind farm is selected; if a wind farm is not selected simply enter 2 instead of 3 on the third line of the* .wtd *file and omit CT from the list of variables and from the columns of data*

---

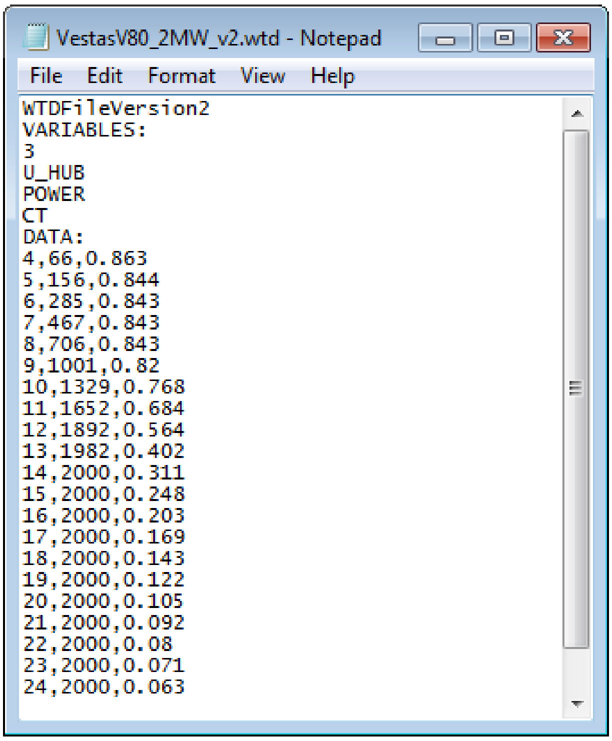
A *.wtd* file must be provided for each type of turbine defined in the model. Note that the *.wtd* file defines the range of wind speeds for which the wind turbine can be operational. If the hub height wind speed for a given met line is outside this range then the wind turbine will be considered not to be operational and will therefore produce no power and have no effect on the flow field for that met line.

---

*An example* .wtd *file is included in the* \Data *folder in the install directory.*

*If you have created your* .wtd *files with a spreadsheet package, it is always advisable to check the file format in a text editor afterwards to see if it is correct. In particular, trailing commas may need to be removed from the "*`VARIABLES:`*" section.*

---




```
WTDFileVersion2
VARIABLES:
3
U_HUB
POWER
CT
DATA:
4,66,0.863
5,156,0.844
6,285,0.843
7,467,0.843
8,706,0.843
9,1001,0.82
10,1329,0.768
11,1652,0.684
12,1892,0.564
13,1982,0.402
14,2000,0.311
15,2000,0.248
16,2000,0.203
17,2000,0.169
18,2000,0.143
19,2000,0.122
20,2000,0.105
21,2000,0.092
22,2000,0.08
23,2000,0.071
24,2000,0.063
```

**Figure 3.11** – Example *.wtd* file

### 3.4.2 Wind farm

To include the effect of wind turbines on the flow field or to obtain wind energy predictions for individual wind turbines that account for wind turbine wake effects, a wind farm must be selected on the **Wind farm** screen, shown in **Figure 3.12**. This screen is optional; potential wind energy output requires only wind turbine type definitions and the user can obtain this output without completing the **Wind farm** screen.

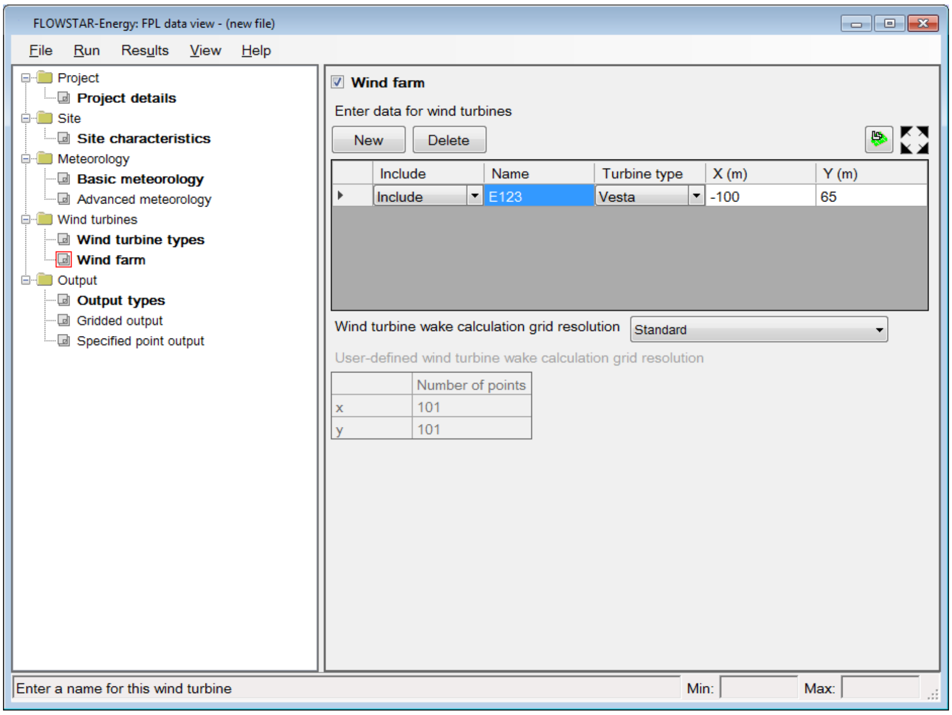



**Figure 3.12** – The **Wind farm** screen

The **Wind farm** screen is used to input the locations of each turbine and the make, or type, of turbine at each location. A wind farm is defined as one or more turbines. For each turbine the following information should be given:

- **Include**: choose whether or not to include the turbine in the current model simulation.
- **Name**: the name of the turbine at the location, e.g. "Turbine 1".
- **Turbine type**: the name of the make or type of turbine at the location, selected from a drop-down list of available turbine types. The user should complete the **Wind turbine types** screen before the **Wind farm** screen.
- **X(m)**: the easting of the turbine location in metres.
- **Y(m)**: the northing of the turbine location in metres.

Wind turbines may be added to or deleted from the table by clicking the **New** and **Delete** buttons. To copy data to another application such as Microsoft Excel, select the cells in the table and then right-click and choose **Copy**. To paste data from another application, select the first destination cell in the table, right-click and choose **Paste**. When editing the wind farm data **Full screen editing** can be selected by pressing the button in the top right hand side of the table. The **map view** cannot be used to edit data while the **FPL data view** is in **Full screen editing** mode.

To add a wind turbine in the **map view**, follow the steps below.

**Step 1** Click the map view button to open the **map view**.

**Step 2** Select the **Wind turbines** layer in the Legend.

**Step 3** Click on the **Add Feature** button on the toolbar to select it.

**Step 4** Click in the map at the location where you want the wind turbine to be placed.

**Step 5** A new turbine is created and displayed in the **Wind farm** screen of the **FPL data view**. The turbine coordinates have been filled in automatically but you will need to fill in the other parameters for the turbine as described above.

To move a wind turbine in the **map view**, follow the steps below.

**Step 1** Select the **Wind turbines** layer in the **Legend**.

**Step 2** Click on the **Shift Feature** button on the toolbar to select it.

**Step 3** Click on the appropriate turbine, and drag it to a new location; that is to say, while holding down the mouse button move the cursor to the new location for the turbine before releasing the mouse button.

**Step 4** Click on the **Save Edits** button on the toolbar to save the changes to the **FPL data view**.

**Step 5** Repeat this process to move other turbines.

To delete a wind turbine in the **map view**, follow the steps below.

**Step 1** Select the **Wind turbines** layer in the **Legend**.

**Step 2** Click on the **Edit Feature** button on the toolbar to select it.

**Step 3** Select the turbine you want to delete by clicking on it in the map. The selected turbine will have a red dot on it.

**Step 4** Click on the **Delete Feature** button on the toolbar.

**Step 5** This will bring up a dialogue box asking if you are sure you wish to delete the feature. Click **Yes** to delete the turbine.

**Step 6** Click on the **Save Edits** button on the toolbar to save the changes to the **FPL data view**.

**Step 7** Repeat this process to delete other turbines.

The **Wind turbine wake calculation grid resolution** is set within the **Wind farm** screen of the **FPL data view**. The user has the choice from a drop down list of a **Standard**, **High** or **User-defined** resolution grid. The **Standard** resolution uses the smallest wind turbine diameter being modelled as the grid spacing; the **High** resolution uses half the smallest wind turbine diameter being modelled as the grid spacing and will increase model run time compared with **Standard** resolution. If a **User-defined** resolution grid is chosen, a table below the drop down selection box becomes editable. The number of points required in the x and y directions (between 2 and 501) can be entered in the table.

Note that the grid resolution here refers to the internal grid used to calculate the wind turbine wakes and is different to that in the **Gridded output** screen, which refers to the grid of points on which output is given.

---

*Care should be taken with high-resolution calculation grids since the model run time and memory requirements partly depend on the number of calculation points.*

---

# 3.5 Output

## 3.5.1 Output types

This section provides an overview of the output types available from FLOWSTAR-Energy; please refer to **Section 4** for a detailed description of each individual output file type available. The **Output types** screen is shown in **Figure 3.13**.

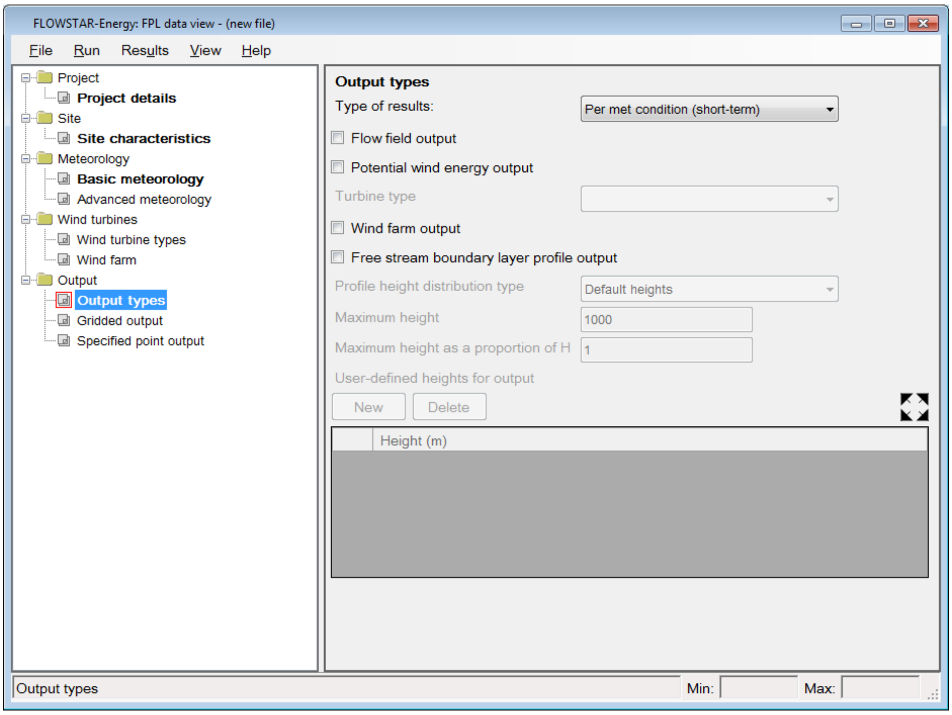



**Figure 3.13** – The **Output types** screen

### *Type of results*

There are two possible types of result, **Per met condition (short-term)** or **Summary results (long-term)**. The type of result required is selected using the drop down list. If **Per met condition (short-term)** is chosen then results will be output for every met line modelled. If **Summary results (long-term)** is chosen then the output results for all the meteorological data will be output.

### *Flow field output*

Flow field output gives the calculated wind speed, wind direction and components of turbulent velocity at the locations specified in the **Gridded output** screen and the **Specified point output** screen. This type of output includes the effect of hills, spatially varying roughness and wind turbines (if a wind farm is selected) on the flow.

### *Potential wind energy output*

This selection box should be checked to output the wind speed and wind energy available for a selected turbine type at the locations specified in the **Gridded output** screen and the **Specified point output** screen. Once this option is selected, the turbine type required should be selected from the drop down list of available wind turbine types. It is not necessary to complete the **Wind farm** screen in order to use this option, but if a wind farm is selected then the potential wind energy result will be based on a flow field that includes the effect of the wind farm. This type of output is useful, for example, where a hilly site or a site near to an existing wind farm is being considered as a site for a new wind farm, but the locations and/or number of new wind turbines in the proposed new wind farm is not yet known.

#### *Wind farm output*

This selection box should be checked to output the free stream, gross and net wind speed, turbulence and energy for each wind turbine in the wind farm. It is necessary to complete the **Wind farm** screen in order to use this option. Here the term 'free stream' is used to mean the values ignoring the effect of hills, spatially-varying surface roughness and wind turbines; 'gross' is used to mean the values including the effect of hills and/or spatially varying surface roughness but ignoring wind turbine wake effects; 'net' is used to mean the values including the effect of hills, spatially-varying roughness and wind turbine wake effects. This type of output is useful, for example, to obtain predicted annual energy production figures for a wind farm.

#### *Free stream boundary layer profile output*

FLOWSTAR-Energy creates a vertical profile of the free-stream boundary layer properties, and these properties may be output to a *.pro* file at a range of heights. Check this selection box to output a*.pro* file. There is a choice of heights at which to output the data. A drop down list gives four choices of output type, the choices are:

- **Default heights** - A geometric series of 100 heights up to the boundary layer height will be used.

- **Define absolute max height** – the entry box for **Maximum height** becomes enabled when this choice is selected; the maximum height required (in metres) for the output should be entered. A geometric series of 100 heights up to the specified fixed height will be used.

- **Define max height relative to H** – the entry box **Maximum height as a proportion of H** becomes enabled when this choice is selected. The proportion of boundary layer height (H) entered should be a factor between 0 and 5. A geometric series of 100 heights up to the specified proportion of the boundary layer will be used.

- **Define all heights** – a table to enter **User-defined heights for output** becomes enabled when this option is chosen. Up to 200 heights may be defined (in metres).

### 3.5.2 Gridded output

The **Gridded output** and **Specified point output** screens allow the user to define output points at which flow field output and/or potential wind energy output will be given by the model. Both these screens are optional: output points may be defined as a grid, individually as specified points, or as both.

FLOWSTAR-Energy uses a Cartesian coordinate system which refers to a point by a pair of X and Y coordinates. Units are metres. Usually, the X direction points eastwards; and the Y direction northwards.

The **Gridded output** screen shown in **Figure 3.14** allows the user to define a regular output grid. To enable gridded output, check the box **Enter output grid details for flow field and potential wind energy output**. Grids of output points may be defined with regular or variable spacing.

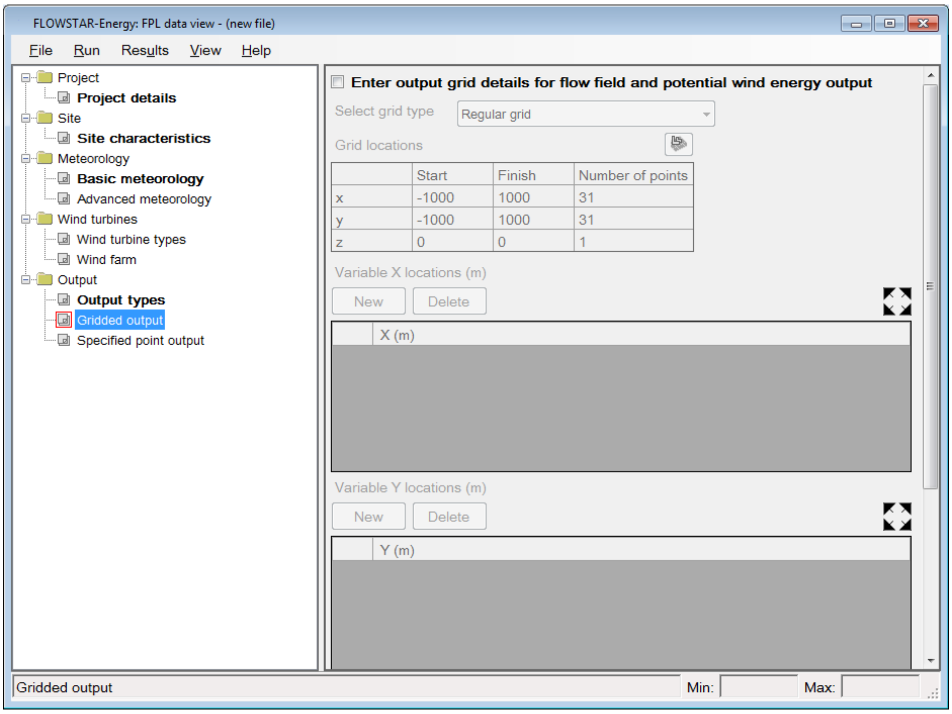



**Figure 3.14** – The **Gridded output** screen

### *Regular grid*

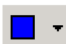
**Regular** output grids are specified by the **Start** and **Finish** X, Y and Z coordinates, representing the minimum and maximum values in each direction, together with the **Number of points** along the X, Y and Z axes. The maximum resolution in each direction is 2001 points.  The grid spacing is automatically calculated once the number of points in each direction has been set. The regular output grid can be edited in the **FPL data view** by typing the desired values, or in the **map view**.

To define a regular output grid in the **map view**, follow the steps below.

**Step 1** In the **FPL data view**, check the box **Enter output grid details for flow field and potential wind energy output** and select **Regular grid** as the grid type.

**Step 2** Click the map view button  to open the **map view**.

**Step 3** Select the **Output grid extent** layer in the **Legend**.

**Step 4** Click on the **Add Feature** button  on the toolbar to select it.

**Step 5** Click on the location of one corner of the output grid in the map, and while holding down the mouse button, move the cursor to draw the output grid required before releasing the mouse button.

**Step 6** The output grid extent is then displayed in the **FPL data view** table **Grid**

**locations**. The **Start** and **Finish** values for **X** and **Y** have been filled in automatically.

**Step 7** In the **Grid locations** table, enter the **Number of points** for all three directions, and the **Start** and **Finish** values for **Z**.

#### *Variable grid*

**Variable** output grids are specified by X, Y and Z coordinates entered in the **Variable X locations (m)**, **Variable Y locations (m)** and **Variable Z locations (m)** tables as shown in **Figure 3.14**. To enter **X**, **Y** and **Z** values, click the **New** button above the relevant table, then type the value in the table. To remove values, select them in the table, and click the **Delete** button. To copy data to another application such as Microsoft Excel, select the cells in the table and then right-click and choose **Copy**. To paste data from another application, select the first destination cell in the table, right-click and choose **Paste**. The table may be edited in full screen mode by pressing the button in the top right corner: . Variable output grids can't be viewed or edited in the **map view**.

Results will be calculated on a grid of points at the specified **X**, **Y** and **Z** values. For example, if three **X** values, four **Y** values and one **Z** value are specified, results will be calculated on a grid of 12 points. The maximum size of the variable grid in any dimension is 101 points.

### 3.5.3 Specified point output

To create output at specified points, click the **Enter specified points for flow field and potential wind energy output** option. **Specified points** may be added to or deleted from the table shown in **Figure 3.15** by clicking on the **New** and **Delete** buttons, respectively, or with the **map view** (see below).

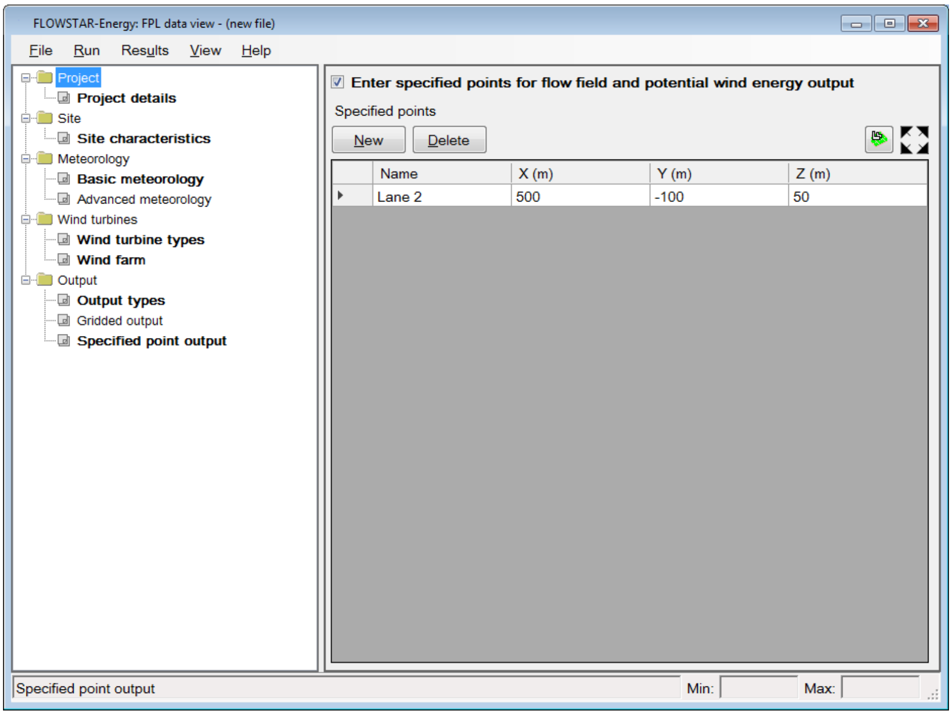



**Figure 3.15** – **Specified point output** screen

The maximum number of specified points is 1000. Points may be given a name by the user (e.g. "point 1") and have to be specified with **X, Y** and **Z** coordinates. The names given to the specified points must not contain commas.

To copy data to another application such as Microsoft Excel, select the cells in the table and then right-click and choose **Copy**. To paste data from another application, select the first destination cell in the table, right-click and choose **Paste**. The table may be edited in **Full screen editing** mode by pressing the button in the top right corner: . The **map view** cannot be used to edit data while the **FPL data view** is in **Full screen editing** mode. The specified points may be viewed or edited in the **map view** by pressing the map view button: .

To add a specified point in the **map view**, follow the steps below.

**Step 1** Click the map view button to open the **map view**.

**Step 2** Select the **Specified point** layer in the Legend.

**Step 3** Click on the **Add Feature** button on the toolbar to select it.

**Step 4** Click in the map at the location where you want the specified point to be placed.

**Step 5** A new specified point is created and displayed in the **Specified point output** screen of the **FPL data view**. The X and Y coordinates have been filled in automatically but you will need to fill in the other parameters (**Z** and **Name)** as described above.

To move a specified point in the **map view**, follow the steps below.

**Step 1** Select the **Specified point** layer in the **Legend**.

**Step 2** Click on the **Shift Feature** button on the toolbar to select it.

**Step 3** Click on the appropriate specified point, and drag it to a new location; that is to say, while holding down the mouse button move the cursor to the new location for the specified point before releasing the mouse button.

**Step 4** Click on the **Save Edits** button on the toolbar to save the changes to the **FPL data view.**

**Step 5** Repeat this process to move other specified points.

To delete a specified point in the **map view**, follow the steps below.

**Step 1** Select the **Specified points** layer in the **Legend**.

**Step 2** Click on the **Edit Feature** button on the toolbar to select it.

**Step 3** Select the specified point you want to delete by clicking on it in the map. The selected point will have a red dot on it.

**Step 4** Click on the **Delete Feature** button on the toolbar.

**Step 5** This will bring up a dialogue box asking if you are sure you wish to delete the feature. Click **Yes** to delete the specified point.

**Step 6** Click on the **Save Edits** button  on the toolbar to save the changes to the **FPL data view.**

**Step 7** Repeat this process to delete other specified points.

# SECTION 4  Model Output

A variety of output data is produced by FLOWSTAR-Energy according to the run configuration. This section describes the possible output and presents the tools available from the **FPL data view Results** menu, which is shown in **Figure 4.1**.

The first two options of the menu are features for visualising model results: plots can be made of potential wind energy or flow fields either in the **map view** (**Plot in map view...**) or in Surfer (**Plot in Surfer...**). Plotting in the **map view** is discussed in Section 4.1 and plotting in Surfer is discussed in Section 4.2.

The next four options on the **Results** menu are used to view numerical short-term average results in the chosen viewing application; these options are discussed in Section 4.3.

There are then three options to view numerical long-term average results, which are discussed in Section 4.4.

Viewing processed met data and plotting a wind rose is discussed in Section 4.5.

Viewing the report, log, warnings and errors files produced during the run is discussed in Section 4.6.

The last option on the **Results** menu is to **Open results folder**, this starts a new Windows Explorer window open in the folder containing all the results files.

---

***Results** menu options activate the results for the .fpl file currently loaded in the **FPL data view**; each option is therefore only enabled if the relevant results file exists.*

---

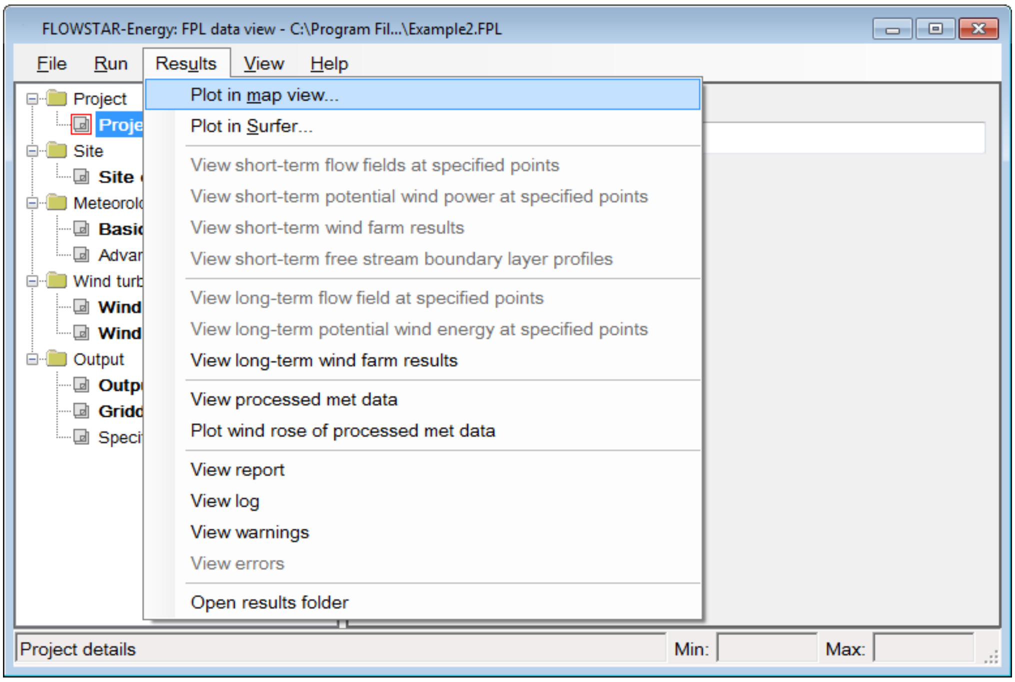



**Figure 4.1 – Results** menu.

## 4.1 Plot in map view

Contour and vector plots of model output results can be created and viewed in the **map view** using the **2-D Output Plotter**; this can produce contour plots using the in-built interpolator, or using Golden Software's Surfer, if the user has this program installed.

There are two toolbar buttons for plotting different types of output in the FLOWSTAR-Energy **map view**:

- **Wind energy**:  for plotting contour plots of gridded potential wind energy output including: wind speed, potential wind energy and estimated capacity factor. Specified point data can also be plotted if there are enough specified points in the results file.

- **Flow field**:  for plotting contour or vector plots of flow field output, including wind speed and turbulent velocity.

Section 4.1.1 describes how to plot contours of potential wind energy output. Section 4.1.2 describes how to plot flow field output.

### 4.1.1 Plotting potential wind energy output in the map view

Potential wind energy output contains wind speed at hub height, potential wind energy and estimated capacity factor calculated for the selected wind turbine type from the simulated flow field:

- for each line of meteorological data if short-term output is selected
- as average wind speed and total wind energy over all meteorological data lines if long-term output is selected.

***Potential wind energy output: .gst files***

The *.gst* file is only created for short-term calculations with an output grid if potential wind energy output is selected; it contains wind speed, potential wind power and estimated capacity factor at the locations defined for the output grid for each line of meteorological data.

Variables are:

- **Year**, **Day**, **Hour**, **Time(s)**: Year, Julian day, hour of the day and time from the meteorological data
- **X(m)**: X coordinate of output point
- **Y(m)**: Y coordinate of output point
- **Z(m)**: Redundant variable
- **Wind|m/s|Hub height = Xm|Turbine type|-|1hr**: horizontal wind speed (m/s); **X** is the turbine hub height (m); **Turbine type** is the name of the turbine type selected.
- **Power|kW|Hub height = Xm|Turbine type|-|1hr**: Wind power (kW) that has the potential

to be produced by the selected wind turbine type at the current met line and the current output point; **X** is the turbine hub height (m); **Turbine type** is the name of the turbine type selected.

- **CapFac|%|Hub height = Xm|Turbine type|-|1hr**: Estimated capacity factor for the selected wind turbine at the current met line and output point, defined as the **Power** divided by the maximum power that could be produced by the selected wind turbine type and expressed as a percentage (%); **X** is the turbine hub height (m); **Turbine type** is the name of the turbine type selected.

### *Potential wind energy output: .glt files*

The *.glt* file is only created for long-term calculations with an output grid if potential wind energy output is selected; it contains average wind speed, total potential wind energy and estimated capacity factor at the locations defined for the output grid.

Variables are:

- **X(m)**: X coordinate of output point
- **Y(m)**: Y coordinate of output point
- **Z(m)**: Redundant variable
- **Wind|m/s|Hub height = Xm|Turbine type|- |1hr**: Mean horizontal wind speed (m/s); **X** is the turbine hub height (m); **Turbine type** is the name of the turbine type selected.
- **Energy|kWh|Hub height = Xm|Turbine type|- |1hr**: Wind energy (kWh) that has the potential to be produced by the selected wind turbine type over all the modelled met lines at the current output point; **X** is the turbine hub height (m); **Turbine type** is the name of the turbine type selected.
- **CapFac|%|Hub height = Xm|Turbine type|- |1hr**: Estimated capacity factor for the selected wind turbine at the current output point, defined as the **Energy** divided by the energy that could be produced by the selected wind turbine type if it operated continuously at its maximum power output and expressed as a percentage (%); **X** is the turbine hub height (m); **Turbine type** is the name of the turbine type selected.

To produce contour plots of potential wind energy output in the **map view** follow the instructions below:

**Step 1** If necessary, activate the **map view**, for example by choosing **Plot in map view** from the **Results** menu in the **FPL data view**.

*You can choose whether to use the built-in interpolator or Golden Software's Surfer (if you have that installed). Use the **File** – **Preferences** – **Gridding option** menu to select the **Interpolator** or the **Surfer** option as desired.*

**Step 2** Click on the **Wind Energy** button. 

**Step 3** The **2-D Output Plotter** screen will appear as shown in **Figure 4.2**. You can click **Cancel** at any time to close this screen and return to the **map view**.

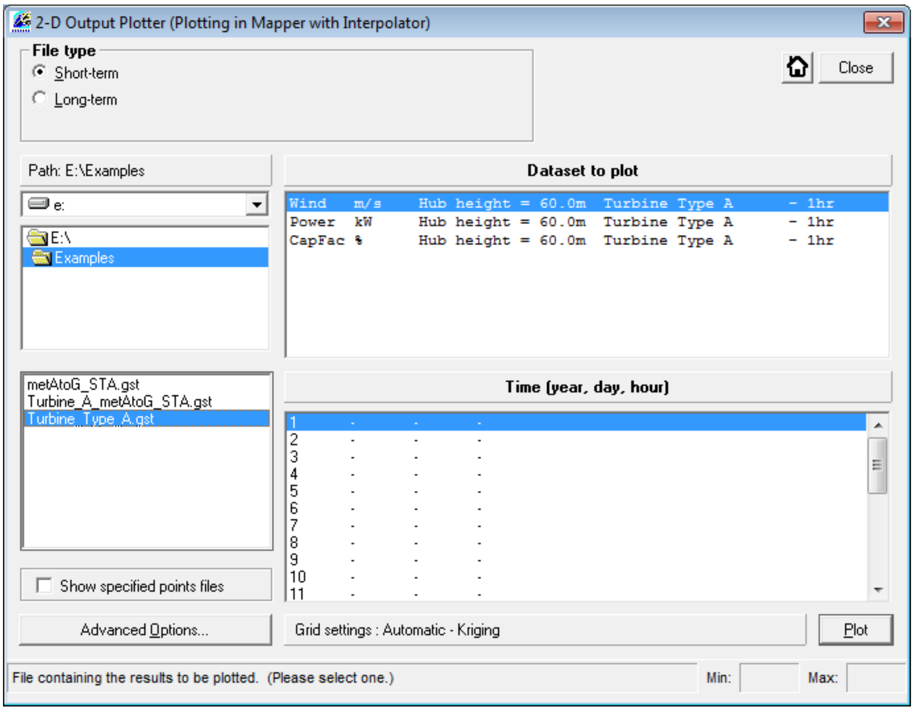



**Figure 4.2** - The **2-D Output Plotter** with Interpolator screen.

**Step 4** Choose the **File type** from **Short-term** and **Long-term** (at the top of the screen). This defines the types of files listed in the lower left hand box. If **Short-term** is chosen, all the files with the extension *.gst* (in the chosen directory) are listed, if **Long-term** is chosen, all the files with the extension *.glt* are listed.

*Checking the* ***Show specified points files*** *box also allows for specified points files (*.pst, .plt,*) to be plotted providing they contain sufficient points.*

**Step 5** Select the appropriate folder and click on the name of the file containing the data to plot. Right-clicking on **Path:** allows the path of a file or directory to be pasted in. Files or directories can also be dragged from Windows Explorer onto **Path:** to use that file or directory.

**Step 6** Click on the dataset to plot (**Dataset to Plot** box), and if relevant also click on the time for which the data are to be plotted (**Time (year, day, hour)** box).

The **Dataset to Plot** box shows a list of all the variables that can be used for the plot. Each record in the list shows the type of output (e.g. 'Wind' for wind speed), the units of output (e.g. 'm/s'), the hub height, the turbine type and the averaging time (e.g. '1hr').

For short-term runs, a **Time (year, day, hour)** box is also displayed. The box displays the time (year, day, hour) as given in the meteorological data. If no time information is given in the meteorological data file, numbers corresponding to each consecutive meteorological condition are used. For example, numbers 1 to 7, corresponding to meteorological conditions A to G respectively, are listed when the example file *R91a-g.met* is used.

**Step 7** If required, use the **Advanced Options...** button to change the number of grid lines or to specify user defined contour levels, as shown in **Figure 4.3**. The **Gridding method** can also be changed. The options available with the

**Interpolator** option are **Kriging, Kriging (all points)** and **Inverse distance** (additional options are available with Surfer). The appearance of the output contours will depend on which of these has been chosen. **Kriging** is a suitable choice in most instances. Once advanced options have been selected, click **Close** to return to the main **2-D Output Plotter** screen.

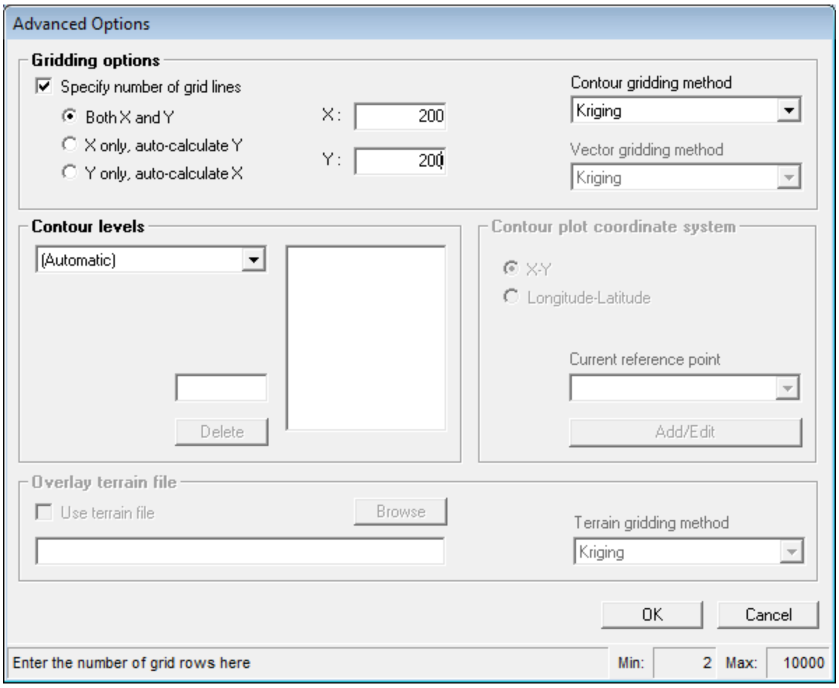



**Figure 4.3** - The **Advanced Options** screen for the **2-D Output Plotter** with Interpolator.

**Step 8** Click on the **Plot** button.

**Step 9** You are asked to enter a name for the grid file (*.grd* file). This file contains the converted FLOWSTAR-Energy flow field output in suitable format for the **map view**. You are advised to save this in the directory where the *.fpl* file is located. After entering a file name click on the **Save** button.

**Step 10** The contour plot will appear in the **map view**.

**Figure 4.4** shows an example of a wind power contour plot in the **map view**.

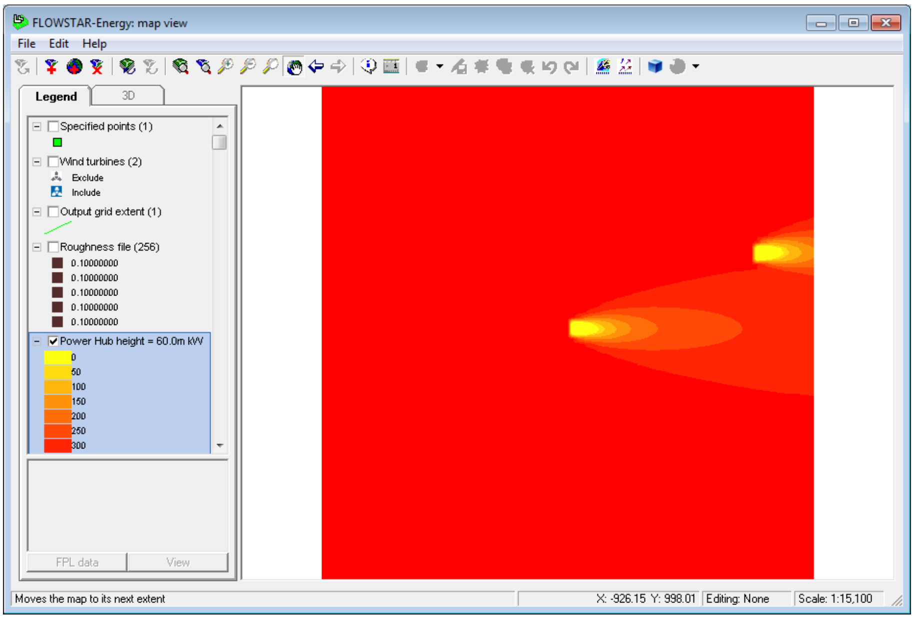



**Figure 4.4** – Example wind power contour plot.

### *Modifying the appearance of a contour layer in the map view*

The colour scheme and number of contour levels used for a contour plot can be changed either manually, by altering each of the individual levels; or by applying a new colour scheme; or by importing a previously-exported colour scheme.

The contour levels can be manually altered by double clicking on the appropriate contour layer in the **map view Legend** to open up the layer properties window and then selecting the **Grid** tab, as shown in **Figure 4.5**.

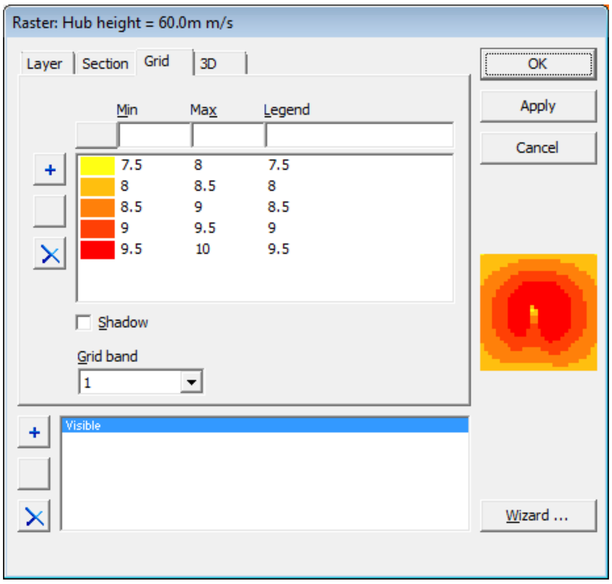



**Figure 4.5** - The **Grid** tab for the layer properties screen for a contour layer.

To select a contour level click on it; the following options are then available:

- Add a new contour level with the same properties as the current selected contour level just above the current contour level by clicking on the **Plus** button next to the contour levels.
- Delete the current selected contour level by clicking on the **Minus** button next to the contour levels.
- Delete all of the contour levels by clicking on the **Cross** button next to the contour levels.
- Alter the colour of the contour level by clicking on the **Colour** button and selecting a new colour.
- Alter the **Min**, **Max** or **Legend** values of the contour level by editing the contents of the text boxes.

When the changes have been made, click on the **OK** button to make the changes and go back to the **map view**, click the **Apply** button to make the changes but remain in the layer properties window, or click on the **Cancel** button to discard the changes and return to the **map view**. The changes can be saved permanently to disk so that the contour plot will have the same appearance if you reopen the *.grd* file; for details on how to do this by saving layer settings see Section 5.2. Layer settings can also be exported for reuse with another plot, so that it has identical colour scales.

A facility exists to automatically create a new graduated colour scheme for a contour layer. To alter the colour scheme in this way, follow these instructions:

**Step 1** Double click on the contour layer you wish to alter in the **map view Legend** to bring up the layer properties window.

**Step 2** Select the **Grid** tab, see **Figure 4.5**.

**Step 3** Click on the **Wizard...** button.

**Step 4** This brings up the **Grid Wizard** window, as shown in **Figure 4.6**.

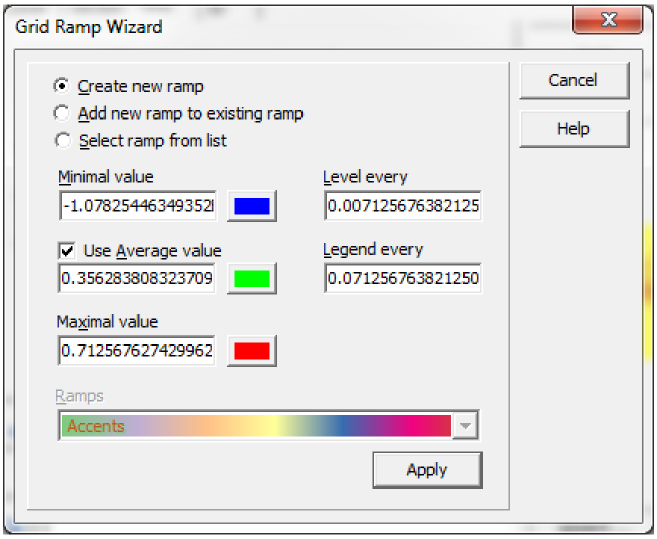



**Figure 4.6** - The **Grid Wizard** screen for creating a new contour colour scheme.

**Step 5** From here set the **Minimal** and **Maximal** values and the colours to be used to represent these.

**Step 6** If required, set a middle value and the colour to be used to represent this. If this is not required then uncheck the **Use Average value** check box.

**Step 7** Set the spacing required between colour levels, **Level Every**, and the spacing between legend entries, **Legend Every**.

**Step 8** Click on the **Apply** button.

**Step 9** Uncheck the **Shadow** checkbox on the **Grid** tab.

**Step 10** If required, manually edit contour levels.

**Step 11** Click on the **OK** button to make the changes and return to the **map view**, or click on the **Apply** button to make the changes but remain in the layer properties window, or click on the **Cancel** button to discard the changes and return to the **map view**. The changes can be saved permanently to disk so that the contour plot will have the same appearance if you reopen the *.grd* file; for details on how to save layer settings in this way see section **5.2**. Layer settings can also be exported for reuse with another plot, so that it has identical colour scales.

*Alternatively a colour scale from an existing colour ramp can be selected using the radio button and selecting the ramp from the drop-down menu.*

### 4.1.2 Flow field plotting in the map view

The flow field output files contain the wind and turbulence fields:

- for each line of meteorological data if short-term output is selected.
- as averages of these quantities over all meteorological data lines if long-term is selected.

#### *Flow field output: .W01 files*

The *.W01*, *.W02*, etc, files are only created for short-term calculations with an output grid if flow field output is selected. The *.W01*, *.W02*, etc. files contain flow field data at the locations defined for the output grid. These files are only created for the first 24 lines of met. data.

Variables are:

- **X(m)**: X coordinate of output point
- **Y(m)**: Y coordinate of output point
- **Z(m)**: Height above terrain of output point
- **U(m/s)**: Component of wind velocity in west-east direction
- **V(m/s)**: Component of wind velocity in south-north direction
- **W(m/s)**: Vertical component of wind velocity (relative to sea level)
- **Ux(m/s)**: Longitudinal component of wind velocity (i.e. in the direction of the free

stream wind; the direction specified for the wind in the meteorological data)

- **Uy(m/s)**: Transverse component of wind velocity (i.e. perpendicular to the direction of the free stream wind)
- **Angle(deg)**: Wind direction (degrees measured anticlockwise, $0^o$ = wind from west)
- **Magnitude of horizontal wind (m/s)**: Magnitude of horizontal wind velocity ($=\sqrt{U^2+V^2}$)

### *Flow field output: .T01 files*

The *.T01*, *.T02*, etc, files are only created for short-term calculations with an output grid if flow field output is selected. The *.T01*, *.T02*, etc. files contain turbulence data at the locations defined for the output grid. These files are only created for the first 24 lines of met. data.

Variables are:

- **X(m)**: X coordinate of output point
- **Y(m)**: Y coordinate of output point
- **Z(m)**: Height above terrain of output point
- **Sig-U(m/s)**: Longitudinal component of turbulent velocity $\sigma_u$
- **Sig-V(m/s)**: Transverse component of turbulent velocity $\sigma_v$
- **Sig-W(m/s)**: Vertical component of turbulent velocity $\sigma_w$

### *Flow field output: .wlt file*

The *.wlt* file is only created for long-term calculations with an output grid if flow field output is selected. The *.wlt* file contains flow field data, averaged over all the lines of met. data, at the locations defined for the output grid.

Variables are:

- **X(m)**: X coordinate of output point
- **Y(m)**: Y coordinate of output point
- **Z(m)**: Height above terrain of output point
- **U(m/s)**: Component of mean wind velocity in west-east direction
- **V(m/s)**: Component of mean wind velocity in south-north direction
- **W(m/s)**: Vertical component of mean wind velocity (relative to sea level)
- **Mean horizontal wind speed (m/s)**: Mean value of the horizontal wind speed ($=\overline{\sqrt{U^2+V^2}}$)
- **Magnitude of mean horizontal wind vector (m/s)**: Magnitude of the mean horizontal wind vector ($=\sqrt{\overline{U}^2+\overline{V}^2}$)
- **Angle(deg)**: Direction of the mean horizontal wind vector (degrees measured

anticlockwise, $0^o$ = wind from west).

- **Magnitude of mean horizontal wind vector (m/s)**: Magnitude of the mean horizontal wind vector (Note that this column appears twice in the file – this is for use by the flow field plotting tool described)

### *Flow field output: .tlt file*

The *.tlt* file is only created for long-term calculations with an output grid if flow field output is selected. The *.tlt* file contains turbulence data, averaged over all the lines of met. data, at the locations defined for the output grid.

Variables are:

- **X(m)**: X coordinate of output point
- **Y(m)**: Y coordinate of output point
- **Z(m)**: Height above terrain of output point
- **Sig-U(m/s)**: Mean longitudinal component of turbulent velocity $\sigma_u$
- **Sig-V(m/s)**: Mean transverse component of turbulent velocity $\sigma_v$
- **Sig-W(m/s)**: Mean vertical component of turbulent velocity $\sigma_w$

To create vector plots or contour plots of the flow field output, follow the steps below:

**Step 1** If necessary, activate the **map view**, for example by choosing **Plot in map view** from the **Results** menu in the **FPL data view**.

*You can choose whether to use the built-in interpolator or Golden Software's Surfer (if you have that installed) by using the* ***File*** *–* ***Preferences*** *–* ***Gridding option*** *menu to select the* ***Interpolator*** *or the* ***Surfer*** *option as desired.*

**Step 2** Click on the **Flow field** button in the **map view** toolbar.  The rest of this example assumes that the in-built interpolator has been selected.

**Step 3** The **2-D Output Plotter** screen will appear, as shown in **Figure 4.7**.

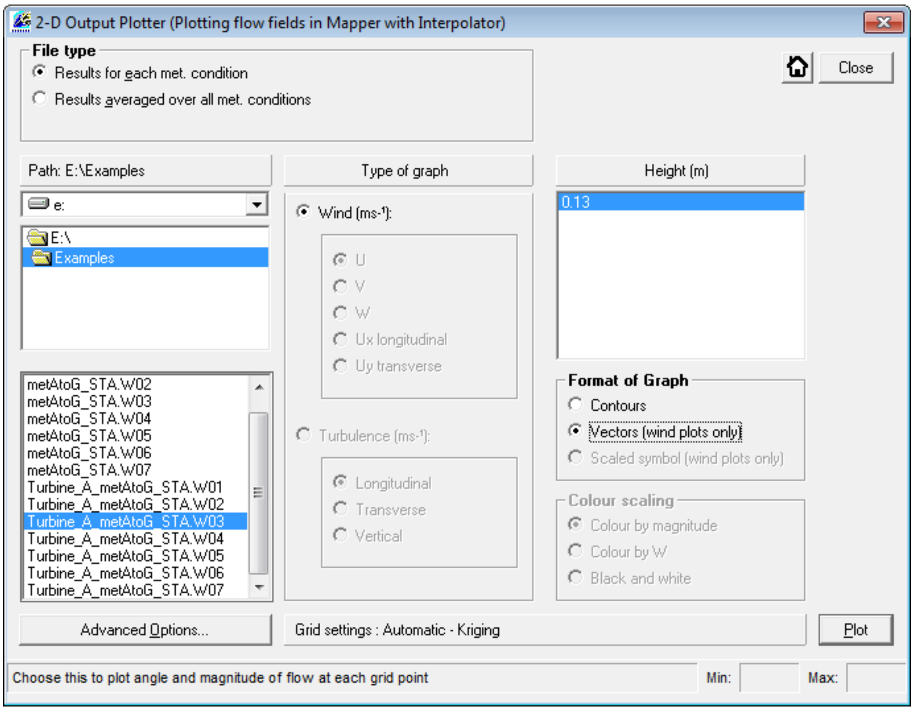



**Figure 4.7** - The **2-D Output Plotter** with Interpolator screen.

**Step 4** Select whether to plot **Results for each met. condition** (short-term) or **Results averaged over all met. conditions** (long-term).

**Step 5** In the **Type of graph** box, select the appropriate data to plot (see further on for a description of all variables available).

**Step 6** Select whether a contour plot or a vector plot is to be made, from the **Format of Graph** box. Scaled symbol plots are not available in the **map view**; consequently the associated colour scaling options are also not available.

**Step 7** From the boxes on the left, select the folder and then the file that contains the data to plot. Right-clicking on **Path:** allows the path of a file or directory to be pasted in. Files or directories can also be dragged from Explorer onto **Path:** to use that file or directory.

**Step 8** Choose the height from the **Height (m)** box.

*Note that these heights are measured from the surface of the terrain.*

**Step 9** If required, use the **Advanced Options...** button to change the number of grid lines or to specify user defined contour levels, as shown in **Figure 4.8**. The **Gridding method** can also be changed. The options available are **Kriging, Kriging (all points)** and **inverse distance weighting** and the appearance of the output contours will depend on which of these has been chosen. The **Kriging** method is a suitable choice in most instances. Once advanced options have been selected, click **OK** to return to the main **2-D Output Plotter** screen.

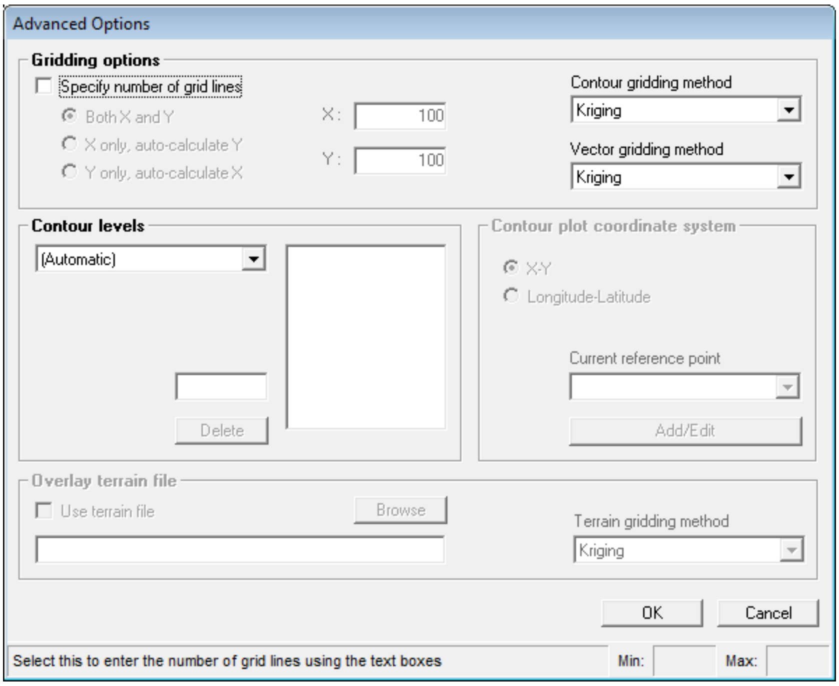



**Figure 4.8** - The **Advanced Options** screen for the FLOWSTAR-Energy **2-D Output Plotter** with Interpolator.

**Step 10** Click on the **Plot** button.

**Step 11** You are asked to enter a name for a file. This file will contain the converted FLOWSTAR-Energy output in suitable format for the **map view**: vector plots are saved in *.csv* format and contour plots are saved in *.grd* format. You are advised to save this in the directory where the *.fpl* file is located. After entering a file name click on the **Save** button.

**Step 12** The plot will appear in the **map view**.

### *Type of graph.*

This section lists all the possible variables that can plotted:

*Wind (ms$^{-1}$)*

1. **U**: component of wind velocity in west-east direction.
2. **V**: component of wind velocity in south-north direction.
3. **W**: vertical component of wind velocity (relative to sea level).
4. **Ux longitudinal**: longitudinal component of wind velocity, i.e. in the direction of the free stream wind (the wind direction specified in the meteorological data). Short-term output only.
5. **Uy transverse**: transverse component of wind velocity, i.e. perpendicular to the direction of the free stream wind. Short-term output only.
6. **Mean Speed**: Mean value of the horizontal wind speed. Long-term output only.
7. **Mean vector magnitude**: Magnitude of the mean horizontal wind vector (refer to Section 7.6 for definition). Long-term output only.

*Turbulence ($ms^{-1}$)*

1. **Longitudinal**: longitudinal component of turbulent velocity ($\sigma_u$).
2. **Transverse**: transverse component of turbulent velocity ($\sigma_v$).
3. **Vertical**: vertical component of turbulent velocity ($\sigma_w$).

Contour plots of the flow field can be customised in the same way as contour plots of potential wind energy: see Section 4.1.1 above.

### *Vector plot*

The horizontal wind, i.e. the vectors (U,V), can also be displayed as a vector plot by selecting **Wind ($ms^{-1}$)** in the **Type of graph** box and **Vectors (only for wind plots)** in the **Format of Graph** box. The **Colour scaling** box is not available when producing vector plots in the **map view.** Vector plots in the **map view** are colour scaled by default; this can be changed manually later. **Figure 4.9** shows an example of a flow field vector plot in the **map view.** The length, size and colour of the arrows depend on the magnitude of the horizontal wind. The direction of the arrows shows the direction towards which the wind is blowing.

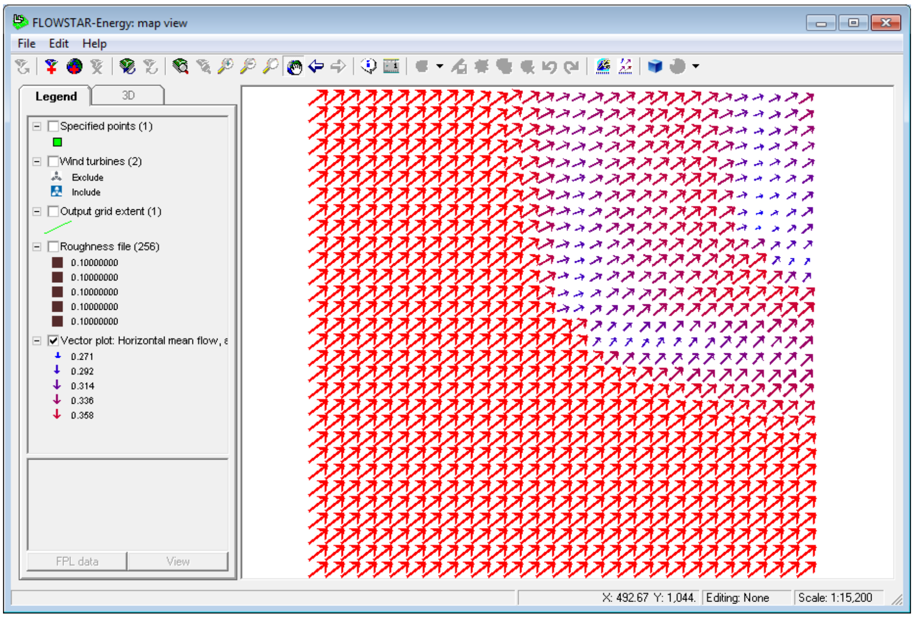



**Figure 4.9** – Example vector flow field plot.

By selecting the **Information** button, , then clicking on a vector on the plot, it is possible to display all the output data for the vector in the information panel of the **Legend** window.

### *Modifying the appearance of a flow field plot in the map view*

There are various ways in which the flow field wind vector plots may be altered visually, including changing the size, colour and frequency of the wind vector arrows. The wind vector symbol may also be changed. The steps below describe how to change all these features of the wind vector plot.

**Step 1** Double click on the appropriate wind vector layer in the **map view Legend** to open up the layer properties window and then select the **Renderer** tab with the **First** tab selected, as shown in **Figure 4.10**.

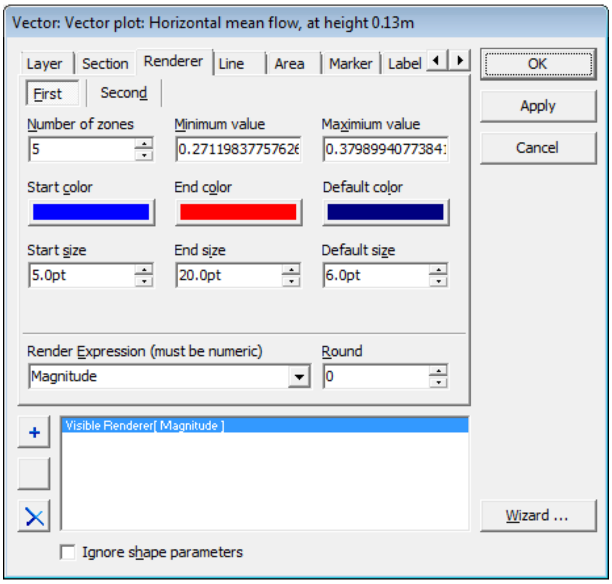



**Figure 4.10** – The **Renderer** tab of the layer properties window

**Step 2** In order to change the number of horizontal wind speed levels in the plot the integer given under **Number of Zones** can be changed. The default value is 5.

**Step 3** The colour of the wind vector arrows relates to the magnitude of the horizontal wind at each point. The colours are changed by altering **Start color** and **End color** in the **Renderer** tab. The **Minimum value** and **Maximum value** specify the horizontal wind magnitude that is used for the **Start color** and **End color**. The default **Minimum value** and **Maximum value** are chosen so that arrows will be displayed at every point; it is possible to hide the arrows at points where the horizontal speed is outside a certain range by altering the **Minimum value** and **Maximum value**, and then changing the **Default color** to clear as shown in **Figure 4.11**.

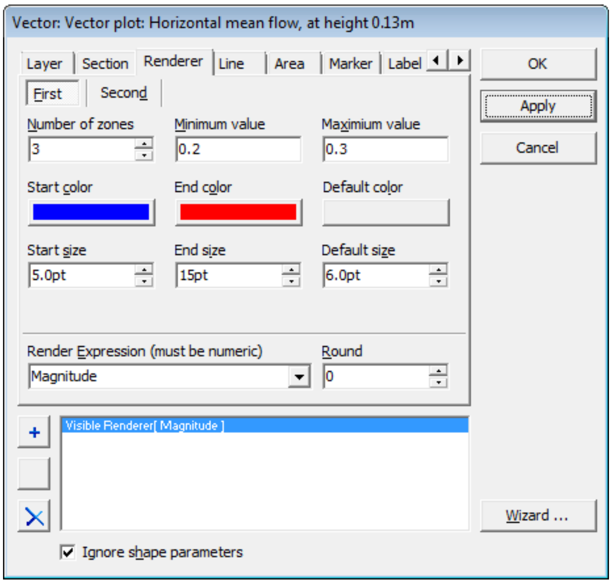



**Figure 4.11** – Changing the default colour in the **Renderer** tab

**Step 4** The length and size of the wind vector arrows also relates to the magnitude of the horizontal wind at each point. As for the colours, this can be controlled in the **Renderer** tab of the layer properties window; the **Start size** and **End size** values can be edited and the **Apply** button then pressed to display the change in the **map view** window without closing the layer properties window.

**Step 5** If it is considered that the wind vector plot contains too much detail, it is possible to reduce the number of vector arrows shown on the plot. To do this, go to the **Section** tab of the layer properties window. Under the box labelled **Render if match query (must be logical)** use the drop down box to select **GIS_UID** and then type **%X=0**, where X is an integer value. If X=1, then every arrow will still be present on the plot, if X=2 then only every second vector will be shown and so on. An example where every fourth vector will be shown is given in **Figure 4.12**.

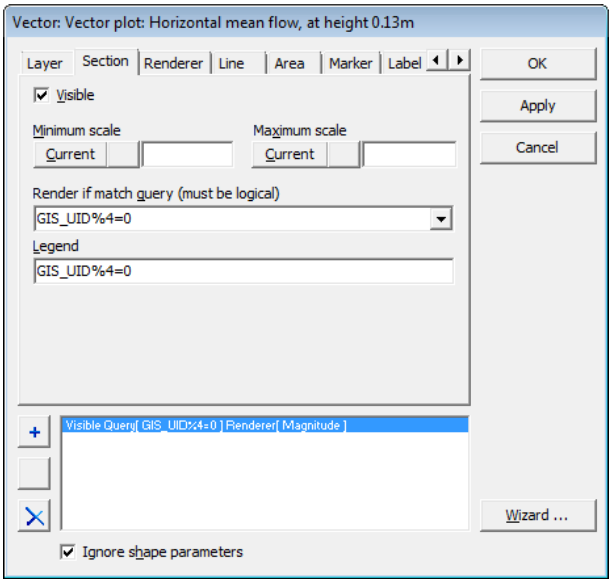



**Figure 4.12** – changing the frequency of vectors in the **Section** tab

**Step 6** The symbol used for the vector arrow may be changed. The symbol is chosen in the **Marker** tab of the layer properties window, shown in **Figure 4.13**.

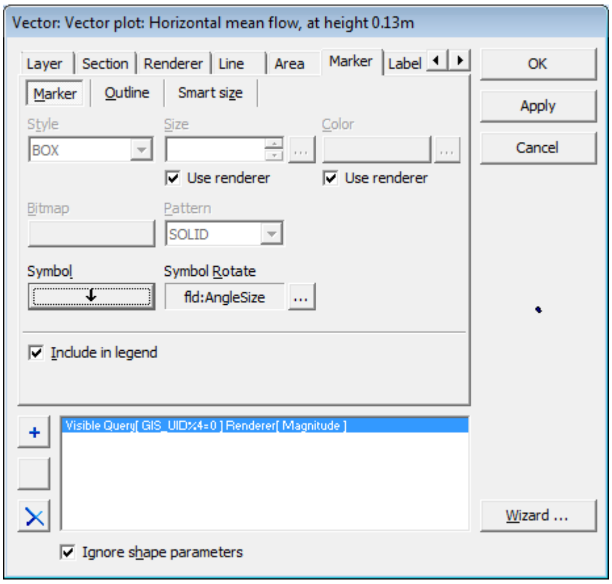



**Figure 4.13** – changing the vector symbol in the **Marker** tab

Once the **Symbol** button is pressed the **File symbol** window shown in **Figure 4.14** is opened.

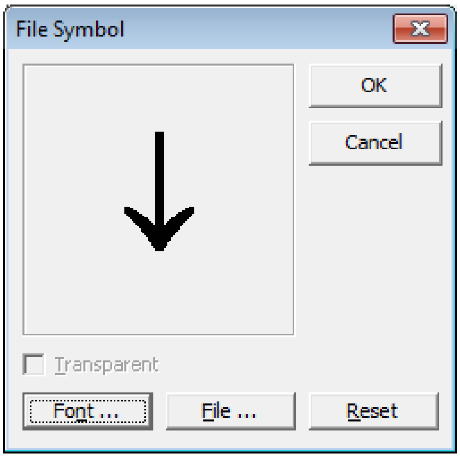



**Figure 4.14** – the **File Symbol** window

The symbol is changed by pressing the **Font** button and selecting a new symbol from those available in the **Font Symbol** window. It is crucial to choose a new symbol which points in a downwards direction, such as the V symbol shown in **Figure 4.15**, as otherwise the vector arrows will point in an incorrect direction.

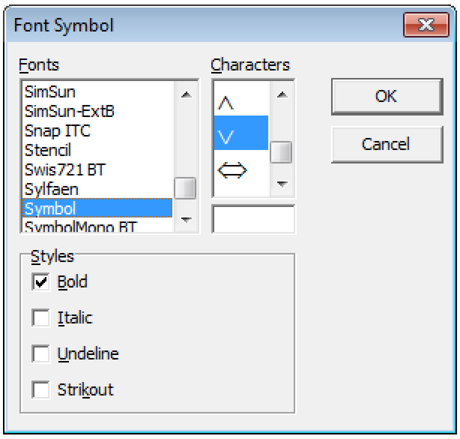



**Figure 4.15** – the **Font Symbol** window

**Step 7** Once all the changes have been made to the appearance of the flow field plots, the new settings of the plot may be saved permanently to disk so that the plot will have the same appearance if you reopen this **map view** flow field *.csv* file. To save the changes to disk, right click on the appropriate wind vector layer in the **Legend** to bring up the menu shown in **Figure 4.16**.

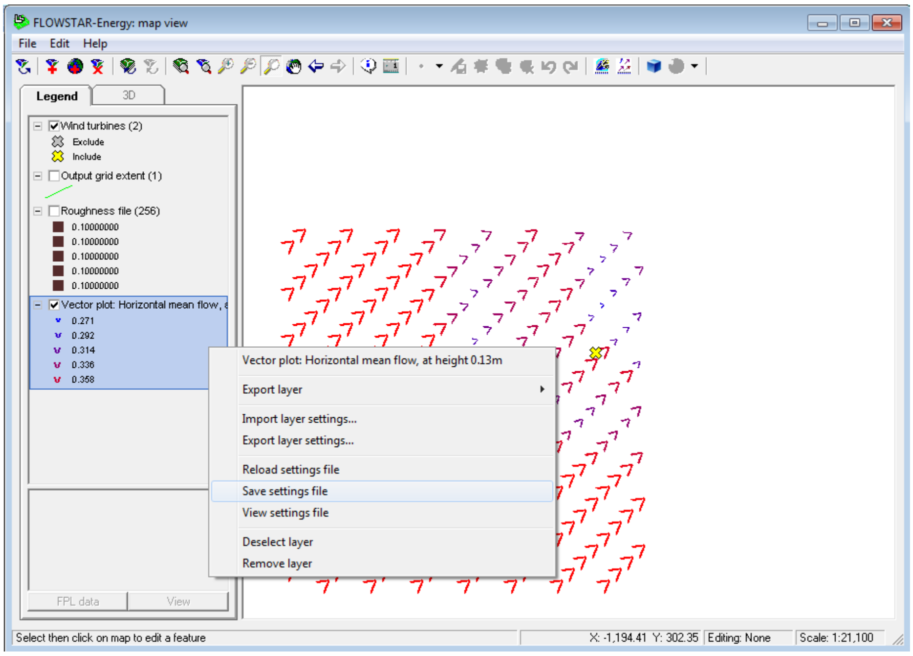



**Figure 4.16** – the options available from right clicking the vector layer

Choose **Save settings file** from this menu. Once this has been done, whenever the flow field vector plot is added to a new **map view**, the symbology currently being used will be loaded with the file. The saved flow field vector plot is in a comma delimited (*.csv*) format. Previously saved flow field plots can be added to the **map view** by selecting the **Add layer** tool  at the top of the map view, then selecting the appropriate *.csv* file. The screen shown in **Figure 4.23** will appear. Press the **OK** button to load the flow field plot into the **map view** window.

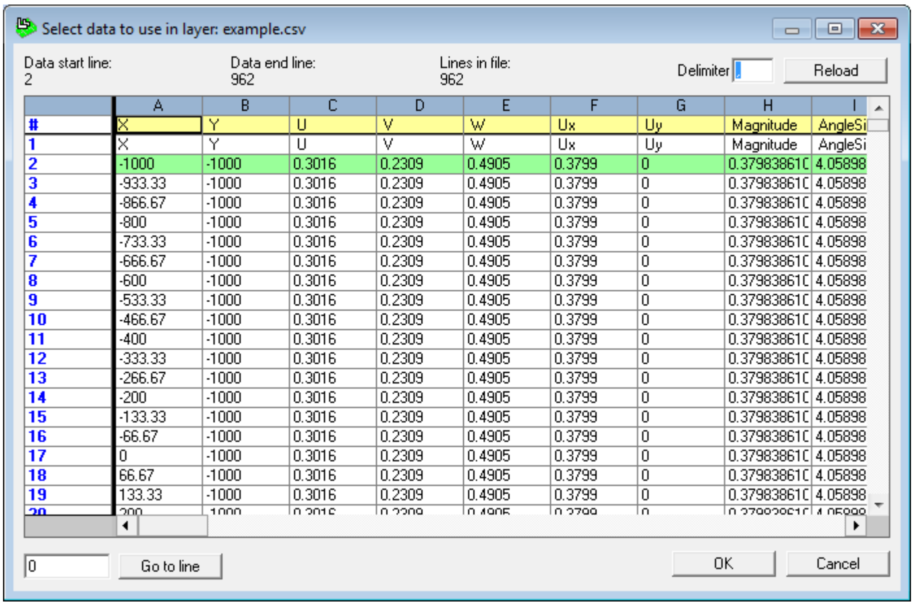



**Figure 4.17** – *.csv* file import

## 4.2 Plot in Surfer

If Surfer is installed, the user can create visualisations directly in Surfer rather than by using the FLOWSTAR-Energy **map view**. This section explains how to produce contour and vector plots in Surfer in this way, using the **2-D Output Plotter** utility launched from the **FPL data view**. It is also possible to produce contour and vector plots in the **map view** via Surfer, as described in Section 4.1 above.

To launch the **2-D Output Plotter (Plotting in Surfer)** utility, select **Results**, **Plot in Surfer...** from the menu at the top of the **FPL data view**. The **Plot in Surfer...** menu item will be disabled (greyed-out) if you do not have Surfer installed.

---

*If the* **Plot in Surfer...** *menu item is disabled but you do have Surfer installed, this may be caused by a problem with the Surfer installation on your computer. Please contact CERC for assistance.*

---

The **Results**, **Plot in Surfer...** menu item brings up the screen shown in **Figure 4.18**.

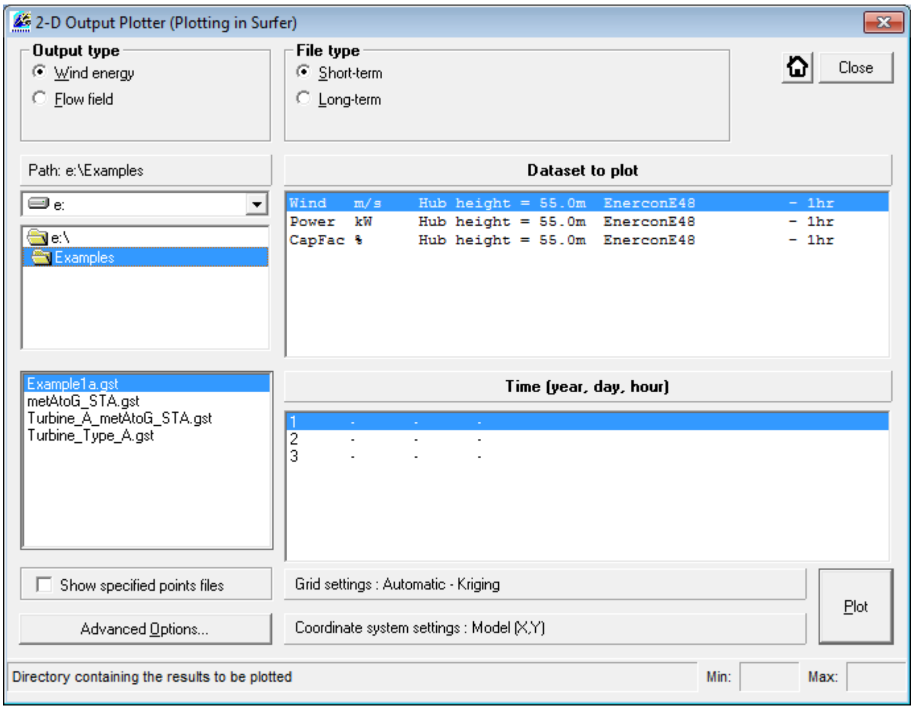



**Figure 4.18** – The **2-D Output Plotter** screen.

The **2-D Output plotter** allows for plotting different types of gridded output from FLOWSTAR-Energy. The **Output type** should first be selected:

- **Wind energy**: for plotting contour plots of gridded potential wind energy output including: wind speed, potential wind energy and estimated capacity factor. Specified point output can also be plotted if there are enough specified points in the results file.
- **Flow field**: for plotting contour or vector plots of flow field output, including wind

speed and turbulence.

### 4.2.1 Potential wind energy plots in Surfer without the map view

The potential wind energy output files are described in Section 4.1.2.

When **Wind energy** is selected as the **Output type** in the **2-D Output Plotter** will look as shown in **Figure 4.18**. Here are the steps to follow in order to create a contour plot of data:

**Step 1** Choose the file type from **Short-term** and **Long-term** (at the top of the screen). This defines the types of files listed in the lower left hand box. If **Short-term** is chosen, all the files with the extension *.gst* (in the chosen directory) are listed, if **Long-term** is chosen, all the files with the extension *.glt* are listed.

*Checking the* ***Show specified points files*** *box also allows for specified points files (*.pst, .plt,*) to be plotted providing they contain sufficient points.*

**Step 2** Select the appropriate folder and click on the name of the file containing the data to plot. Right-clicking on **Path:** allows the path of a file or directory to be pasted in. Files or directories can also be dragged from Explorer onto **Path:** to use that file or directory.

**Step 3** Click on the dataset to plot (**Dataset to Plot** box), and if relevant also click on the time for which the data are to be plotted (**Time (year, day, hour)** box).

The **Dataset to Plot** box shows a list of all the variables that can be used for the plot. Each record in the list shows the type of output (e.g. 'Wind' for wind speed), the units of output (e.g. 'm/s'), the hub height, the turbine type and the averaging time (e.g. '1hr').

For short-term runs only, a **Time (year, day, hour)** box is also displayed. The box displays the time (year, day, hour) as given in the meteorological data. If no time information is given in the meteorological data file, numbers corresponding to each consecutive meteorological condition are used. For example, numbers 1 to 7, corresponding to meteorological conditions A to G respectively, are listed when the example file *R91a-g.met* is used.

**Step 4** Click on the **Advanced Options...** button to set some properties of the plot (optional – refer to Section 4.2.3 for details).

**Step 5** Click on **Plot** to plot the selected data in Surfer.

*The FLOWSTAR-Energy output file is converted to a grid file (*.grd*) for Surfer to plot. Save this in the directory where the* .fpl *file is located. Once the Surfer drawing has been saved, the* .grd *file can be deleted as it is no longer required.*

In the upper right hand corner, the **Close** button closes the **2-D Output Plotter** and the  button brings the **FPL data view** to the front without closing the utility.

### 4.2.2 Flow field plots in Surfer without the map view

The flow field output files are described in Section 4.1.2.

The results in these files can be plotted by selecting **Flow field** as the **Output type** the **2-D Output Plotter**, as shown in **Figure 4.19**.

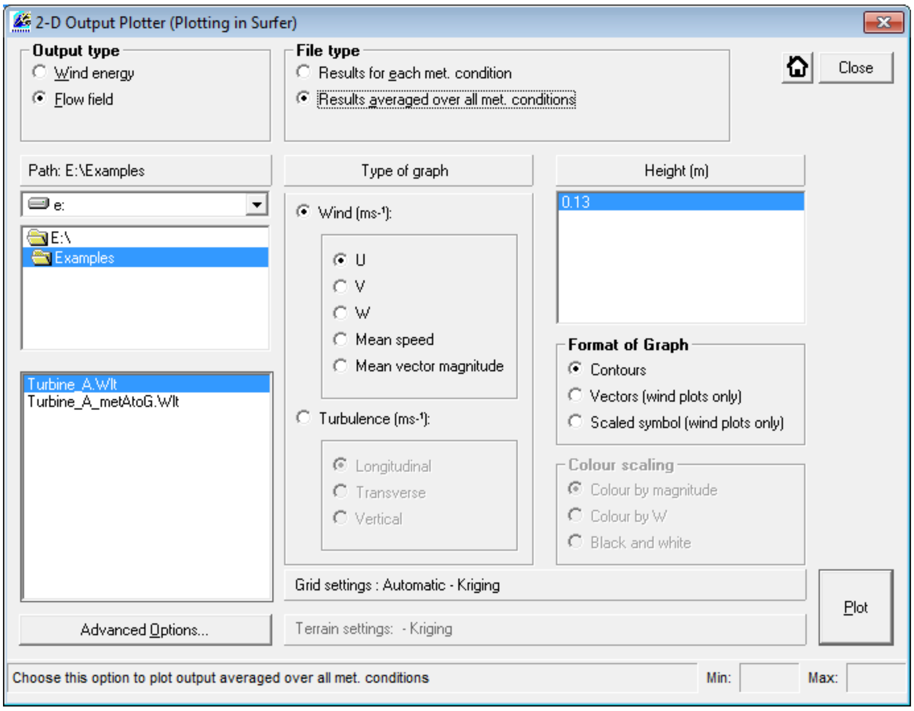



**Figure 4.19** – The **2-D Output plotter** utility with **Flow field** selected.

In order to create a flow field plot, proceed as follows:

**Step 1** Select whether to plot **Results for each met. condition** (short-term) or **Results averaged over all met. conditions** (long-term).

**Step 2** In the **Type of graph** box, select the appropriate data to plot (see further on for a description of all variables available).

**Step 3** Select whether a contour plot, a vector plot or a scaled symbol plot is to be made, from the **Format of Graph** box.

**Step 4** If making a **Vectors (wind plots only)** plot select the colour scaling to use from the **Colour scaling** box.

**Step 5** From the boxes on the left, select the folder and then the file that contains the data to plot. Right-clicking on **Path:** allows the path of a file or directory to be pasted in. Files or directories can also be dragged from Explorer onto **Path:** to use that file or directory.

**Step 6** Choose the height from the **Height (m)** box.

*Note that these heights are measured from the surface of the terrain.*

**Step 7** Click on **Plot** to display the plot in Surfer.

An example of a contour plot is shown in **Figure 4.20** and an example of a vector plot in **Figure 4.21**.

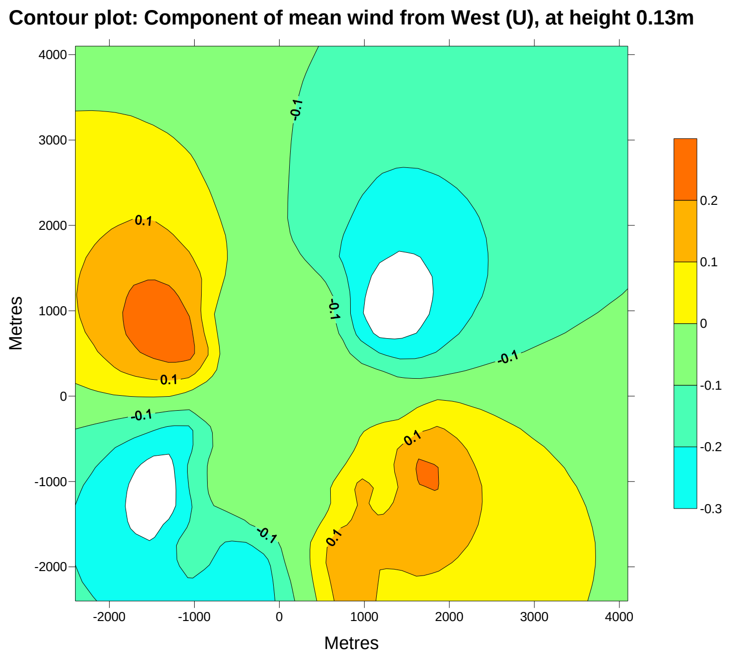



**Figure 4.20** − Example flow field contour plot.

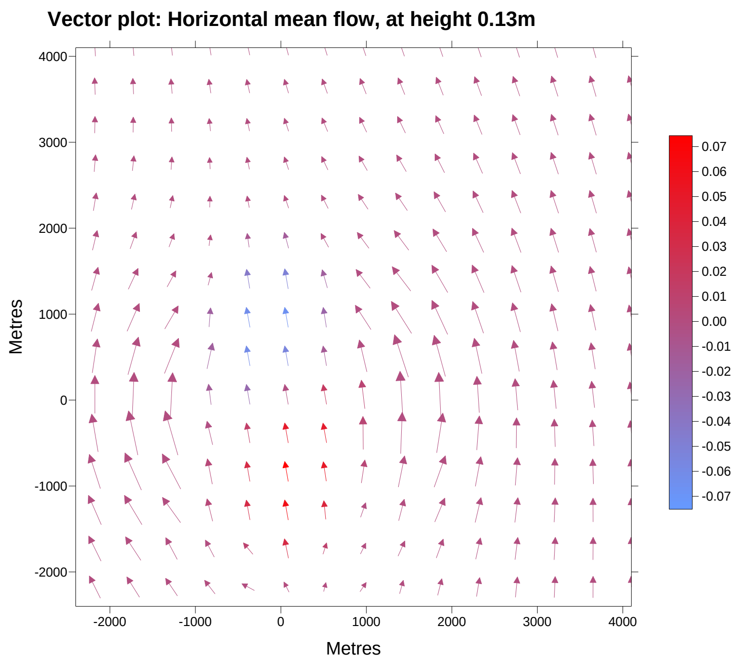



**Figure 4.21** − Example flow field vector plot with the colour scaled by vertical velocity.

### *Type of graph*

This section lists all the possible variables that can plotted:

*Wind ($ms^{-1}$)*

1. **U**: component of wind velocity in west-east direction.
2. **V**: component of wind velocity in south-north direction.
3. **W**: vertical component of wind velocity (relative to sea level).
4. **Ux longitudinal**: longitudinal component of wind velocity, i.e. in the direction of the free stream wind (the wind direction specified in the meteorological data). Short-term output only.
5. **Uy transverse**: transverse component of wind velocity, i.e. perpendicular to the direction of the free stream wind. Short-term output only.
6. **Mean Speed**: Mean value of the horizontal wind speed. Long-term output only.
7. **Mean vector magnitude**: Magnitude of the mean horizontal wind vector (refer to Section 7.6 for definition). Long-term output only.

*Turbulence ($ms^{-1}$)*

1. **Longitudinal**: longitudinal component of turbulent velocity ($\sigma_u$).
2. **Transverse**: transverse component of turbulent velocity ($\sigma_v$).
3. **Vertical**: vertical component of turbulent velocity ($\sigma_w$).

### *Vector plot*

The horizontal wind, i.e. the vectors (U,V), can also be displayed as a vector plot by selecting **Wind ($ms^{-1}$)** in the **Type of graph** box and **Vectors (only for wind plots)** in the **Format of Graph** box. A choice of colour scaling is then available in the **Colour scaling** box:

- **Colour by magnitude**: the vectors are coloured by the horizontal wind speed
- **Colour by W**: the vectors are coloured by the vertical velocity
- **Black and white**: the vectors are not coloured

An example of a vector plot is shown in **Figure 4.21**.

### *Scaled symbol plot*

The horizontal wind, i.e. the vectors (U,V), can also be displayed as a scaled symbol plot by selecting **Wind ($ms^{-1}$)** in the **Type of graph** box and **Scaled symbol (only for wind plots)** in the **Format of Graph** box. The scaled symbol plot is similar to the vector plot but plots arrows at each of the locations in the output file rather than gridding the data as is carried out in the vector plot. An example plot is shown in **Figure 4.22**

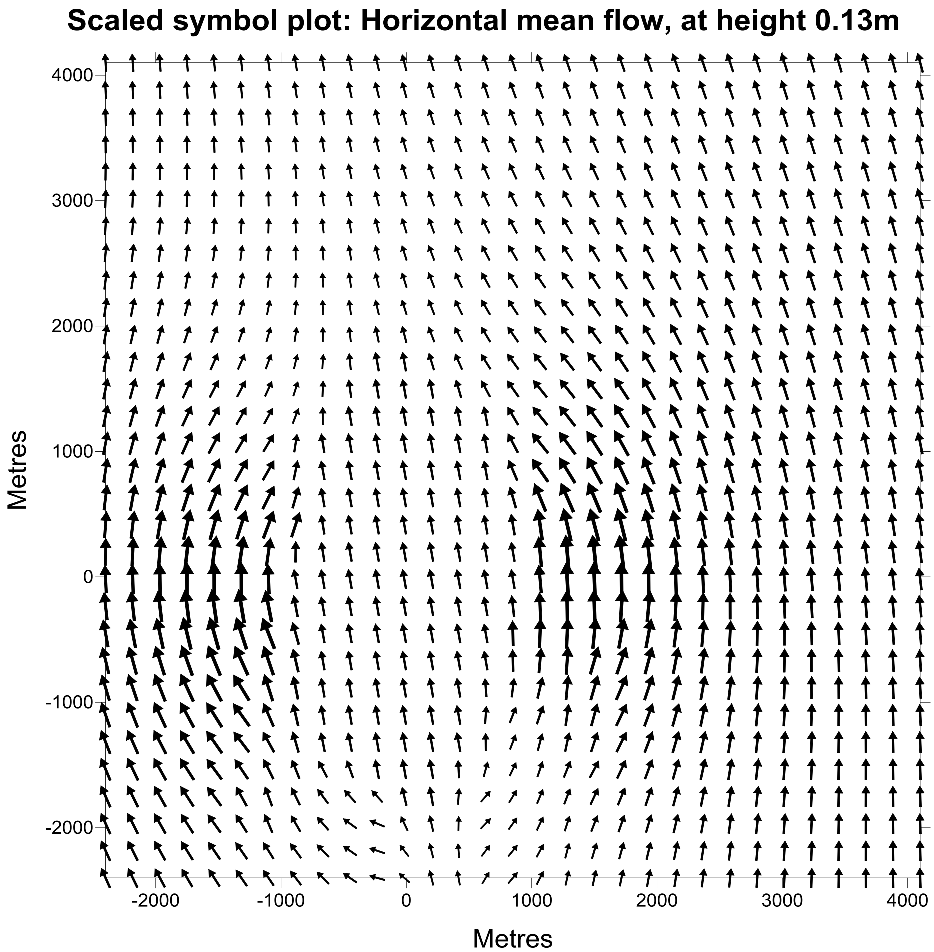



**Figure 4.22** − Example scaled symbol plot.

## 4.2.3 Advanced options

The **Advanced Options...** button gives the user a number of options to enhance the plot. Clicking on it brings up the **Advanced Options** screen shown in **Figure 4.23**. From that screen, you can set the contour levels, the coordinate system of the grid, the Surfer gridding options, and whether to overlay the terrain file.

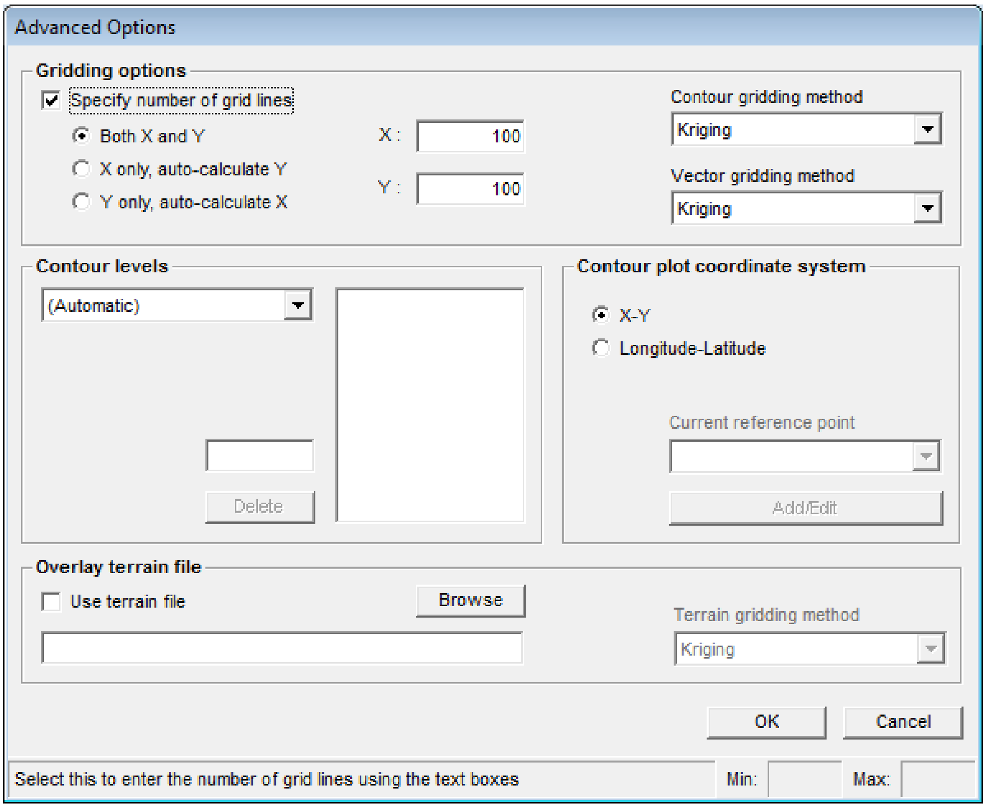



**Figure 4.23** – The **Advanced Options** screen.

When you have finished with the **Advanced Options** screen, click on the **OK** button to return to the **2-D Output Plotter (Plotting in Surfer)** main screen. Once the required information has been selected in this screen, click on **Plot** to create the plot.

*Some of the options on the **Advanced Options** screen are only available for some plot types.*

### *Gridding options*

A number of different gridding methods for a contour plot can be selected from the **Contour Gridding method** drop-down list. Information about the available gridding methods can be found in the Surfer user guide.

To specify the grid resolution, tick the **Specify number of grid lines** box. Select the appropriate option to specify the number of grid lines in both the X and Y directions, or in one direction only. If the number of lines is specified in one direction only, the contouring interface will ensure that the resolution is the same in the X and Y directions. Enter the number of grid lines in the **X:** and/or **Y:** boxes.

*If the **Specify number of grid lines** option is not selected, the **2-D Output Plotter** will choose the resolution based on the resolution of the results to be plotted.*

### *Contour levels*

To choose the contour levels to plot, click on **(User-specified)** from the drop-down list. For each level you want to plot, type the level value in the small box above the **Delete** button and press the space bar to enter it into the larger box. If you want to delete any level, click on that number in the larger box and click on the **Delete** button.

### *Contour plot coordinate system*

To convert your output from X-Y coordinates to longitude-latitude coordinates, select **Longitude-Latitude** and then click on **Add/Edit reference points**. You must supply the X-Y and longitude-latitude coordinates of a reference point, as shown in **Figure 4.24** (left). Longitude and latitude should typically be given to 6 decimal places.

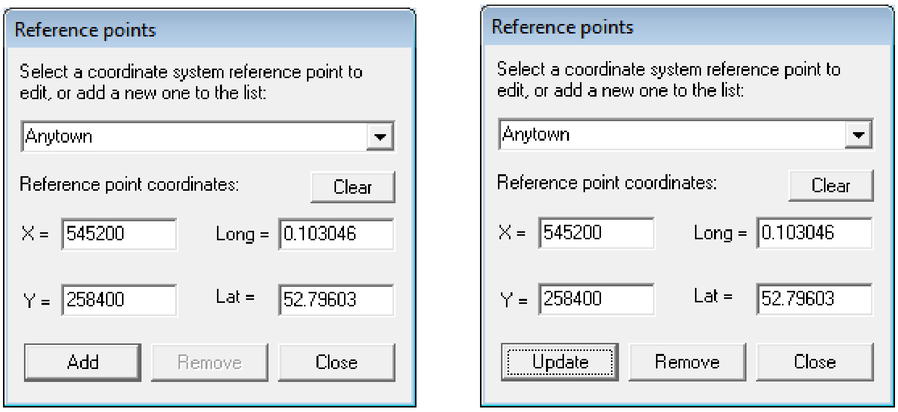



**Figure 4.24** − Left: entering coordinates of a datum point. Right: revising and updating coordinates of a datum point.

*In the UK, the X-Y coordinates will typically be national grid (OSGB) references.*

Click on **Add** to store the coordinate data and on **Close** to leave the screen. Datum information can be deleted by clicking on **Remove** or revised by entering the new data and then clicking on **Update** as in **Figure 4.24** (right).

If you have entered several datum points, because you are studying sites in different locations that use different local co-ordinate systems, choose the appropriate datum point from the drop down list.

### *Overlay terrain file*

If the effects of hills have been included in the model run, it may be appropriate to overlay the plot on the terrain data. To do this, check the **Use terrain file** box, then click on the **Browse** button to locate the *.ter* file. The gridding method for the terrain plot can be chosen from the **Terrain gridding method** list. You should ensure that the terrain file is the right one for the plot being created, i.e. the plot is contained within the terrain contours.

The **Grid settings** and the **Coordinate system** or **Terrain settings** selected on the **Advanced Options** screen are displayed at the bottom of the main screen, as shown at the bottom of **Figure 4.19**.

## 4.3 Viewing short term results

The numerical output files created by FLOWSTAR-Energy are mainly comma-separated text files. Input and output files can be accessed from the **FPL data view** and viewed in WordPad, Notepad, Microsoft Excel or some other application of the user's choice. The application used to view files is set in the **Preferences** menu (see **Figure 4.25**).

To change the preferred viewing application for input data and output files, select **File**, **Preferences**... from the menu bar of the **FPL data view**. This will bring up the **Preferences** screen shown in **Figure 4.25**.

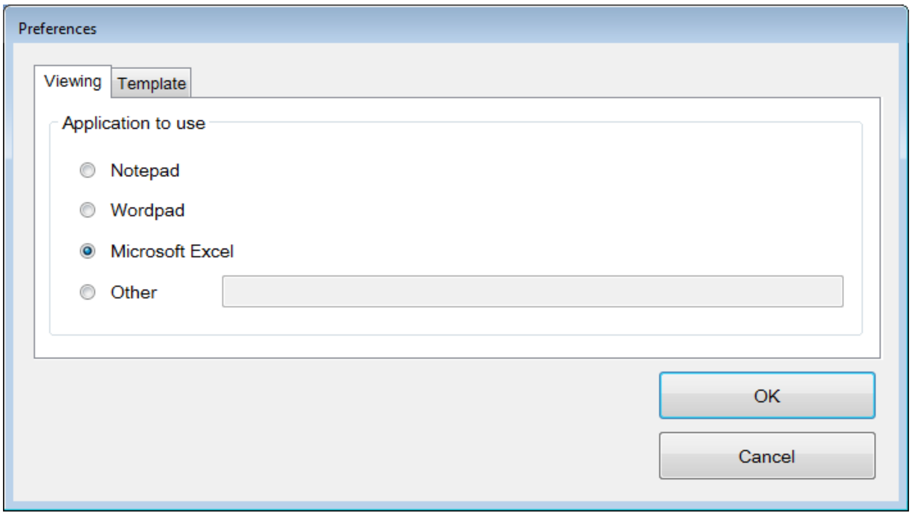



**Figure 4.25** – The **File Viewing Preferences** window.

**Viewing** allows the user to select which application to use to **View** files. If an application other than Notepad, WordPad or Microsoft Excel is to be used, then select **Other** and enter the path of the application.

---

*If the file is being opened in Microsoft Excel, it will automatically be opened as a comma-separated file.*

*If the column headings are not visible in Microsoft Excel because the columns are too narrow, select all the columns containing data and click on the* ***Format*** *button in the* ***Cells*** *section of the* ***Home*** *tab on the ribbon and select* ***Autofit Column Width*** *to increase the column widths to fit the longest entry in each column.*

---

### 4.3.1 View short-term flow fields at specified points

The **Results** menu option **View short-term flow fields at specified points** will open a *.zst* file for the current *.fpl* file. The *.zst* file is only created for short-term calculations with specified points if flow field output is selected. The *.zst* file contains flow field and turbulence data at the specified points for each line of met. data.

Variables are:

- **Year**: Year of the current line of met. data
- **Day**: Julian day number of the current line of met. data
- **Hour**: Hour of the current line of met. data
- **Receptor name**: Name of the specified point
- **X(m)**: X coordinate of specified point
- **Y(m)**: Y coordinate of specified point
- **Z(m)**: Height above terrain of specified point
- **U(m/s)**: Component of wind velocity in west-east direction
- **V(m/s)**: Component of wind velocity in south-north direction
- **W(m/s)**: Vertical component of wind velocity (relative to sea level)
- **Ux(m/s)**: Longitudinal component of wind velocity (i.e. in the direction of the free stream wind; the direction specified for the wind in the meteorological data)
- **Uy(m/s)**: Transverse component of wind velocity (i.e. perpendicular to the direction of the free stream wind)
- **Angle(deg)**: Wind direction (degrees measured anticlockwise, $0^o$ = wind from west)
- **Magnitude of horizontal wind (m/s)**: Magnitude of horizontal wind velocity ($=\sqrt{U^2+V^2}$)
- **Sig-U(m/s)**: Longitudinal component of turbulent velocity $\sigma_u$
- **Sig-V(m/s)**: Transverse component of turbulent velocity $\sigma_v$
- **Sig-W(m/s)**: Vertical component of turbulent velocity $\sigma_w$

### 4.3.2 View short-term potential wind power at specified points

The **Results** menu option **View short-term potential wind power at specified points** will open a *.pst* file for the current *.fpl* file. The *.pst* file (points, short-term) contains output values resulting from short-term calculations at every specified point and for each line of meteorological data. The output variables (other than the year, day and hour variables, which have been defined in Section 4.3.1) are listed in **Table 4.1**.

| Variable | Description |
|---|---|
| Receptor name | Specified point name |
| X(m) | X coordinate of specified point in metres |
| Y(m) | Y coordinate of specified point in metres |
| Z(m) | Redundant variable |
| Wind\|m/s\|Hub height = Xm\|Y\|- \|1hr | Wind speed at hub height (m/s) |
| Power\|kW\|Hub height = Xm\|Y\|- \|1hr | Potential wind power at hub height (kW) |
| CapFac\|%\|Hub height = Xm\|Y\|- \|1hr | Estimated capacity factor (%); the potential wind power expressed as a percentage of the power that could be produced if the turbine generated power at its maximum capacity |

**Table 4.1** − Variables contained in a *.pst* file (X represents the hub height of Turbine Y)

### 4.3.3 View short-term wind farm results

The **Results** menu option **View short-term wind farm results** will open a *.wfs* file for the current *.fpl* file. The *.wfs* file (wind farm, short-term) contains output values resulting from short-term calculations at each turbine and for each line of meteorological data. The output variables are listed in **Table 4.2**.

| Variable | Description |
|---|---|
| Turbine Name | Name of wind turbine |
| Turbine Type | Name of wind turbine type |
| X(m) | X coordinate of turbine in metres |
| Y(m) | Y coordinate of turbine in metres |
| Hub height (m) | Height of hub height in metres |
| Free stream wind speed (m/s) | Wind speed at the turbine location at hub height ignoring the effect of complex terrain and wind turbine wakes on the flow field |
| Gross wind speed (m/s) | Wind speed at the turbine location at hub height including the effect of complex terrain on the flow field but ignoring the effect of wind turbine wakes |
| Net wind speed (m/s) | Wind speed at the turbine location at hub height accounting for both the effect of complex terrain and the effect of wind turbine wakes |
| Net wind speed deficit (%) | 100 x (Gross wind speed – Net wind speed) / Gross wind speed |
| Free stream turbulence intensity (%) | Turbulence intensity at the turbine location at hub height ignoring the effect of complex terrain and wind turbine wakes on the flow field |
| Gross turbulence intensity (%) | Turbulence intensity at the turbine location at hub height including the effect of complex terrain on the flow field but ignoring the effect of wind turbine wakes |
| Net turbulence intensity (%) | Turbulence intensity at the turbine location at hub height accounting for both the effect of complex terrain and the effect of wind turbine wakes |
| Free stream power output (kW) | Power generated by the current wind turbine at the free stream wind speed |
| Gross power output (kW) | Power generated by the current wind turbine at the gross wind speed |
| Net power output (kW) | Power generated by the current wind turbine at the net wind speed |
| Topographic efficiency (%) | 100 x Gross power output / Free stream power output |
| Array efficiency (%) | 100 x Net power output / Gross power output |
| Free stream estimated capacity factor (%) | The free stream power output divided by the maximum power output possible from the current wind turbine at any wind speed, expressed as a percentage |
| Gross estimated capacity factor (%) | The gross power output divided by the maximum power output possible from the current wind turbine at any wind speed, expressed as a percentage |
| Net estimated capacity factor (%) | The net power output divided by the maximum power output possible from the current wind turbine at any wind speed, expressed as a percentage |

**Table 4.2** − Variables contained in a *.wfs* file for each modelled met line and turbine.

### 4.3.4 View short-term free stream boundary layer profiles

The **Results** menu option **View short-term free stream boundary layer profiles** will open a *.pro* file for the current *.fpl* file. The *.pro* file contains the free stream boundary layer properties output for a series of heights for each meteorological data line. The *.pro* file is produced if the **Free stream boundary layer profile output** option is enabled on the **Output types** screen. The possible output variables are listed in **Table 4.3**.

| Variable | Description |
|---|---|
| Met line no. | Met line number |
| z(m) | Height of output |
| z/h | Height of output as fraction of boundary layer height |
| U(z) (m/s) | Mean wind speed |
| DUDZ(z) ((m/s)/m) | Gradient of mean wind speed |
| SigU(z) (m/s) | Longitudinal turbulence |
| SigV(z) (m/s) | Transverse turbulence |
| SigW(z) (m/s) | Vertical turbulence |
| Rlv(z) (m) | Transverse turbulence length scale |
| Rlw(z) (m) | Vertical turbulence length scale |
| TL(z) (s) | Lagrangian time scale |
| Eps(z) (m2/s3) | Energy dissipation rate |
| Pot Temp(z) (K) | Potential temperature |
| Temp(z) (K) | Temperature |
| Press(z) (mbar) | Pressure |
| N(z) (1/s) | Buoyancy frequency |

**Table 4.3** − Variables contained in a *.pro* file.

## 4.4 Viewing long term average results

### 4.4.1 View long-term flow field at specified points

The **Results** menu option **View long-term flow field at specified points** will open a *.zlt* file for the current *.fpl* file. The *.zlt* file is only created for long-term calculations with specified points if **Flow field output** is selected on the **Output types** screen. The *.zlt* file contains flow field and turbulence data, averaged over all of the lines of met. data, at each of the specified points.

Variables are:

- **Receptor name**: Name of the specified point
- **X(m)**: X coordinate of specified point
- **Y(m)**: Y coordinate of specified point
- **Z(m)**: Height above terrain of specified point
- **U(m/s)**: Component of mean wind velocity in west-east direction
- **V(m/s)**: Component of mean wind velocity in south-north direction
- **W(m/s)**: Vertical component of mean wind velocity (relative to sea level)
- **Mean horizontal wind speed (m/s)**: Mean value of the horizontal wind speed ($=\overline{\sqrt{U^2+V^2}}$)
- **Magnitude of mean horizontal wind vector (m/s)**: Magnitude of the mean horizontal wind vector ($=\sqrt{\overline{U}^2+\overline{V}^2}$)
- **Sig-U(m/s)**: Mean longitudinal turbulence $\sigma_u$
- **Sig-V(m/s)**: Mean transverse turbulence $\sigma_v$
- **Sig-W(m/s)**: Mean vertical turbulence $\sigma_w$

### 4.4.2 View long-term potential wind energy at specified points

The **Results** menu option **View long-term potential wind energy at specified points** will open a *.plt* file for the current *.fpl* file. The *.plt* file (points, long-term) contains output values resulting from long-term calculations at every specified point averaged over all meteorological data. The output variables are listed in **Table 4.4**.

| Variable | Description |
|---|---|
| Receptor name | Specified point name |
| X(m) | X coordinate of specified point in metres |
| Y(m) | Y coordinate of specified point in metres |
| Z(m) | Redundant parameter |
| Wind\|m/s\|Hub height = Xm\|Y\|- \|1hr | Average wind speed at hub height (m/s) |
| Energy\|kWh\|Hub height = Xm\|Y\|- \|1hr | Total potential wind energy (kWh) |
| CapFac\|%\|Hub height = Xm\|Y\|- \|1hr | Estimated capacity factor (%); the total potential wind energy expressed as a percentage of the total energy that could be produced if the turbine continuously generated power at its maximum capacity |

**Table 4.4** − Variables contained in a *.pst* file (X represents the hub height of Turbine Y).

### 4.4.3 View long-term wind farm results

The **Results** menu option **View long-term wind farm results** will open a *.wfl* file for the current *.fpl* file. The *.wfl* file (wind farm, long-term) contains output values resulting from long-term calculations at each turbine summarised over all the meteorological data. The output variables are listed in **Table 4.5**.

| Variable | Description |
|---|---|
| Turbine Name | Name of wind turbine |
| Turbine Type | Name of wind turbine type |
| X(m) | X coordinate of turbine in metres |
| Y(m) | Y coordinate of turbine in metres |
| Hub height (m) | Height of hub height in metres |
| Mean free stream wind speed (m/s) | Mean wind speed at the turbine location at hub height ignoring the effect of complex terrain and wind turbine wakes on the flow field |
| Mean gross wind speed (m/s) | Mean wind speed at the turbine location at hub height including the effect of complex terrain on the flow field but ignoring the effect of wind turbine wakes |
| Mean net wind speed (m/s) | Mean mind speed at the turbine location at hub height accounting for both the effect of complex terrain and the effect of wind turbine wakes |
| Mean net wind speed deficit (%) | 100 x (Mean gross wind speed – Mean net wind speed) / Mean gross wind speed |
| Mean free stream turbulence intensity (%) | Mean turbulence intensity at the turbine location at hub height ignoring the effect of complex terrain and wind turbine wakes on the flow field |
| Mean gross turbulence intensity (%) | Mean turbulence intensity at the turbine location at hub height including the effect of complex terrain on the flow field but ignoring the effect of wind turbine wakes |
| Mean net turbulence intensity (%) | Mean turbulence intensity at the turbine location at hub height accounting for both the effect of complex terrain and the effect of wind turbine wakes |
| Total free stream energy output (kWh) | Total energy generated by the current wind turbine at the free stream wind speed |
| Total gross energy output (kWh) | Total energy generated by the current wind turbine at the gross wind speed |
| Total net energy output (kWh) | Total energy generated by the current wind turbine at the net wind speed |
| Topographic efficiency (%) | 100 x Total gross energy output / Total free stream energy output |
| Array efficiency (%) | 100 x Total net energy output / Total gross energy output |
| Free stream estimated capacity factor (%) | The total free stream energy output expressed as a percentage of the total energy that could be produced if the turbine continuously generated power at its maximum capacity |
| Gross estimated capacity factor (%) | The total gross energy output expressed as a percentage of the total energy that could be produced if the turbine continuously generated power at its maximum capacity |
| Net estimated capacity factor (%) | The total net energy output expressed as a percentage of the total energy that could be produced if the turbine continuously generated power at its maximum capacity |

**Table 4.5** − Variables contained in a *.wfl* file for each modelled turbine.

# 4.5 Viewing met data

## 4.5.1 View processed met data

After a calculation has been run, a file containing meteorological data is produced with the extension *.out.met*. The numerical data in this file can be viewed by choosing **View processed met data** from the **Results** menu. The *.out.met* file contains a copy of the input meteorological data, cut down to the subset being used, with the effects of the choice of wind sector size taken into account, and (if necessary) converted from *.TAB* format to *.MET* format. Refer to Section 7.1 for more details on how the wind sector size affects the wind direction. Note the following points for input meteorological data in *.MET* format:

- If **Wind direction is in sectors** is not selected, the data in the *.out.met* file is identical to the input meteorological data file.
- If **Wind direction is in sectors** and **Wind direction cycling within sector** are selected, the data in the *.out.met* file is identical to the input meteorological data, except that the wind directions are altered to take into account the angular cycling applied.
- If **Wind direction is in sectors** and **Average results over wind sector** are selected, for each line of input meteorological data there is a line of data in the *.out.met* file for each wind direction increment. These lines are a copy of the input meteorological data except that the wind directions are altered to give the wind direction used to represent that increment, and the frequency is divided equally between the lines.

Note that there is a second meteorological summary file produced from a FLOWSTAR-Energy run. The *.mop* file (meteorological input and output parameters) contains the meteorological input parameters and the calculated meteorological output values for each line of data in the meteorological data file. The input parameters are in the left hand columns of the file, and the calculated output parameters in the right hand columns.

## 4.5.2 Plot wind rose of processed met data

After a calculation has been run, the option to create a wind rose from the *.out.met* results file is available from the **Results** menu. This option is available whether the input meteorological data was in **.tab* or **.met* format. An example of a wind rose is shown in **Figure 4.26**.

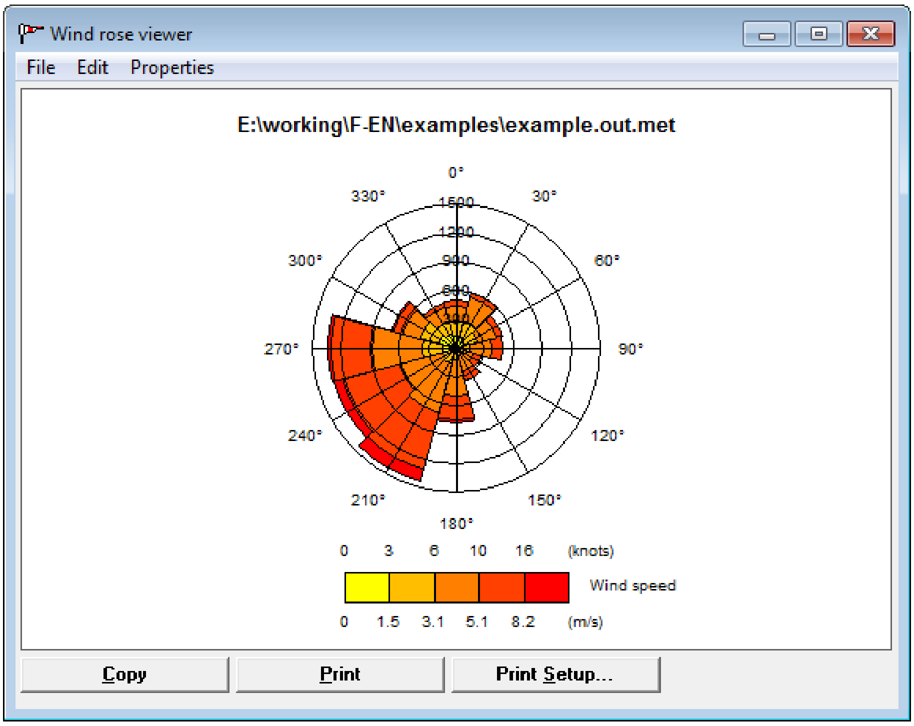



**Figure 4.26** – Wind rose viewer

# 4.6 Model run reporting files

The reporting files for the current *.fpl* file can be viewed from the FLOWSTAR-Energy **FPL data view** at any time by selecting the appropriate file from the **Results** menu. These files provide useful information about the model run.

There are four different reporting files available. If a file does not exist for the current *.fpl* (for instance, if there were no errors found) then the corresponding menu option will be unavailable (greyed out).

## 4.6.1 View report

The *.rpt* file is a summary (report) of the model input data (file pathnames, source characteristics, model options, etc.). It is designed to be easy to import into a Microsoft Word document. It should fit widthways on landscape A4, with standard margins, using Courier font, and minimum size 8 pt.

## 4.6.2 View log

The *.log* file contains a log of the model run. It details the date and time of the start and end of the run, any program messages, and a summary of the meteorological data (including the number of lines of data the model was able to use).

## 4.6.3 View warnings

The *.wng* (warning) file contains any warning messages produced during the run. These are non-fatal errors, i.e. they do not cause the model to stop. The user should check that for each of these warnings the required behaviour is being followed, and make adjustments to model input if necessary.

## 4.6.4 View errors

The *.err* file (error) is a text file produced if an error occurs during the run that causes the model to stop. Each error must be corrected before the model can run the *.fpl* file successfully.

# SECTION 5 Additional map view features

In this section additional features of the map view window, which are not covered elsewhere in this user guide will be discussed.

## 5.1 Modifying the appearance of layers

This section outlines how to modify the appearance of layers. The types of modification that can be made depend on the type of layer you wish to modify. Sections 5.1.1 to 5.1.5 give details of various different types of modification that can be made and the types of layer to which these modifications can be made. After modifying the appearance of the FPL data view layers the new appearance can be saved as the default appearance for layers, by selecting **Save FPL data Layer Settings** from the **File** menu. Note that this will save the current appearance of all of the FPL data layers. The settings for individual layers can be saved or reloaded by right-clicking on the layer and selecting **Save settings file** or **Reload settings file** respectively. The settings for an individual layer can also be exported and then imported to other layers, or imported in new maps later. This can be achieved using the **Import layer settings...** and **Export layer settings...** obtained by right-clicking on the layer. These options are discussed in Section 5.2.

***Showing and hiding layers***

If a layer is visible then the data contained in that layer are shown in the map view window. A layer can be hidden by unchecking the checkbox next to the layer name. To make the layer visible again recheck the checkbox.

***Reordering layers***

The ordering of the layers in the **Legend** determines the order in which they are shown in the map view window. Features from layers at the top of the **Legend** are shown on top of features from layers further down and thus may hide them from view. To reorder the layers within the **Legend** click on the layer you wish to move, holding down the mouse button move it to its new position, and then release the mouse button. The symbol used to represent a layer can be made partially transparent to allow features in layers below them to be seen.

### 5.1.1 Modifying the transparency of a layer

The transparency of any layer can be changed. The transparency determines how "see-through" the layer is and thus whether any features in a lower layer can be seen through features in this layer. To change the transparency of a layer, follow these steps:

**Step 1** Double click on the layer in the **Legend** to bring up the layer properties window.

**Step 2** Select the **Layer** tab and then the **Parameters** section, as shown in **Figure 5.1**.

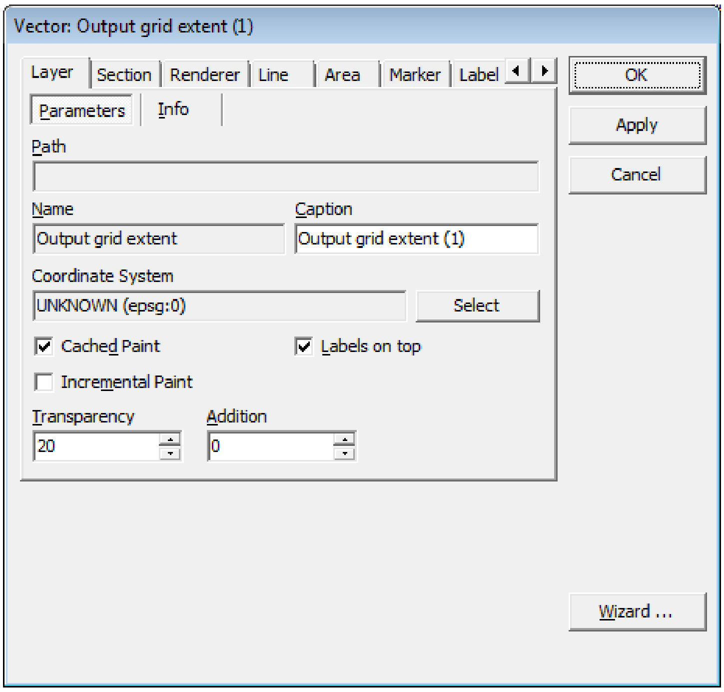



**Figure 5.1** - The **Layer** tab of the layers properties screen.

**Step 3** Alter the **Transparency** value as required, 0 is completely transparent and 100 is completely opaque.

**Step 4** Click on the **OK** button to make the changes and return to the map view window, or click on the **Apply** button to make the changes but remain in the layer properties window, or click on the **Cancel** button to discard the changes and return to the map view window.

### 5.1.2 Modifying the appearance of a marker layer

Markers are used to represent any layer of data defined as single points. For instance, the layers for specified points and wind turbines are marker layers. To modify the appearance of one of these layers follow these steps:

**Step 1** Double click on the layer of interest in the layers panel to bring up the layer properties window, e.g. the **Specified points** layer.

**Step 2** Select the **Marker** tab, as shown in **Figure 5.2**.

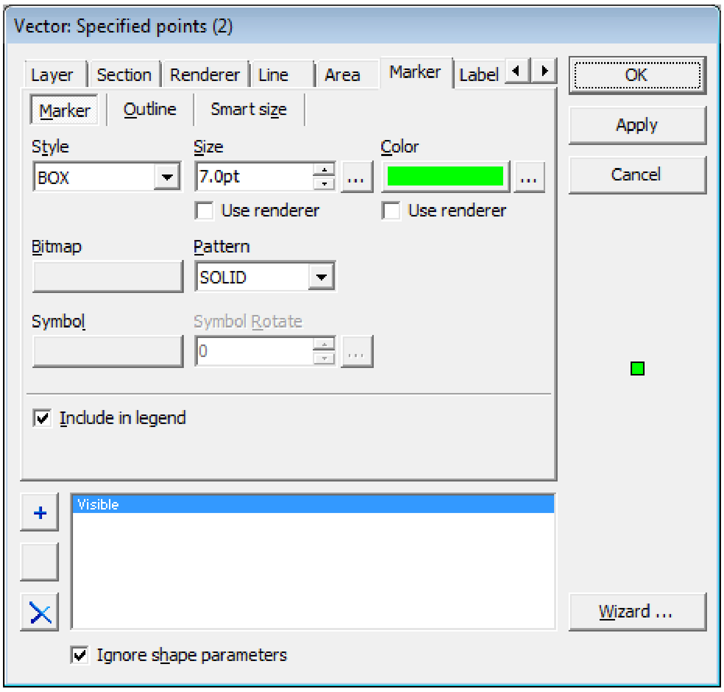



**Figure 5.2** - The **Marker** tab of the layer properties screen.

**Step 3** The **Marker** and **Outline** sections can be used to alter the look of the markers used to represent the features in this layer.

**Step 4** Click on the **OK** button to make the changes and return to the map view window, or click on the **Apply** button to make the changes but remain in the layer properties window, or click on the **Cancel** button to discard the changes and return to the map view window.

### 5.1.3 Modifying the appearance of an area layer

Coloured areas are used to represent any layer of data represented by two-dimensional regions in the map view such as the output grid extent, for instance. To modify the appearance of one of these layers follow these steps:

**Step 1** Double click on the layer of interest in the **Legend** to bring up the layer properties window, e.g. the **Output grid extent** layer.

**Step 2** Select the **Area** tab, as shown in **Figure 5.3**.

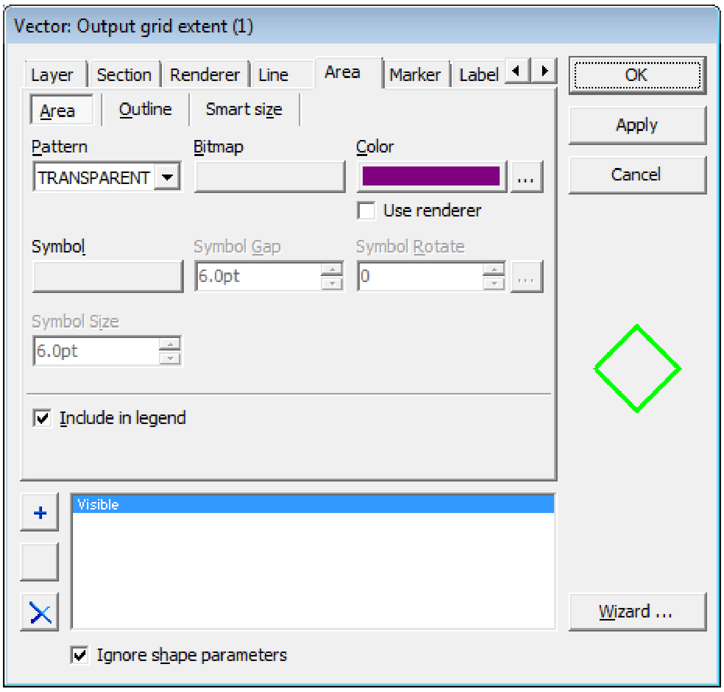



**Figure 5.3** - The **Area** tab in the layer properties screen.

**Step 3** The **Area** and **Outline** sections can be used to alter the appearance of the shapes used in this layer.

**Step 4** Click on the **OK** button to make the changes and return to the map view window, or click on the **Apply** button to make the changes but remain in the layer properties window, or click on the **Cancel** button to discard the changes and return to the map view window.

## 5.1.4 Colouring a layer according to its properties

You may wish to colour a layer according to its properties. For example, you may wish to colour a terrain file according to elevation; or a spatially varying roughness file according to the roughness value. This section describes how to make these changes.

**Step 1** Double click on the layer which you wish to modify, for example the terrain layer, to bring up the properties window, see **Figure 5.4**.

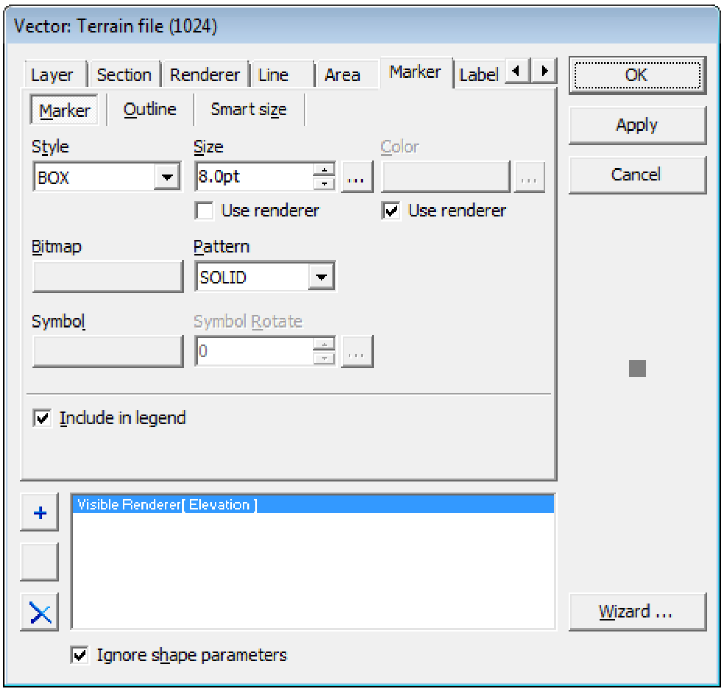



**Figure 5.4** – The properties wizard for the terrain and roughness plots

**Step 2** Click on the **Wizard...** button. The **Rendering Wizard** dialogue box appears as shown in **Figure 5.5**

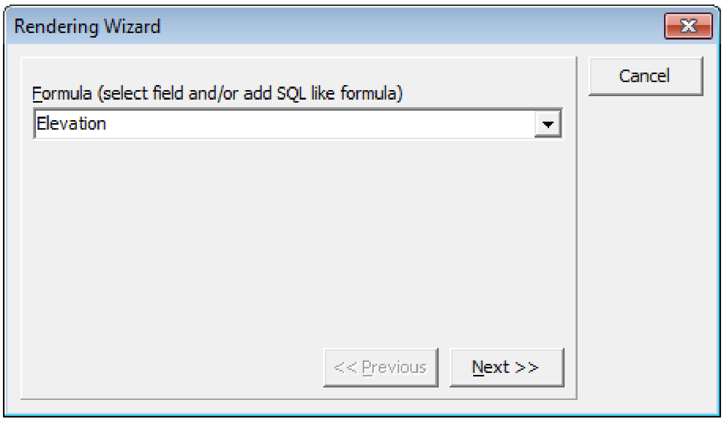



**Figure 5.5** – The **Rendering Wizard** dialogue box

**Step 3** Using the drop down menu select the property according to which you wish to colour the layer. For example, for a terrain layer select **Elevation**. Click on **Next>>.**

**Step 4** The next screen allows a choice between colouring by **Unique values** or by **Continuous values**. The choice depends on the property you are using to colour the layer. For the terrain layer, for example, it is appropriate to choose **Continuous values** so there is a smooth gradation in colours from low elevations to high elevations. On the other hand, when plotting surface roughness, for example, the changes in roughness may be discrete, and it is likely that there will be only a small number of different roughness values across the region, so colouring by **Unique values** is more appropriate. The choice can be made by selecting the appropriate radio button, and in the continuous case you can also specify the range of values to use when colouring the layer, see **Figure 5.6**. Click on **Next>>.**

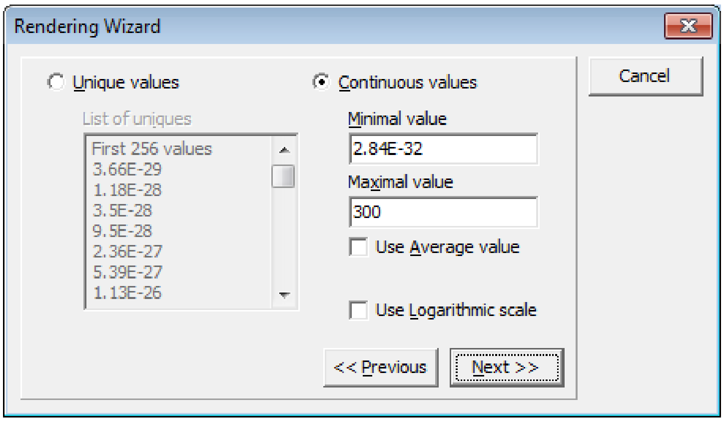



**Figure 5.6** – Selecting unique or continuous values

**Step 5** On the final screen make sure that **Render by Color** is selected and then click on **Apply**.

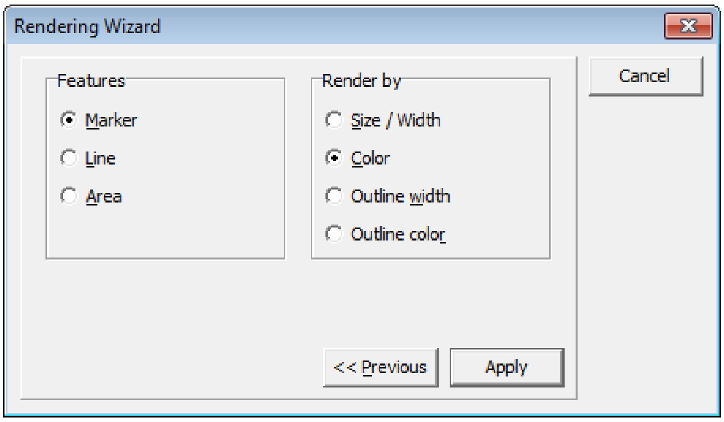



**Figure 5.7 – Render by Color**

**Step 6** The next step is to choose the colour scheme for the layer. The method for doing this depends on whether the layer is being coloured by continuous or unique values.

For continuous values click on the **Renderer** tab in the properties wizard, see **Figure 5.8**. You can then change the minimum and maximum values at which the start and end colours will be applied. You can also choose the colours used at these points by clicking on the start or end colour and selecting a new colour from the palette.

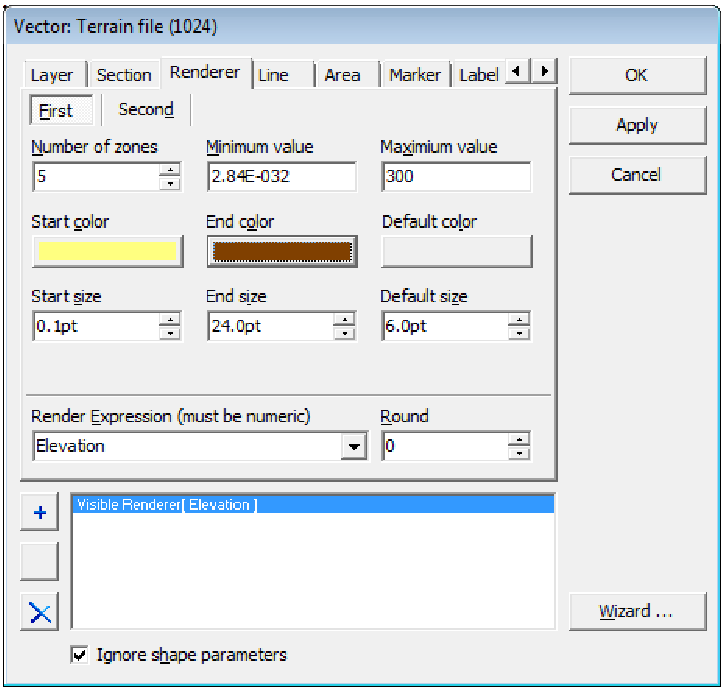



**Figure 5.8** – The **Renderer** tab

For unique values a separate colour can be selected for each individual level. To do this, click on the appropriate tab in the properties wizard, e.g. the **Marker** as shown in **Figure 5.9**. Then select each level in turn from the window and choose the colour by clicking on **Color** and selecting your choice from the palette.

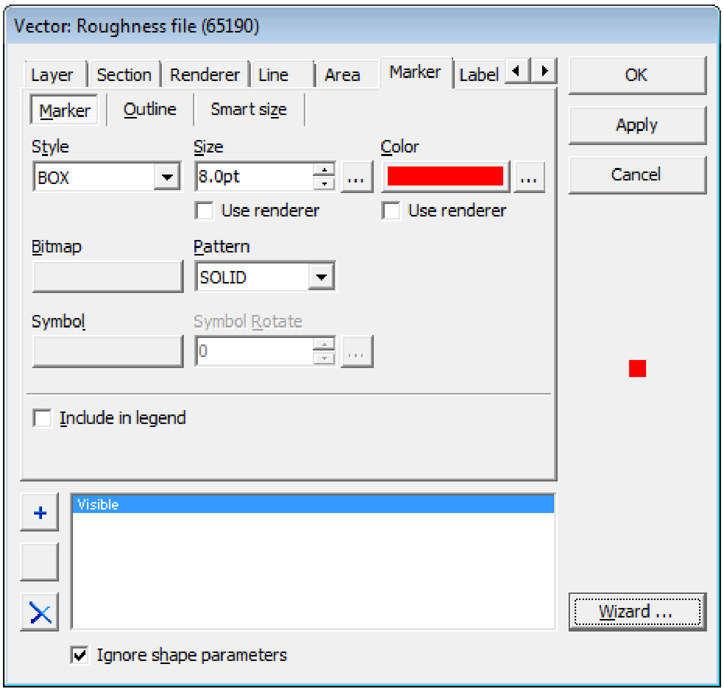



**Figure 5.9** – The **Marker** tab

**Step 7** Finally, click **OK** to apply your changes.

## 5.1.5 Displaying feature names

Any of the default FPL data view layers can display the names of the features within that layer, for instance wind turbine names. To show these names follow these instructions.

**Step 1** Double click on the layer of interest in the layers panel to bring up the layer properties window.

**Step 2** Select the **Label** tab, as shown in **Figure 5.10**.

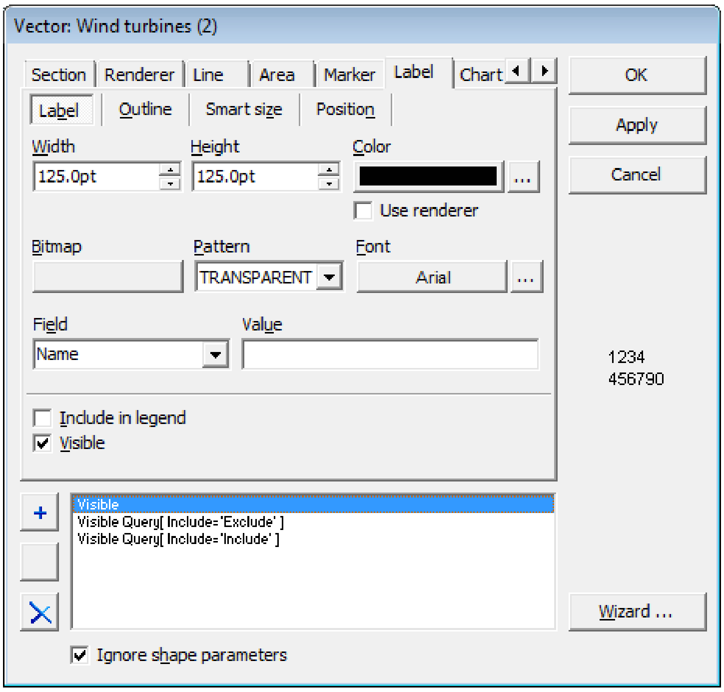



**Figure 5.10** - The **Label** tab of the layer properties screen.

**Step 3** In the **Label** tab set the **Field** to be Name.

**Step 4** The **Label**, **Outline** and **Position** sections can then be used to alter the properties of the labels.

**Step 5** The labels can be removed by unchecking the **Visible** checkbox in the **Label** section.

**Step 6** Click on the **OK** button to make the changes and return to the map view window, or click on the **Apply** button to make the changes but remain in the layer properties window, or click on the **Cancel** button to discard the changes and return to the map view window.

## 5.2 Exporting, importing, saving and reloading layer settings

Once the layers have been modified as desired the layer settings can be exported so that they can be used in other layers or in other maps. Similarly, existing layer settings can be imported. The layer settings can also be saved and reloaded. These features are particularly useful for contour plots and flow field plots. Step-by-step instructions are provided below.

**Step 1** Right click on the layer you wish to export in the map view **Legend**. This brings up the menu shown in **Figure 5.11**.

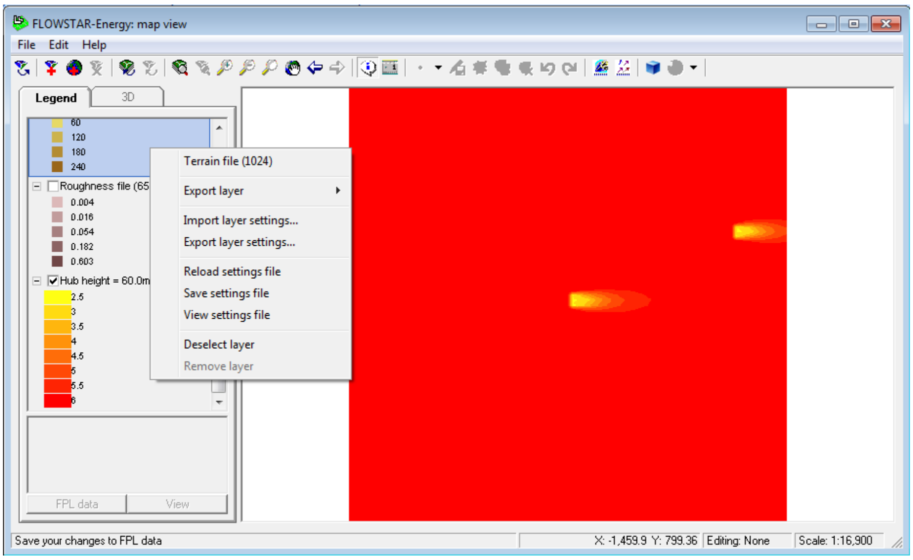



**Figure 5.11** – The menu for exporting/importing layer settings.

**Step 2** Click on **Export layer settings...** .

**Step 3** A window opens prompting you to save the configuration file for the layer. Choose a location and save the file with extension **.ini*.

**Step 4** Right click on the layer to which you wish to import the settings. From the menu select **Import layer settings...** .

**Step 5** Navigate to the layer configuration file that you have just created and click on **Open**.

**Step 6** The layers are now displayed using the same layer settings.

The settings for an individual layer can be saved by right-clicking on the layer and selecting **Save settings file**. These settings are then stored in a file so if you wish to return to the saved settings, after making modifications, you can right-click on the layer and select **Reload settings file**. This undoes any changes you have made.

## 5.3 Setting the coordinate system

There is an option to set the coordinate system within the map view.

The choice of coordinate system affects the way the layers are displayed in the map view window and the coordinates of a point in the map view window (given in the status bar) are in the units associated with the coordinate system defined. For example, if the coordinate system has been set to **WGS 84** then the coordinates are longitude and latitude.

The coordinate system must be selected before certain features in the map view window can be used; for example, before exporting map view files to **.kml* files for use in Google Earth.

To set the coordinate system that is used in the map view select **Set map coordinate system** from the map view **Edit** menu, as shown in **Figure 5.12**. You must also untick **Use the FPL data coordinate system** option from this menu. This option allows you to choose the coordinate system from an extensive list of options.

*You must choose a coordinate system that is the same as, or consistent with, that used for the FPL data view layers.*

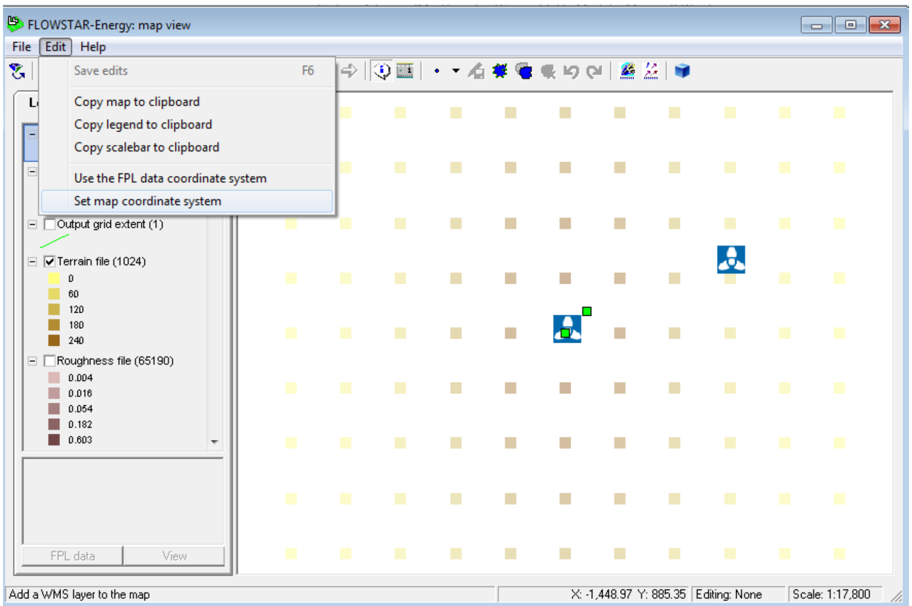



**Figure 5.12** – The coordinate systems options in the **Edit** menu.

Once the **Set map coordinate system** option is selected then a new window appears, as shown in **Figure 5.13**, where the coordinate system can be chosen from an extensive list of options. To select a **Projected system** click on the radio button and then choose the system from the drop-down list.

*Rather than scrolling through the entire list of options to find your desired choice you can click in the box and type the name of the coordinate system you wish to use. For example, to quickly find the Ordnance Survey grid for Great Britain type* ***OSGB****. You can also type part of the name and then click on the drop down arrow to see a shorter list of options.*

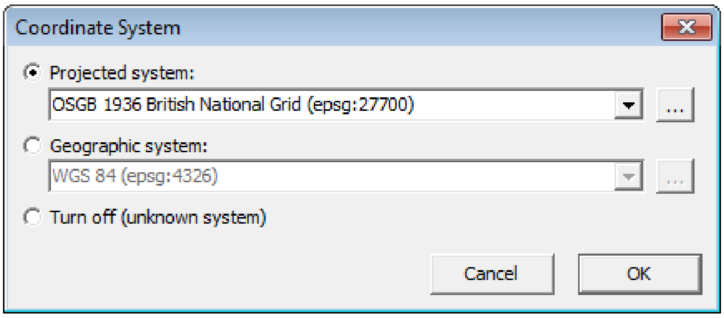



**Figure 5.13** – Choice of the coordinate system

A **Geographic system** can be chosen in a similar way to a **Projected system**.

If you wish to use the **Unspecified regular Cartesian** grid, which is used in the FPL data view, i.e. if the coordinates are site specific, for example distances relative to a wind turbine, then the **Turn off (unknown system)** option must be chosen.

### 5.3.1 Setting the coordinate system for a background layer

To set a coordinate system for a background image you will need to create a **.prj* file as follows. Suppose that the background map you wish to add has the filename *map.tif*. Then follow these steps:

**Step 1** Double click on one of the layers in the **Legend**, a new window will appear.

**Step 2** Select the **Layer** tab to view the layer information, as shown in **Figure 5.14**.

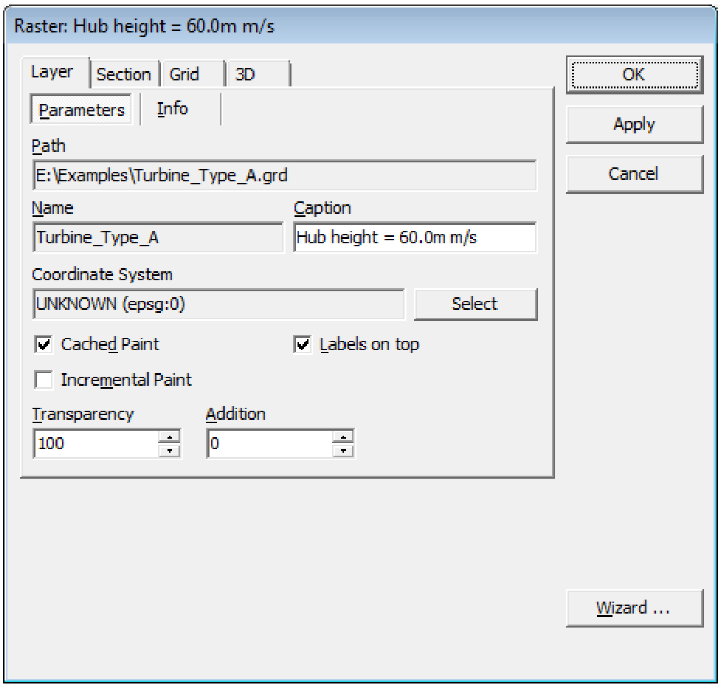



**Figure 5.14** – The **Layer** tab

**Step 3** Click on the **Select** button next to the **Coordinate System** cell to bring up the dialogue box shown in **Figure 5.15**.

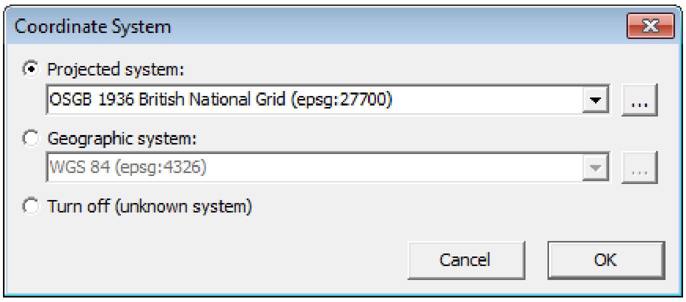



**Figure 5.15** – Choice of the coordinate system

**Step 4** Choose the coordinate system you wish to use in the background layer from the available list and then click on the ellipsis (...). This will bring up the **Coordinate System Setup** screen shown in Figure 26.

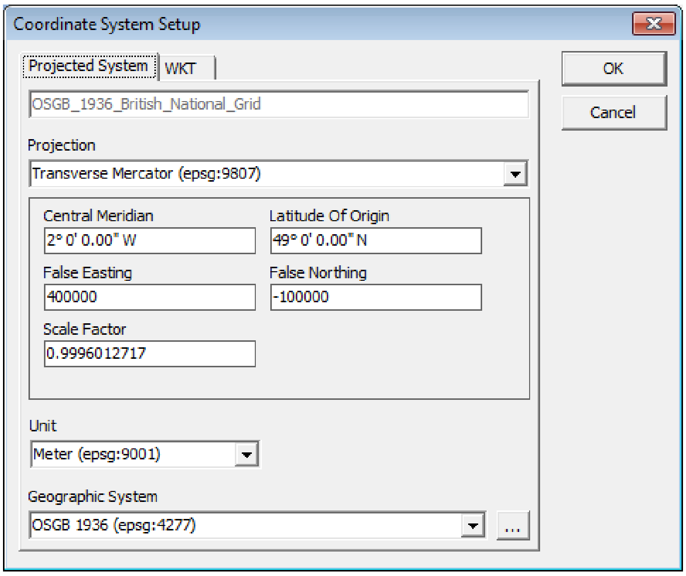



**Figure 5.16** – The **Coordinate System Setup** screen

**Step 5** Click on the WKT tab. The screen will be similar to that shown in **Figure 5.17**.

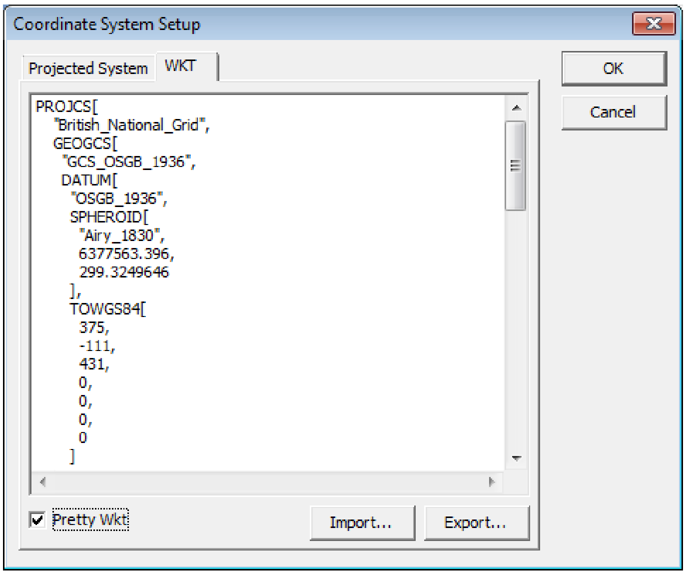



**Figure 5.17** – The **WKT** tab

**Step 6** Click on **Export...** and save the file in the same location as the map.tif file with the file stem map.tif and extension .prj, i.e. save the file with name map.tif.prj.

## 5.4 Copying the map view window to the clipboard

An image of the current view in the map view window can be copied to the clipboard for pasting into other Windows applications. To copy the current view from the map view window select **Copy Map To Clipboard** from the **Edit** menu. This image can then be pasted into a document or picture editing software and saved.

The **Copy Legend To Clipboard** option from the **Edit** menu can be used to copy an image of the **Legend** to the clipboard. If only a part of the **Legend** is required, for instance the legend for a contour plot, then this image of the **Legend** should be pasted into image processing software and then cut down as appropriate.

*The information panel section of the* ***Legend*** *is not copied to the clipboard.*

# 5.5 Adding background map imagery from a Web Map Service (WMS)

It is possible to add a background map layer in the map view from a Web Map Service (WMS). The background map imagery can help display and locate your data and results. There are two methods for adding background map imagery: one is to use the Add WMS layer tool to specify a URL to a web map service; the other is to use a special Protocol Layer Connector file (.*ttkwp*).

### 5.5.1 Adding background map imagery with the Add WMS layer tool

**Step 1** First ensure that the coordinate system in the map view is set appropriately for the map that you wish to use. Also ensure that all the coordinates in the FPL data view are given using that coordinate system.

**Step 2** Click on the Add WMS layer tool

**Step 3** A window appears as shown in **Figure 5.18**. Specify the URL for the web map service and click **OK**.

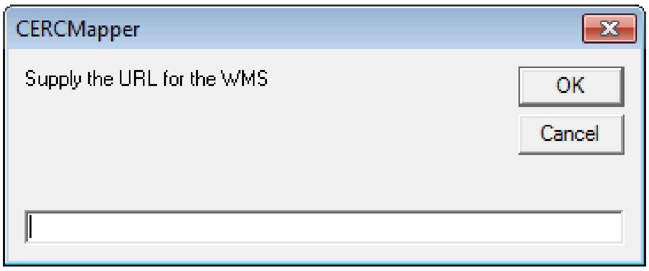



**Figure 5.18** – The **Add WMS Layer** window.

**Step 4** The map will be displayed. The layers may need to be reordered to view the map and other layers properly. An example is shown in **Figure 5.19**.

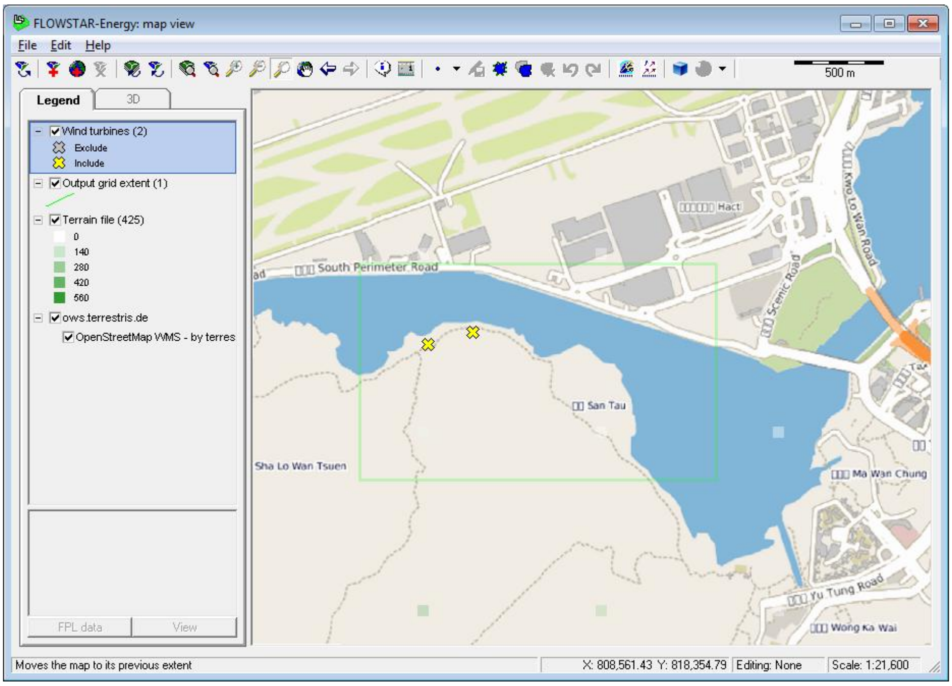



**Figure 5.19** – Background map imagery in the map view; © OpenStreetMap contributors www.openstreetmap.org/copyright.

*For best results, Map layers and data layers should use the same coordinate system. Some coordinate systems may not be mutually compatible.*

### 5.5.2 Adding background map imagery using a Protocol Layer Connector file

Alternatively a background map may be added by using a Protocol Layer Connector file as described here. An example file is included in your FLOWSTAR-Energy install directory: this links to the OpenStreetMap web map service.

**Step 1** Ensure that the coordinate system in the map view is set appropriately for the map that you wish to use.

**Step 2** Click on the **Add Layer** tool .

**Step 3** Navigate to the directory containing your **.ttkwp* file. Select **Files of type: Protocol Layer Connector (*.ttkwp)** to display the files:

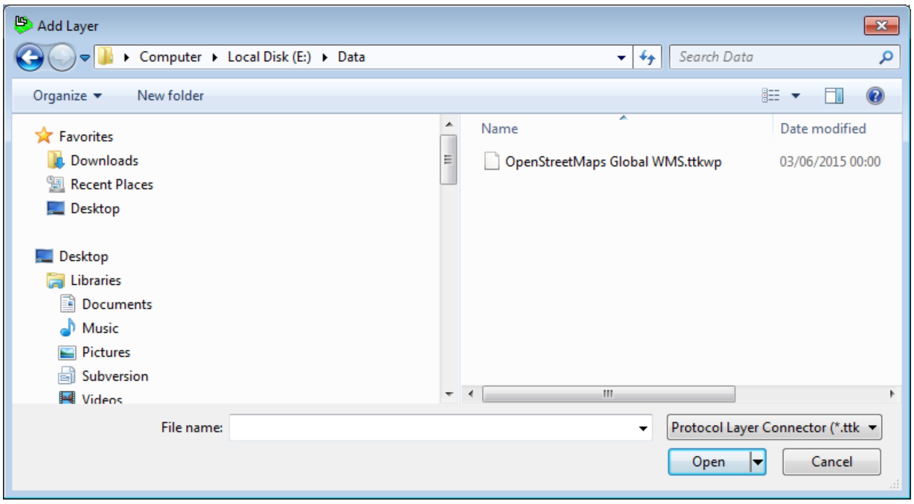



**Figure 5.20** – Selecting Protocol Layer Connector files.

**Step 4** Select a (*.ttkwp) file that covers the geographic location of your study area and click **Open**.

**Step 5** The new layer will be displayed in the map view. The layers may need to be reordered so that the background image does not obscure the view of the other layers. Information on reordering layers can be found in Section 5.1.

**Step 6** Select the output grid extent or largest FPL data layer and select **Zoom to layer** .

**Step 7** This will zoom to the correct location on the map (**Figure 5.21**).

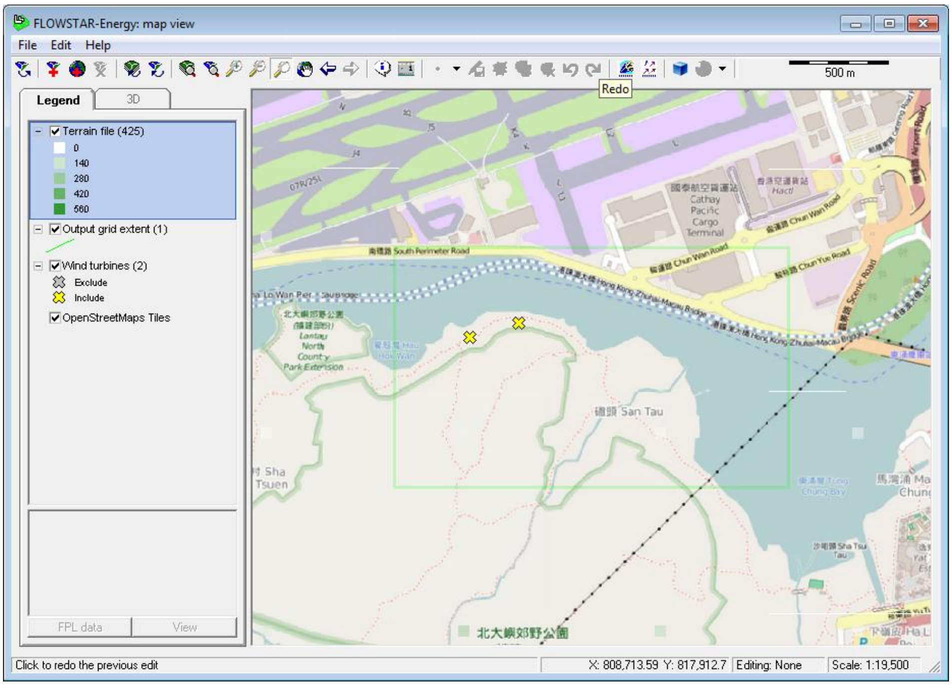



**Figure 5.21** – Background image displayed in the map view; © OpenStreetMap contributors www.openstreetmap.org/copyright.

---

*For best results, Map layers and data layers should use the same coordinate system. Some coordinate systems may not be mutually compatible.*

---

*Contact the CERC helpdesk for help setting up *.ttkwp files for other map service providers.*

## 5.6 Information about a feature

The map view allows you to view information about a model feature from within the map view window such as a wind turbine. To display the information, follow these instructions.

**Step 1** Reorder the layers in the **Legend** so that the layer containing the relevant data is on top.

**Step 2** Click on the **Information** button on the toolbar .

**Step 3** Then click on the feature in the map view window.

**Step 4** Information will be displayed about the feature in the layer panel, as shown in **Figure 5.22**.

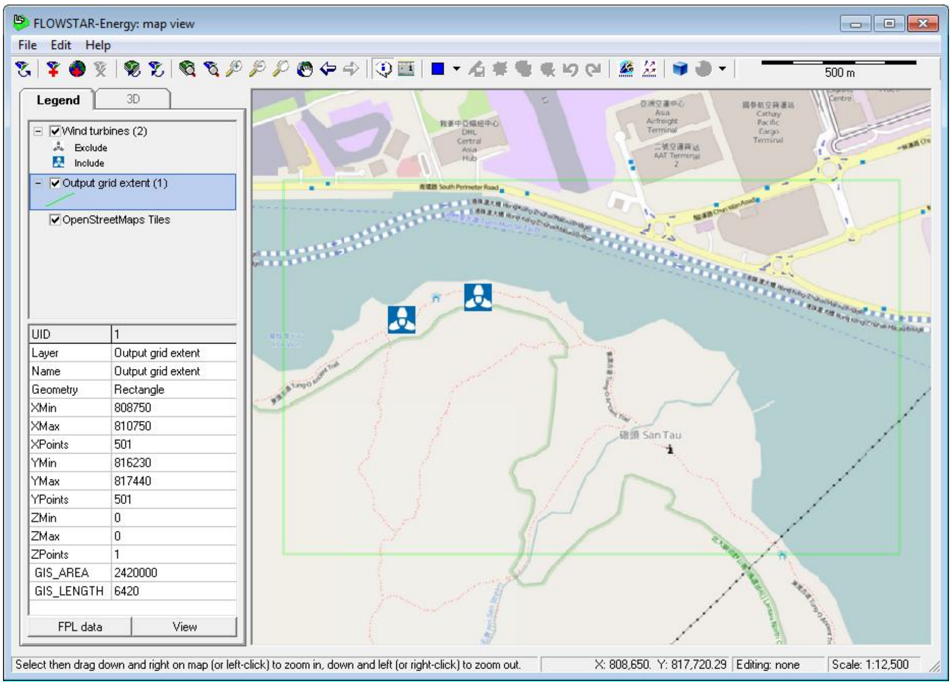



**Figure 5.22** – The feature information screen for an output grid extent.

**Step 5** Click on the **FPL data** button to be directed to the feature in the FPL data view window.

**Step 6** Click on the **View** button for the feature to be highlighted in the map view. The map view will also be panned so that the feature is in the centre of the screen.

## 5.7 Measuring distances

The map view comes with a tool that allows you to measure the distance between two or more points in the map view window. Follow these instructions to measure distances in the map view window.

**Step 1** Click on the **Measure** button on the toolbar. 

**Step 2** Click on the map view window at the first point of interest, and then click on a second point. A dotted line is produced between the two points of interest and the distance between the two points is shown on the status bar in the bottom left hand corner of the map view window, as shown in **Figure 5.23**.

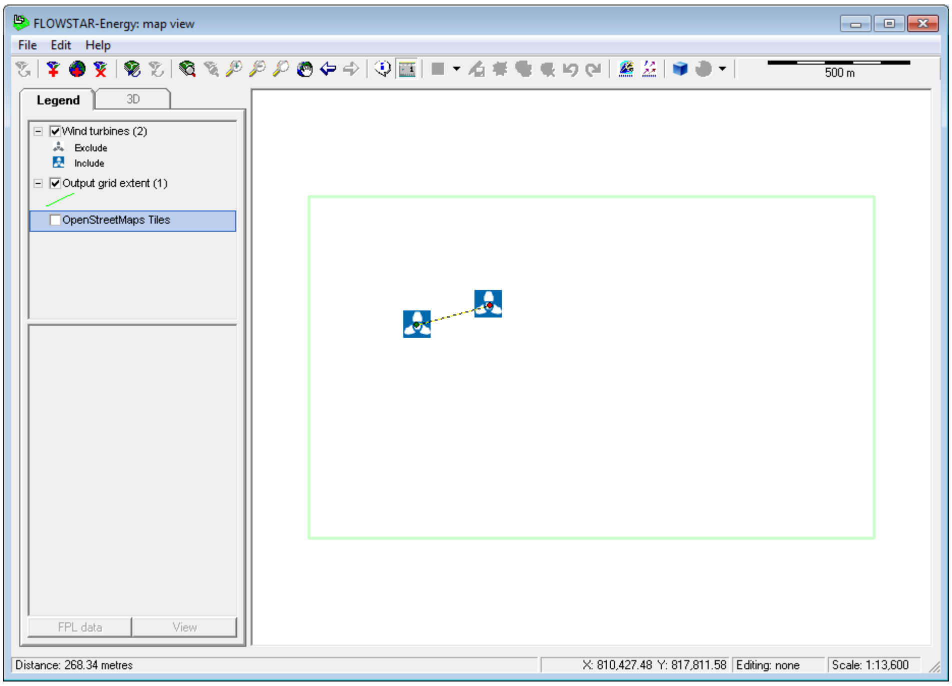



**Figure 5.23** – Measuring distances in the map view, the distance is given in the bottom left corner.

**Step 3** Clicking on further points adds further lines and the cumulative distance is displayed in the status bar.

**Step 4** The position of the points can be modified as follows:

- To add a point, click on the existing point that lies before the desired location of the new point and then click where you want the new point to be.
- To move a point, click on the point, and while holding down the mouse button, move the cursor to the new location for the point before releasing the mouse button.
- To delete a point, first click on the point to select it, and then click on it again.

*The points that the distance is being measured between appear as coloured squares. The current selected point is red and the other points are green.*

**Step 5** To remove all the defined points whilst using the measuring tool, double click anywhere in the map view window.

## 5.8 Displaying the north arrow

It is possible to toggle the display of an arrow in the map view that indicates the direction of north, see **Figure 5.24** . The north arrow is only displayed when the plot is in 2D mode.

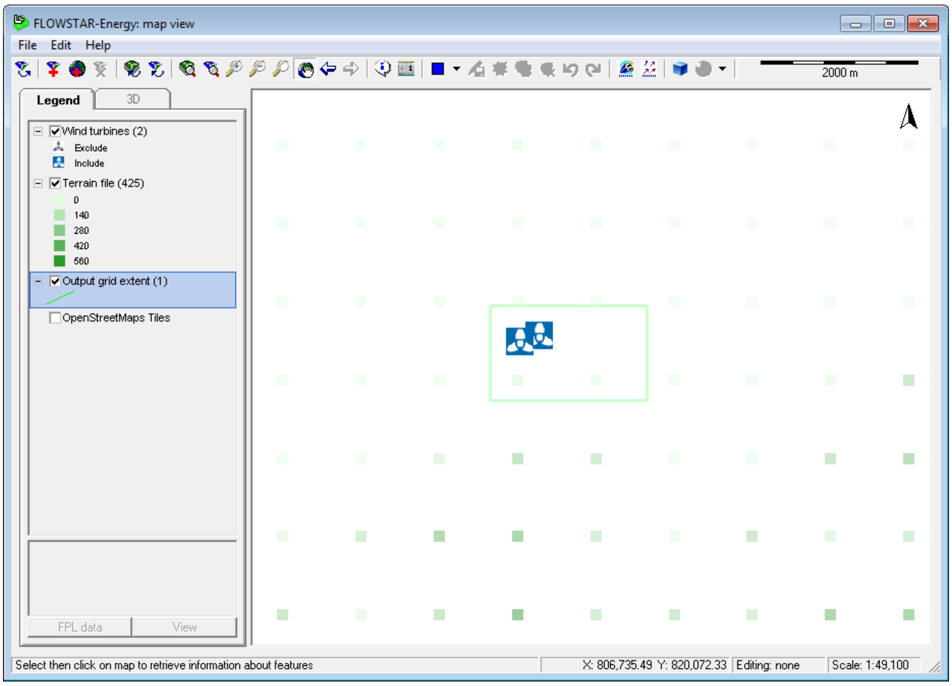



**Figure 5.24** – The north arrow is displayed in the top right hand corner of the map window.

To toggle the north arrow select the **File – Preferences – Viewing option** menu and select or deselect **Display north arrow** as desired.

## 5.9 Exporting features

The default FPL data layers can be exported to a variety of formats for use in third-party software such as ArcView, MapInfo or Google Earth. The formats available are

- ArcView Shape Files (*.*shp*)
- Autocad (*.*dxf*)
- Geographic Markup Language (*.*gml*)
- GPS Exchange Format (*.*gpx*)
- Keyhole Markup Language (*.*kml*)
- MapInfo Interchange (*.*mif*)

To export a layer into any of these formats first click on the layer in the **Legend** to select it. Then click on the **Export Layer** button on the toolbar . This brings up the export layer screen, as shown in **Figure 5.25**. Select the file type you wish to export to, give the file a name then click on the **Save** button.

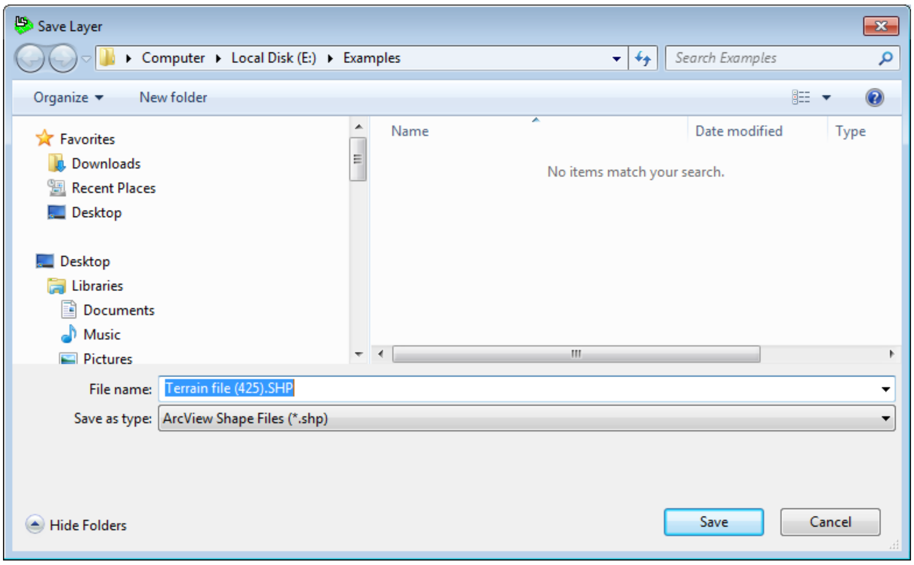



**Figure 5.25** - The **Export Layer** window.

Alternatively a layer can be exported by selecting it and then right-clicking to display the menu. Select **Export layer** and then select the file type from the sub-menu, e.g. **to SHP....** A new window will appear which you can use to save the file in your desired location. The file type will be selected automatically and a default name for the file is suggested, based on the name of the layer.

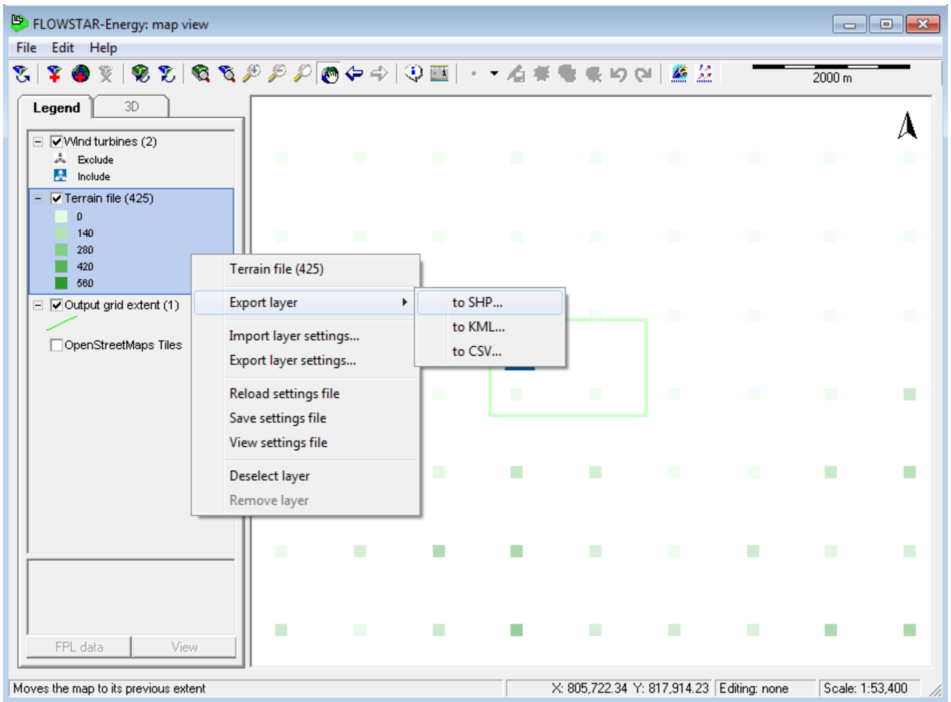



**Figure 5.26** - The **Export Layer** right-click option.

### 5.9.1 Exporting to Google Earth

The layers can be exported so that they can be viewed in Google Earth. This may be useful for checking the locations of wind turbines or specified points against a satellite image of the area being modelled. To do this, follow these steps:

**Step 1** Set the coordinate system for the map, see Section 5.3.

**Step 2** Ensure that a valid projected or geographic coordinate system has been set for the layers that are being exported.

*You must set a coordinate system for the map view before you can export a map view layer to Google Earth.*

**Step 3** Export the layer to Keyhole Markup Language (**.kml*) by following the instructions given in Section 5.9.

**Step 4** Open the **.kml* file in Google Earth.

## 5.10 Viewing features in 3D

Features in the map view can be viewed in 3D. This can be especially helpful when checking turbine heights, or when examining complex terrain or variable roughness.

To see a 3D visualisation of the features in the map view first click on the **Change View** icon . After doing this, the map view will change and the **Change View** icon will also change. To return to the 2D planar view, click again on the new **Change View** icon .

When in 3D mode, three axes are shown: vertical (blue), east-west (red) and north-south (green). By clicking and holding the mouse you can rotate the map to examine the 3D visualisation from various points of view. A new button appears which enables you to change the method of panning the camera – the options available are summarised in **Table 2.6**.

See for example **Figure 5.27** which shows a view of buildings in the map view when in standard mode and when in 3D mode.

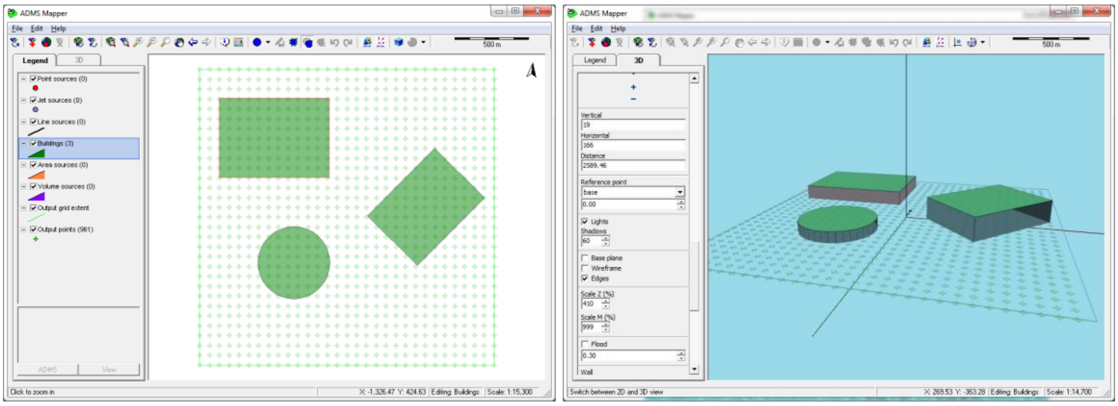


**Figure** 5.27 – Buildings viewed in 2D and 3D modes.

When in 3D mode the view can also be changed a variety of ways by using the options under the 3D tab in the layer panel.

---

*The vertical scale can be exaggerated using the **Scale Z (%)** and **Scale M (%)** options in the 3D tab. This may be required for clarity if the horizontal extent of the modelling region is much larger than the vertical extent of the wind turbines, terrain, etc.*

---

# SECTION 6 Worked Examples

In order to help the user to learn how to use FLOWSTAR-Energy, worked examples have been set up to cover the main functions of the model. These illustrate how to set up a scenario, how to calculate long-term and short-term average results under different meteorological conditions, and how to look at the calculated output.

You can launch FLOWSTAR-Energy in several different ways:

- double-click on the icon for the shortcut created earlier (see Section 2.2); or
- use the Windows **Start** menu and select **Programs**, **FLOWSTAR-Energy.**

**It is strongly recommended to create a directory for setting up and running these examples in order to keep them separate from the examples provided in the** *<install_path>\Examples* **directory supplied with the model.**

---

*New users of FLOWSTAR-Energy are advised to carry out these worked examples before attempting to set up runs.*

---

## 6.1 Example 1: Onshore calculations

The object of this example is to demonstrate how to set up and run an onshore modelling study and to look at the different types of output available from FLOWSTAR-Energy.

In particular, this example looks at modelling the potential wind energy available over a hilly area. The effect of different wind speeds on flow field output is discussed, then a small wind farm is modelled and flow field and wind farm output is examined.

### 6.1.1 Setting up the run

To model the potential wind energy and examine the flow field over a hilly area, proceed as follows.

**Step 1** Open FLOWSTAR-Energy and in the **FPL data view** window click on **File, New** to create a new input file, i.e. a default input file or “clean sheet” in which to enter data.

**Step 2** Click on **File**, **Save** or **Save As...** and save the file as *Example1a.fpl*.

**Step 3** Enter a **Name of project** and **Name of site** of your choice on the **Project details** and **Site characteristics** screens respectively.

These are optional but sensible titles should be entered as they help to differentiate between model runs. The titles will be written out to the log file of the model run.

**Step 4** This study is onshore i.e. the site is on land, not at sea. Click the drop-down next to **Surface roughness at onshore study site** to see a list of land use types. For this example the study area is hilly with bracken and scrub, so select **Agricultural areas (max)**. The roughness length will be set to 0.3m, a suitable value for this land use. The terrain height over the area varies, so the effect of hills is included in this calculation. The terrain file to use in this example is called *Example1.ter* and can be found in your *<install path>/Data/Example1* directory. The default **Hills calculation grid resolution** is 64x64 points, which is adequate for this calculation. This completes input to the **Site characteristics** screen. Compare your input to **Figure 6.1**.

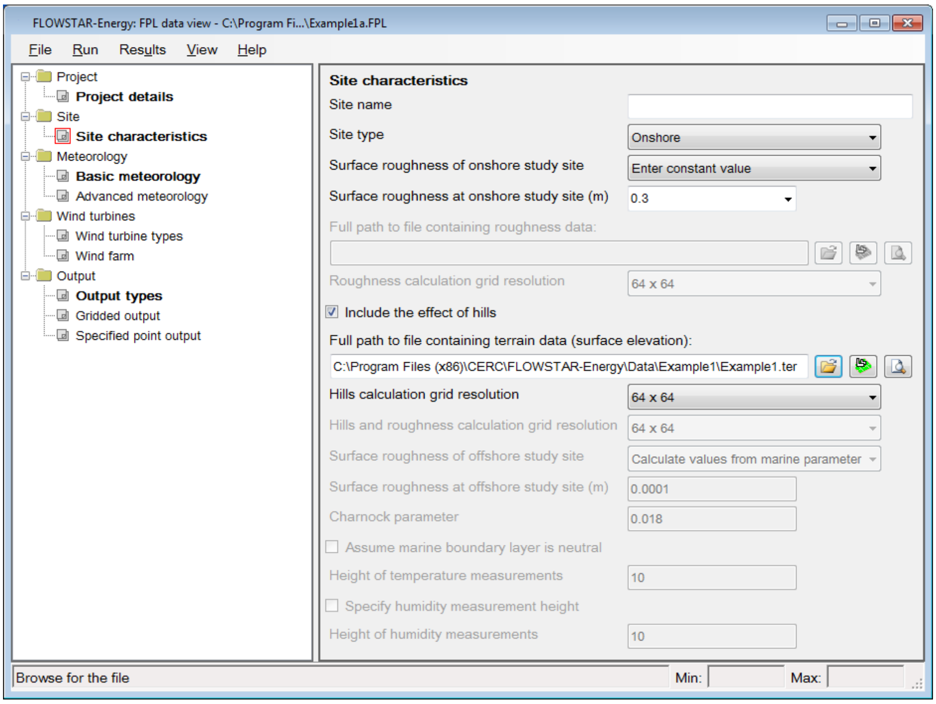



**Figure 6.1** − Example 1a: the **Site characteristics** screen.

**Step 5** Press the map view button next to the terrain file path, , in order to see a visualisation of the terrain being modelled in this example, as shown in **Figure 6.2**. You may need to click the **Refresh layers** button  in the **map view**.

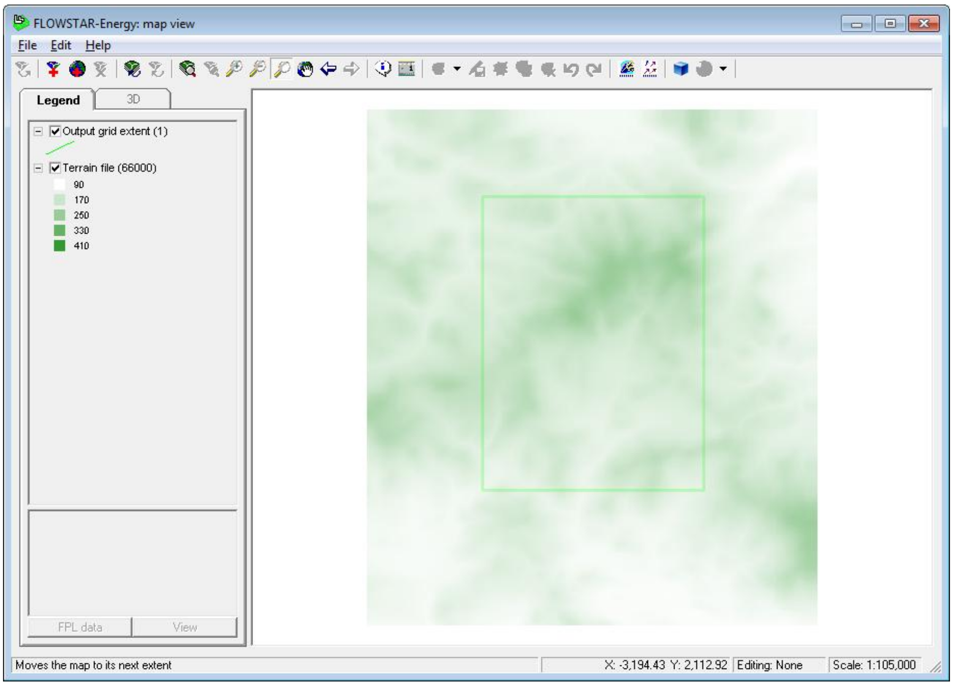



**Figure 6.2** – Example 1a: the **map view** window showing the terrain data.

**Step 6** A short meteorological file has been supplied, named *vary_wind_speed.met*. In this file there are three meteorological conditions, which are identical except for the wind speed; this is set to 8m/s in the first line, 10m/s in the second, and 12m/s in the third. In the **Basic meteorology** screen leave the surface roughness the same as the study site value, and browse to locate the correct meteorological file in your *<install path>/Data/Example1* directory, as shown in **Figure 6.3**.

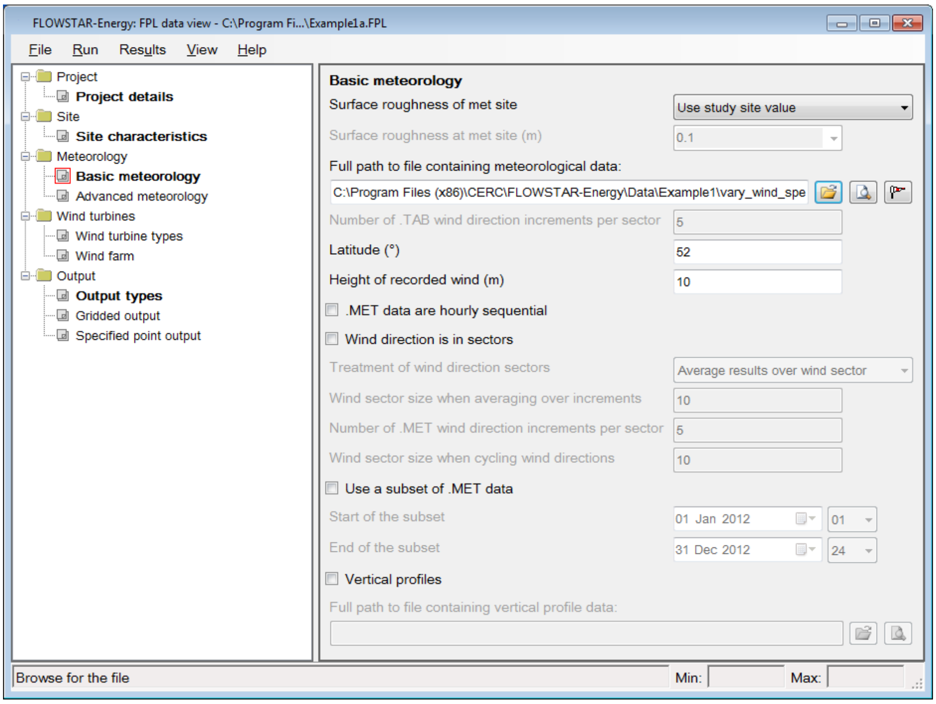



**Figure 6.3** – Example 1a: the **Basic meteorology** screen.

Once the meteorological file has been selected, various options below become editable. For this example leave the **Latitude** as the default value of 52° and the **Height of the recorded wind** as 10m.

**Step 7** In order to obtain potential wind energy output, details of the turbine type need to be entered. The **Wind turbine types** screen is optional; enable it by checking the box at the top-right of the screen or by double-clicking the screen name in the left hand side. In this example an ENERCON E48 turbine will be modelled. A data file named *EnerconE48.wtd* containing the turbine specifications is supplied in the *<install path>/Data/Example1* directory. These data have been obtained from the manufacturer. When carrying out a study for a different turbine you will need to create an equivalent file with power and thrust data obtained from the relevant turbine manufacturer. In the **Wind turbine types** screen enter the ENERCON E48 details, using a 48m diameter and hub height of 55m; select the file as shown in **Figure 6.4**.

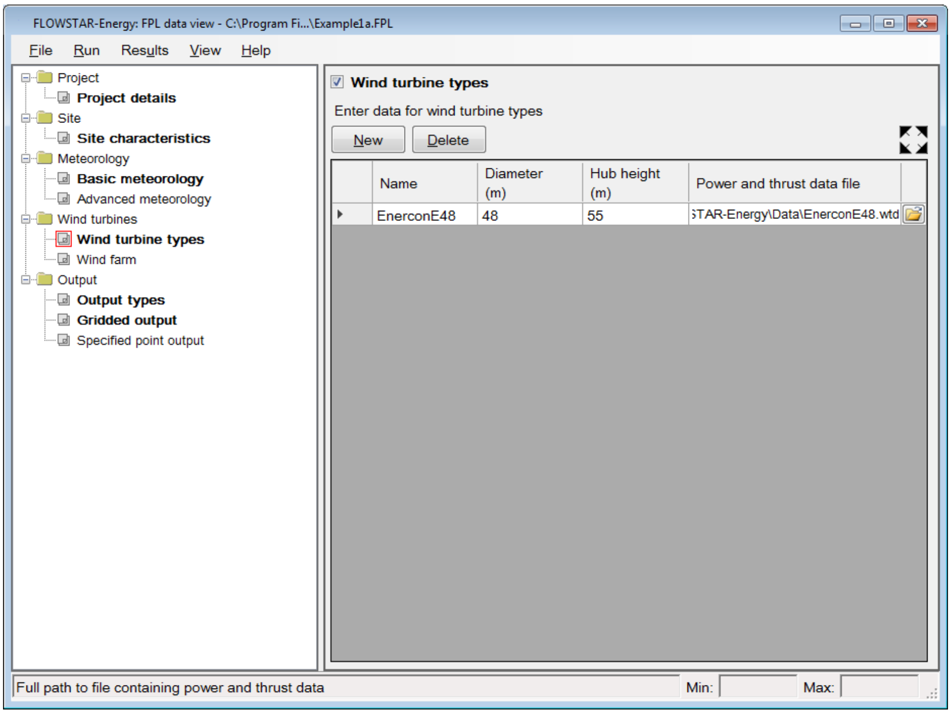



**Figure 6.4** − Example 1a: the **Wind turbine types** screen.

**Step 8** The only data left to enter in are in the **Output** folder. Firstly complete the **Output types** screen. The results of interest are **Flow field output** and **Potential wind energy output;** results are required for each meteorological condition so select **Per met condition** under **Type of results**. Select the **Turbine type** for the potential energy output from the drop-down list: choose **EnerconE48**. Compare your **Output types** screen with **Figure 6.5**.

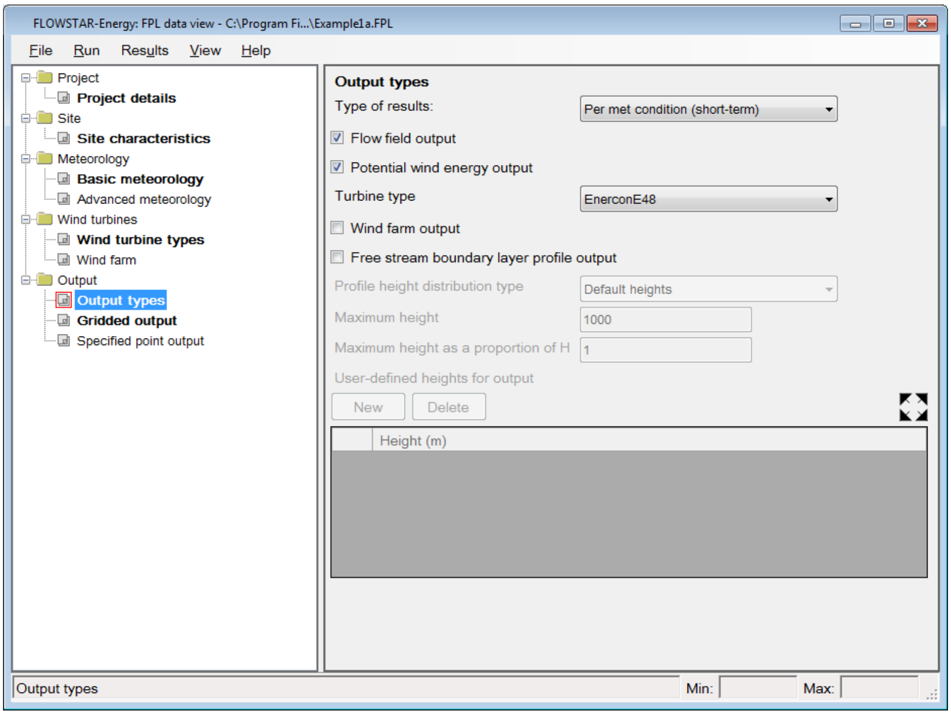



**Figure 6.5** − Example 1a: the **Output types** screen.

**Step 9** Finally specify the geographical area for which output is required. For this exercise it is appropriate to choose output at regular intervals over a rectangular area, so the **Gridded output** screen will be completed. This is an optional screen and should be enabled by checking the box at the top-right of the screen or by double-clicking the screen name in the left hand side. The area of interest for this exercise is covered in the x direction between 3000 and 9000, and in the y direction between 3500 and 11500. These coordinates are in metres. This example uses a study-specific coordinate system. The output resolution required is 100m in each direction, so specify 61 points in the x direction and 81 output points in the y direction. In the vertical direction specify that output should be generated at one z-value: the hub height, 55m. This height is relative to ground level. The input is shown in **Figure 6.6**.

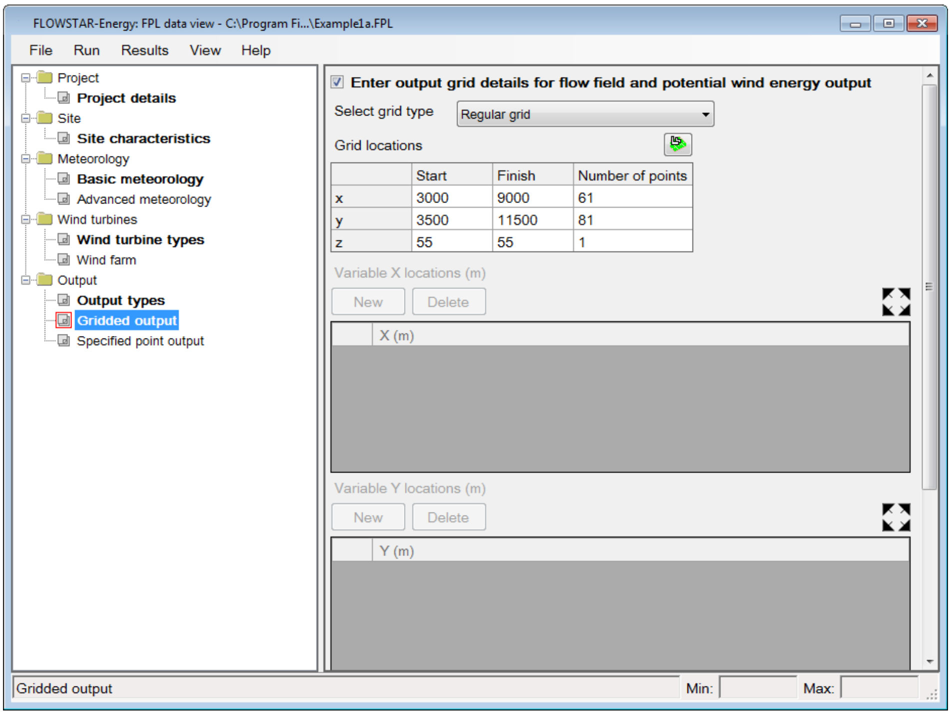



**Figure 6.6** – Example 1a: the **Gridded Output** screen.

Having now specified all the input required, you just need to save the configuration before running the model.

**Step 10** Click on **File**, **Save** or **Save As...** and save the file as *Example1a.fpl*.

**Step 11** Click on **Run** then **Model** to start the calculation.

The run will only take a few seconds for this example. A run window similar to **Figure 6.7** will appear. The file name is shown in the title bar of the window. The first messages to appear on the window are related to the licence details for the model, followed by messages related to the progress of the model run.

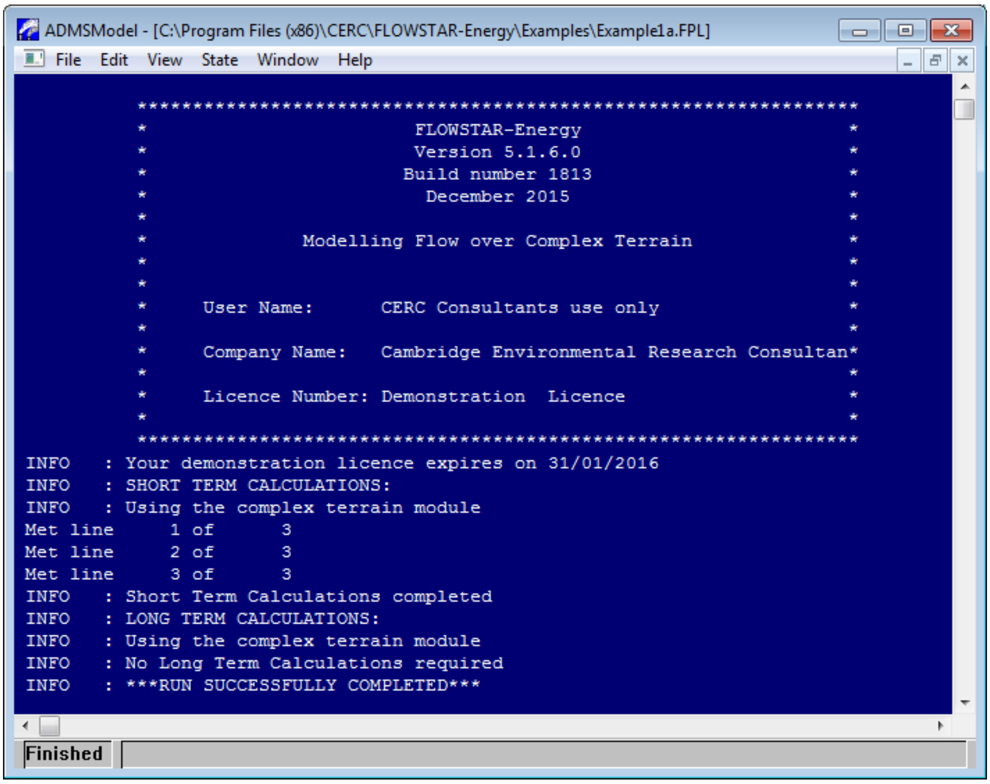



**Figure 6.7** − Example 1a: the run window.

At the end of the run the dialog box shown in **Figure 6.8** will appear. This means that the run has completed successfully.

**Step 12** Click on **Yes** to close the window shown in **Figure 6.8**.

Alternatively, click on **No** to leave the window open and view the screen messages. Use the **File, Exit** command from the menu bar of the run window to return to the **FPL data view** window.

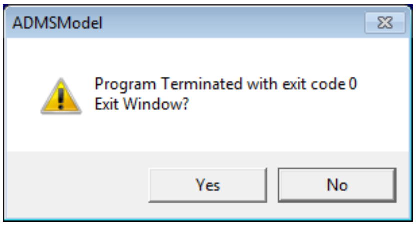



**Figure 6.8** − Example 1a: end-of-run dialog box.

## 6.1.2 Viewing output results

In FLOWSTAR-Energy, the results can be viewed in three ways, namely contour plots, flow field plots and numerically. Go to the main **map view** window to create contour plots or flow field plots.

### *Plotting the potential wind energy output*

The potential wind energy available for the modelled turbine type will be plotted over the area of interest as a contour plot for each of the three modelled wind speeds.

From the **map view** window, press the wind energy button , to open the **2-D Output Plotter** utility.

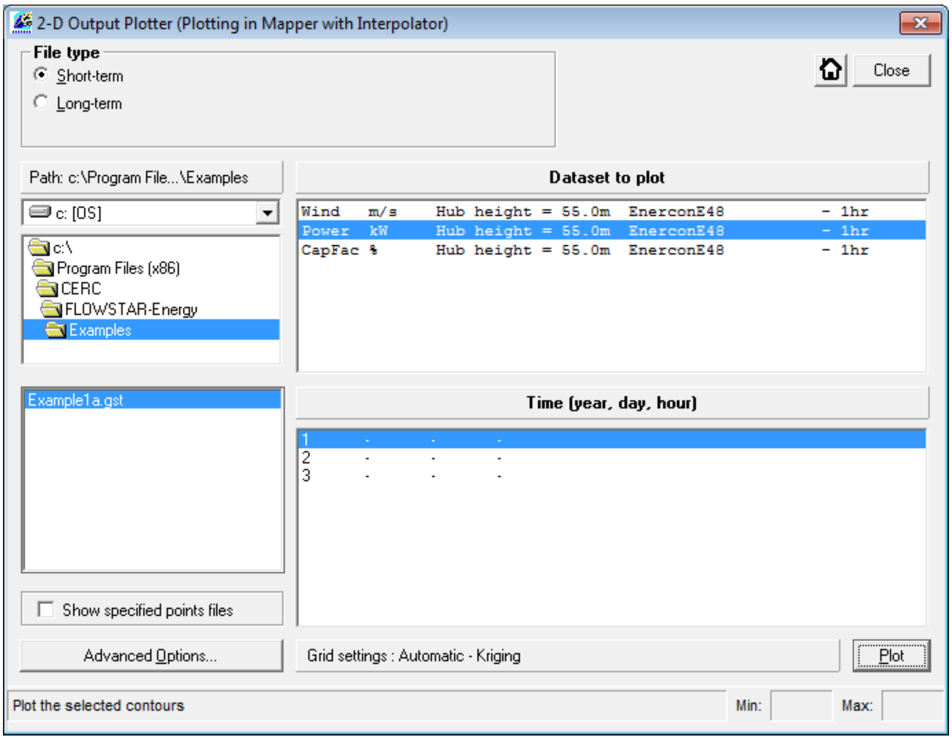



**Figure 6.9** – Example 1a: the **2-D Output Plotter** utility.

To create a contour plot for the run that has just completed, proceed as follows. All the information for contour plotting from this run is contained in a single output file, *Example1a.gst*. This file contains a data set of wind flow field, potential wind power and capacity factor for each line of meteorological data.

**Step 13** Select **Short term** and the correct output file, to produce a screen like that shown in **Figure 6.9**. The output file is created in the same directory where you saved the *Example1a.fpl* file earlier.

**Step 14** Highlight the **Time** and the **Dataset to plot** for the file you want to plot. (You can only make one plot at a time.)

Line 1 in the **Time** box at the bottom right of the screen corresponds to the first line of meteorological data (8m/s wind speed), line 2 to 10m/s wind speed and line 3 corresponds to the last line of data (12m/s wind speed).

**Step 15** Click on **Plot**.

This will first launch the **Save Surfer Grid File As...** dialogue box – save with the default name, in this case *Example1a.grd*. A contour plot of the chosen dataset and meteorological condition will be created in the **map view**. Position the contour plot layer above the terrain layer by clicking on the layer name in the **map view Legend** panel, and hold down the mouse button while dragging the layer above the terrain layer. **Figure 6.10** shows the automatically generated contour plot for potential wind power for line 1 (wind speed = 8m/s).

The plot can be edited to show more contour levels, change colours, etc. See Section 4.1 for more details.

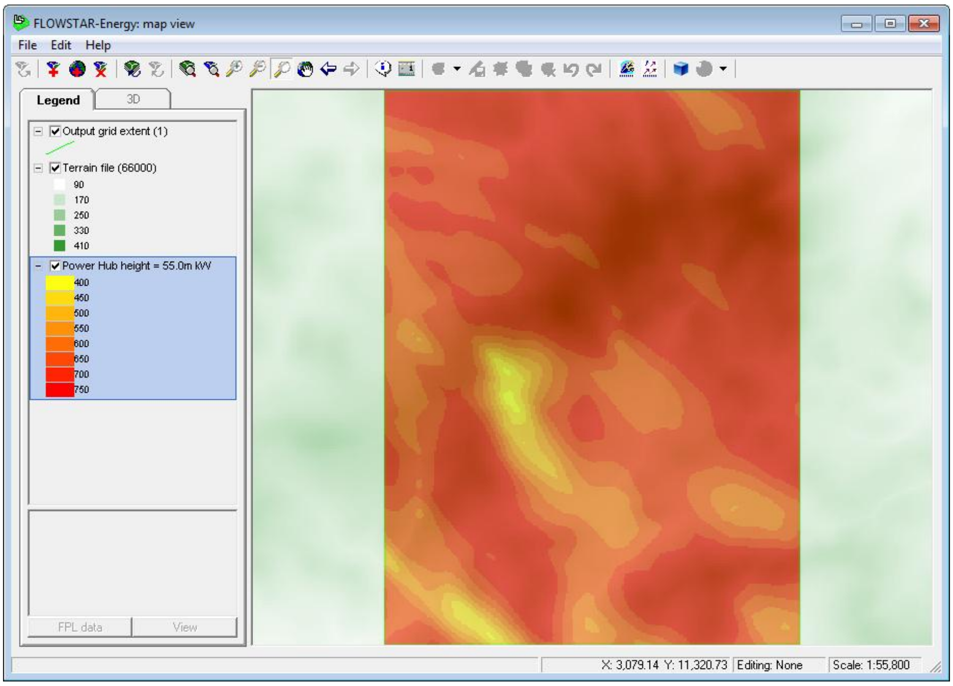



**Figure 6.10 −** Example 1a: potential wind power contour plot (line 1 of met. data).

You can click the wind energy button , again and repeat the steps to create plots from the other met lines or for other datasets. Each contour plot will be produced as a new layer in the **map view**, so that the output may be compared graphically for the different modelled wind speeds. It can be seen that for lower wind speeds a larger part of the study area fails to reach the turbine's rated power output of 810kW. For each of the three wind speed examples the maximum power output is reached at some locations in the study area.

### *Plotting the flow field output*

The flow field output can be plotted over the area of interest as a vector plot for each of the three modelled wind speeds. To create a flow field plot for the run that has just completed, proceed as follows.

**Step 16** From the **map view** window, press the flow field button , to open the **2-D Output Plotter** used to plot flow field output.

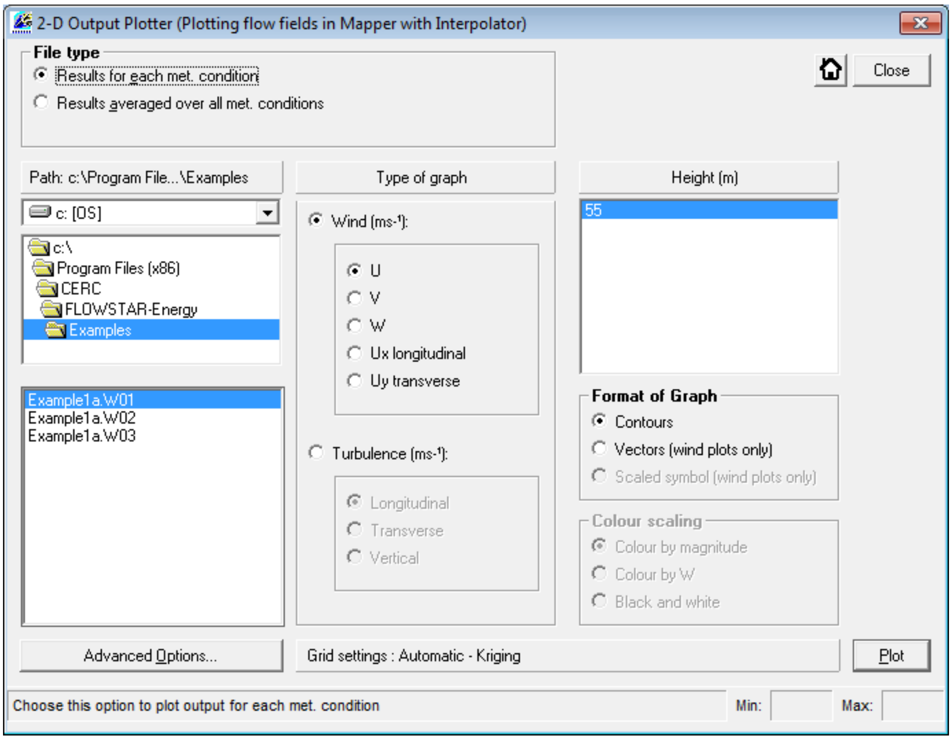



**Figure 6.11** – Example 1: the **2-D Output Plotter** utility for flow fields.

**Step 17** Select **Results for each met condition** and the correct output file for plotting, to produce a screen like that shown in **Figure 6.11**. All the information for flow field plotting from this run is contained in three output files in the same directory as the *.fpl* file: *Example1a.W01*, *Example1a.W02* and *Example1a.W03*. The numerical value at the end of the filename represents the meteorological line modelled. Selecting the file *Example1a.W01* will produce a flow field plot for meteorological line 1, i.e. the wind speed is 8m/s. This output file contains flow fields at a single height, 55m.

**Step 18** Choose the **Format of Graph** to be **Vectors (wind plots only)**.

Alternatively, instead of plotting vectors of the horizontal flow field, the individual components of the wind speed or turbulence can be plotted as a contour plot.

**Step 19** Click on **Plot**.

This will first launch the **Save Vector CSV File As...** dialogue box – save with the default name, in this case *Example1a.csv*. A vector plot is created in the **map view**, showing the horizontal flow field at the selected height, for that particular meteorological condition. **Figure 6.12** shows the flow field plot for line 1 (wind speed = 8m/s).

The plot can be edited to show more wind speed levels, change the colours or arrow sizes, reduce the density of the arrows, etc. See Section 4.1 for more details. The flow field layer in **Figure 6.12** has been moved above the terrain file layer and for clarity the number of arrows on the plot has been reduced.

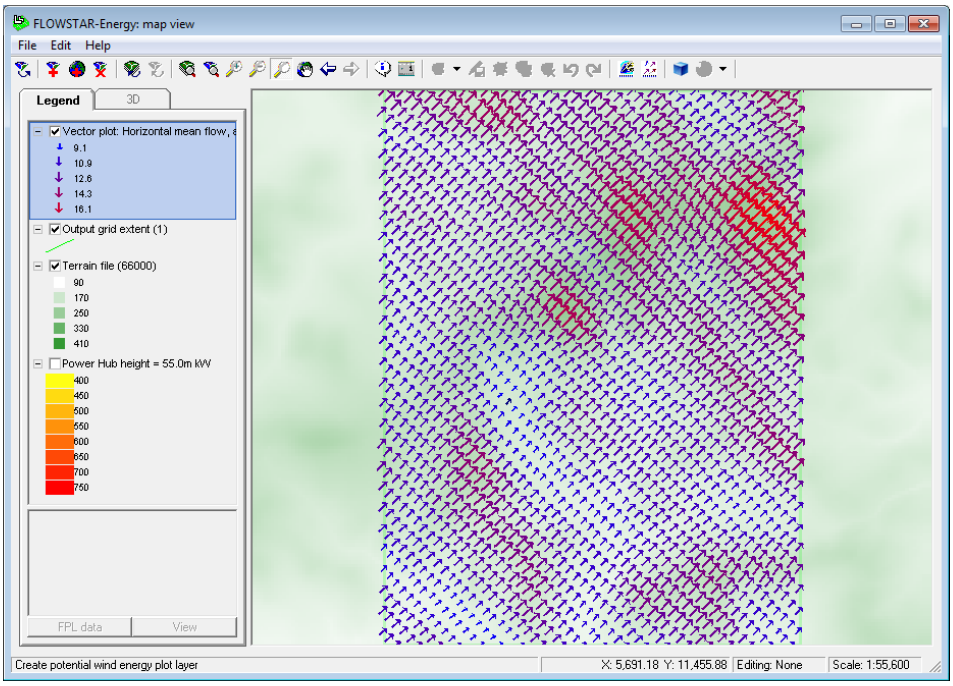



**Figure 6.12** − Example 1a: vector plot of wind flow (line 1 of met. data).

You can click the flow field button, , again and select the other met lines to plot. Each vector plot will be produced as a new layer in the **map view**, so that the output may be compared graphically for the different modelled wind speeds. It can be seen that the same pattern of flow is seen around the terrain, but the magnitude of the wind speed at all points increases with wind speed in the met line.

### 6.1.3 Potential wind energy for a statistical meteorological file

In practical applications a larger meteorological file would typically be used, containing *real recorded* data which is representative of the modelling area, over a particular length of time. For example a year of hourly measurements may be used, or statistical data could be used. In the following example statistical data in a *.tab* file is used to calculate the annual potential wind energy over the terrain.

Proceed as follows.

**Step 1** Open FLOWSTAR-Energy and click on **File**, **Open...** to browse to the previously saved input file, *Example1a.fpl*. Alternatively the file can be selected from **File**, **Recent Files**. Save this file under a new name, using **File**, **Save as...**, and then enter the new name *Example1b.fpl* and press **Save**.

**Step 2** Open the **Basic Meteorology** screen and change the meteorological file to *Example.tab* as shown in **Figure 6.13**. Leave the number of wind direction increments as 5.

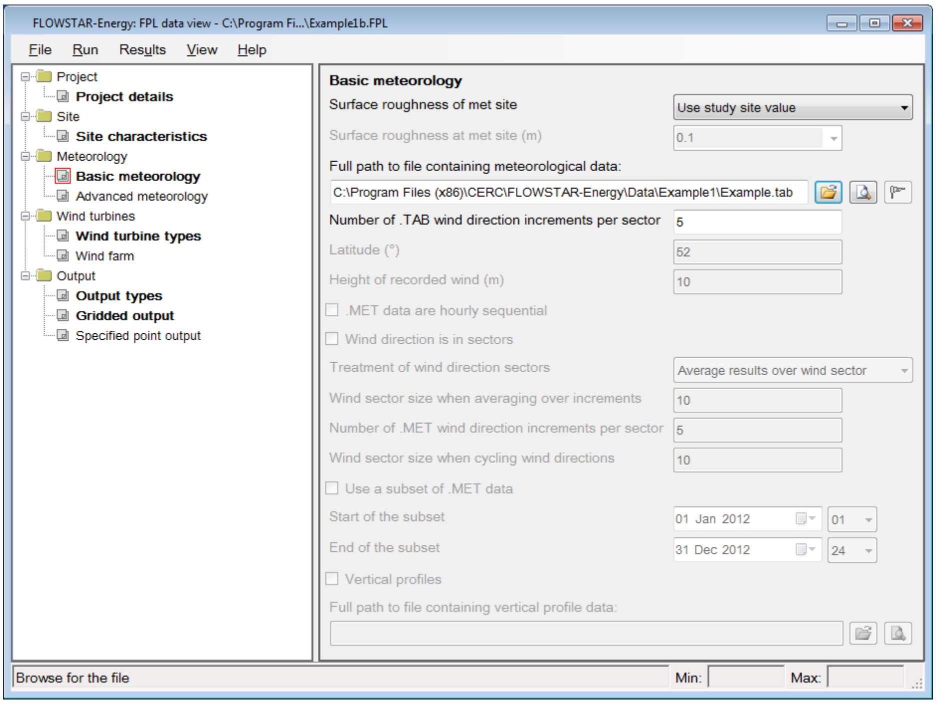



**Figure 6.13** – Example 1b: **Basic meteorology** screen showing the newly entered *.tab* file.

**Step 3** For this example only **Summary results (long term)** are of interest, and the only required output type is the **Potential wind energy output**. Edit the **Output types** screen to reflect the required changes, as shown in **Figure 6.14**.

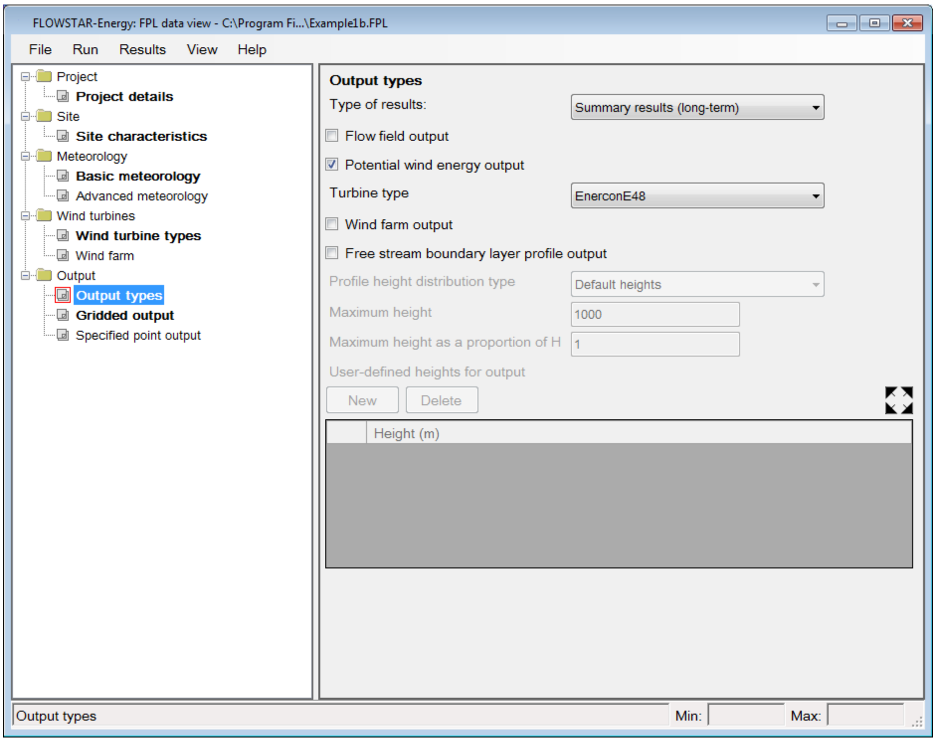



**Figure 6.14** – Example 1b: **Output types** screen showing the changes.

**Step 4** The calculation will take a longer time than the first calculation, as 370 separate meteorological conditions need to be modelled. In order to speed up the model run for this example, change the **Hills calculation grid resolution** on the **Site characteristics** screen to 32x32 points.

**Step 5** Save the file by selecting **File, Save** and run the model by selecting **Run, Model**. The run may take up to twenty minutes to complete.

**Step 6** A wind rose is a useful visualisation of the frequencies of wind speeds and directions in your meteorological data. When a *.tab* file is used a wind rose can only be produced after a model run has been completed. To plot the wind rose, within the **FPL data view** window go to **Results, Plot wind rose of processed met data**. The wind rose is shown in **Figure 6.15**. The prevailing wind can be seen to be from the south-west.

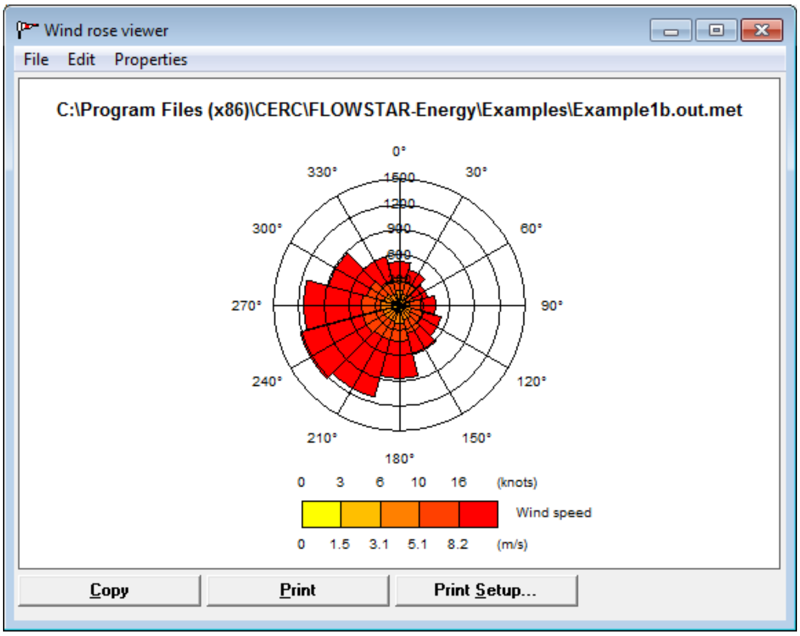



**Figure 6.15** − Example 1b: Wind rose of processed met data.

**Step 7** Create a contour plot of annual potential wind energy for the new run. From the **map view** window, press the wind energy button, , to open the **2-D Output Plotter**. Select **Long-term** and select the output file; choose the **Dataset to plot** (Energy kWh) and click **Plot**. It may take a few seconds for the plot to be created.

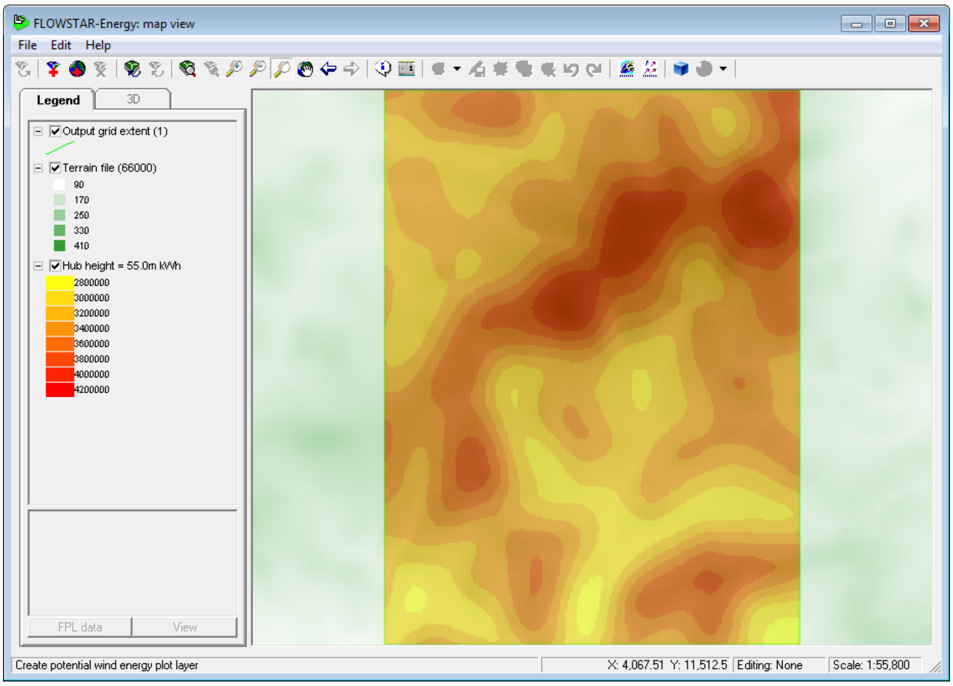



**Figure 6.16 −** Example 1b: Contour plot of annual potential wind energy

**Figure 6.16** shows that in this example the highest annual potential energy is found on top of the highest terrain in the modelled region.

### 6.1.4 Adding a wind farm to the calculation

The modelling carried out so far has shown the areas over the terrain where a single turbine would produce the highest annual energy yield. In the next example the energy production from a small wind farm will be calculated for a single hour, using specific locations of a number of turbines and modelling their wake interactions as well as the influence of the terrain.

Proceed as follows.

**Step 1** Open FLOWSTAR-Energy and click on **File**, **Open...** to browse to the previously saved input file, *Example1b.fpl*. Save this file under a new name, using **File, Save as...**, and then enter the new name *Example1c.fpl* and press **Save**.

**Step 2** In order to reduce the run time for this example only one meteorological line will be modelled; go to the **Basic meteorology** screen and change the file to *270demo.met*, which uses a wind speed of 10m/s.

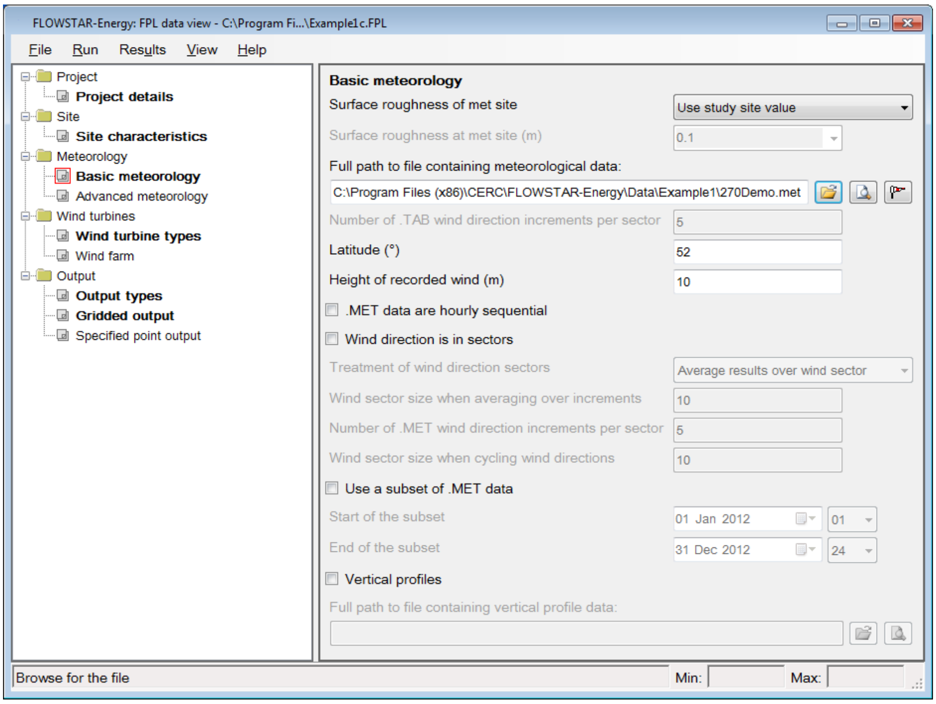



**Figure 6.17** − Example 1c: The **Basic meteorology** screen.

**Step 3** Within the **FPL data view**, go to the **Wind farm** screen and tick the **Wind farm** option, as shown in **Figure 6.18**.

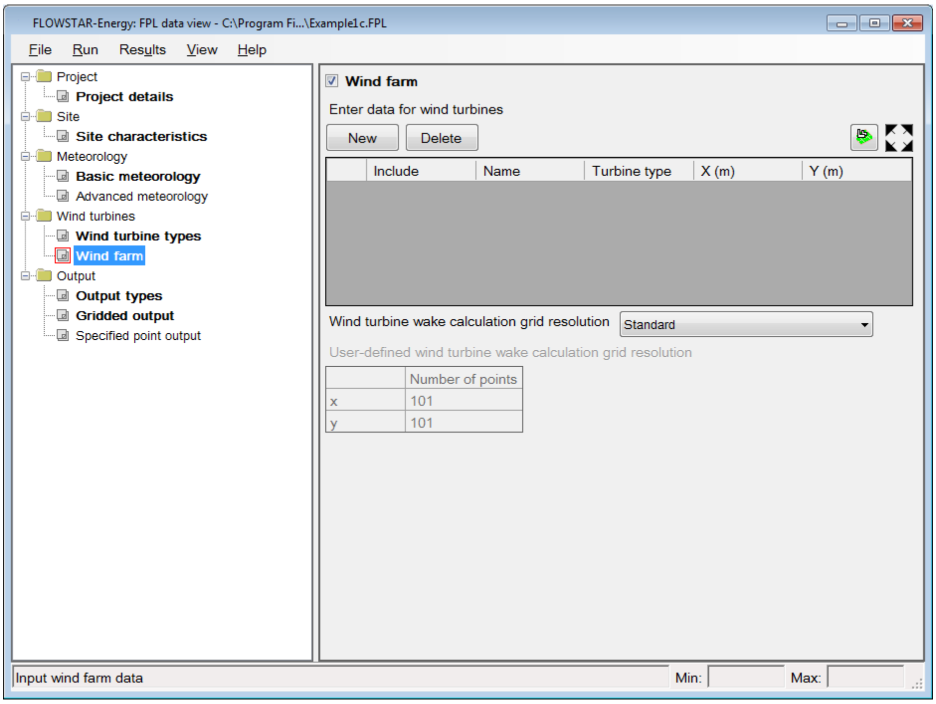



**Figure 6.18** − Example 1c: The **Wind farm** screen.

**Step 4** Add three wind turbines to the calculation; the turbine locations are shown in Table 6.1. These locations are in the areas of highest calculated annual potential wind energy.

| Turbine Name | X coordinate | Y coordinate |
|---|---|---|
| Turbine 1 | 8300 | 9800 |
| Turbine 2 | 6700 | 9600 |
| Turbine 3 | 5600 | 8400 |

Table 6.1 – Wind farm turbine co-ordinates

The turbines can be added either by clicking the **New** button and typing the details into the table below, or via the **map view**. To do the latter, click the button above the turbines table, , and in the **map view**, ensure that the **Add feature** tool, , is active. You may need to click on the tool to activate it. Move the mouse until the coordinates at the bottom right of the **map view** window match those required in Table 6.1. Then click the mouse over the map; the **FPL data view** will become active. The turbine coordinates and name can then be edited to match Table 6.1, giving the input as shown in **Figure 6.19**. In a practical study you might use background maps in the **map view** to help you in position your turbines; or you might copy-and-paste the data into the **FPL data view** table from another package such as Excel.

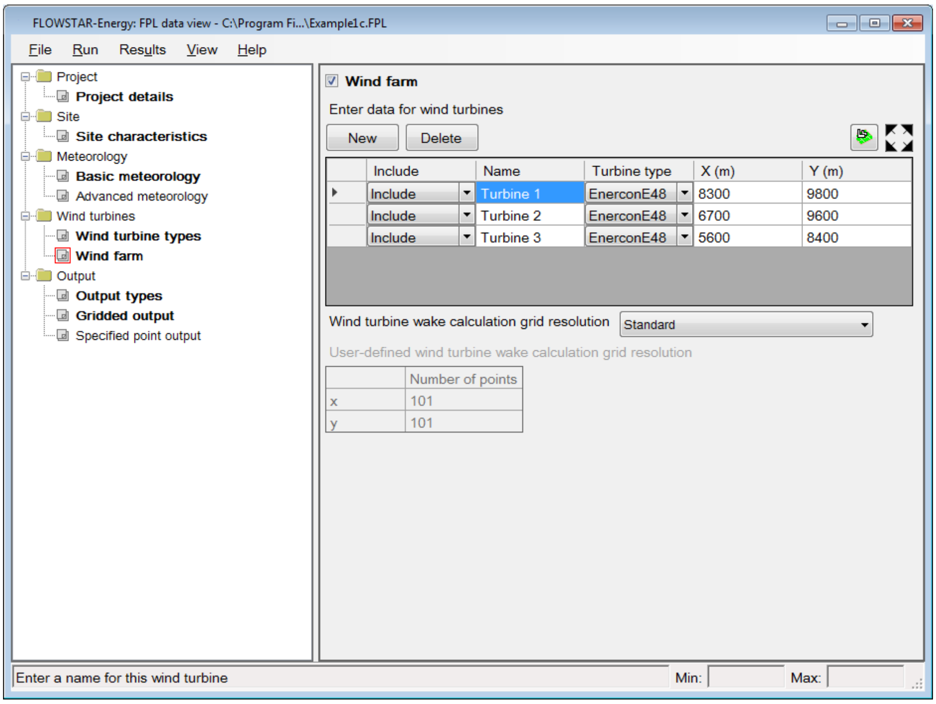



**Figure 6.19** – Example 1c: The **Wind farm** screen with wind turbine data added.

The turbine data can be refreshed in the **map view** by highlighting the layer and pressing the **Refresh layer** button, .

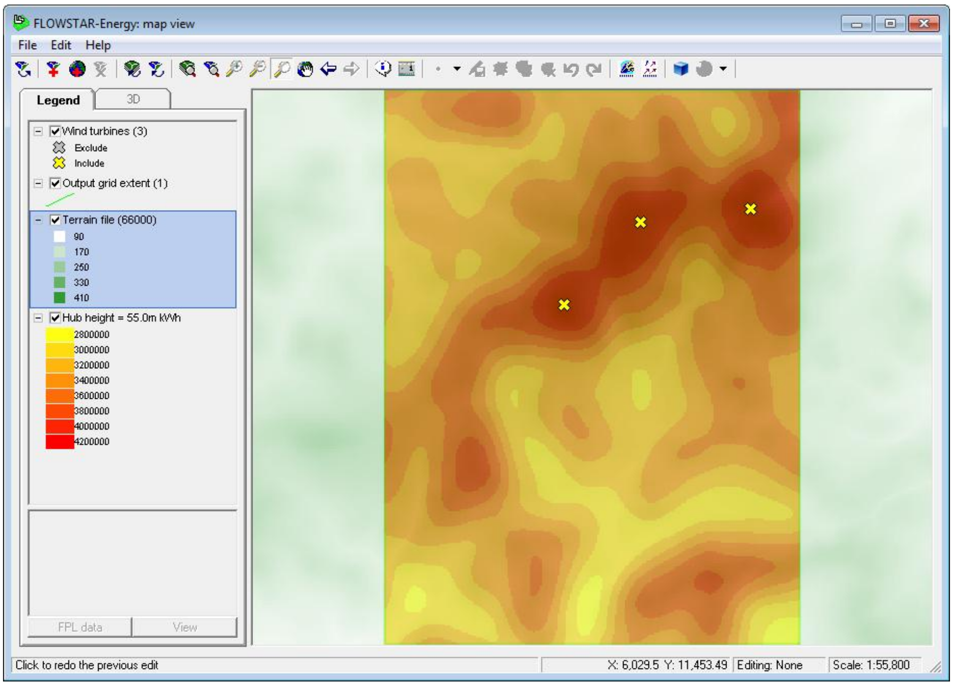



**Figure 6.20** − Example 1c: The wind turbines shown over the annual potential energy map in the **map view**.

**Step 5** Go to the **Output types** screen in the **FPL data view**, deselect **Flow field output**, select **Wind farm output**, and also **Flow field output** as shown in **Figure 6.21**.

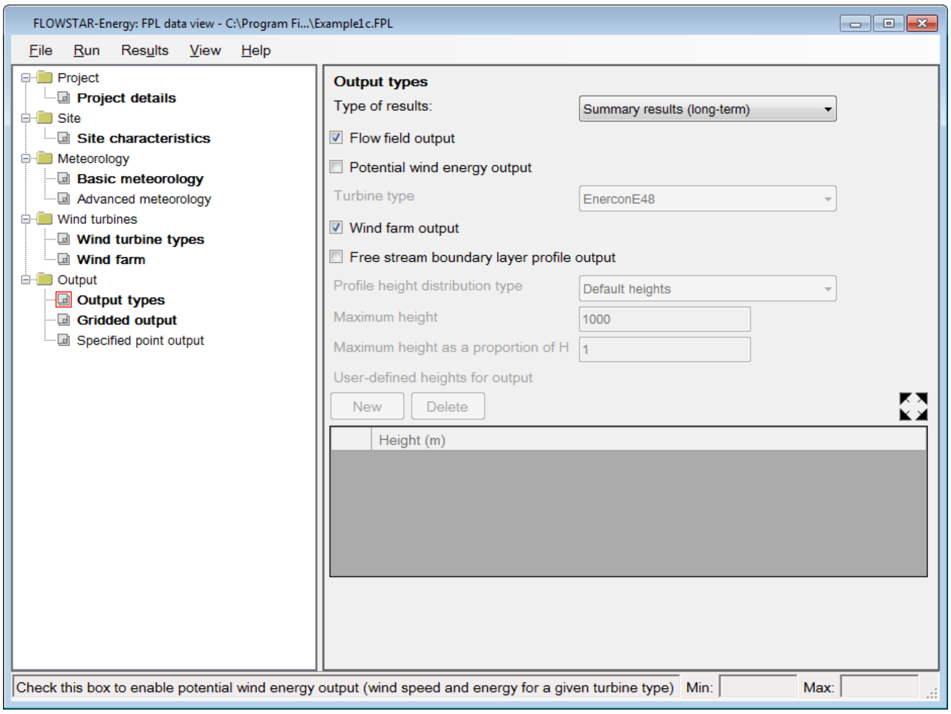



**Figure 6.21** − Example 1c: The **Output types** screen.

**Step 6** Save the current model input by selecting **File, Save** then run by selecting **Run, Model.** This run should take only a few seconds to complete.

**Step 7** The numerical results from the calculation will be inspected. From the **FPL data view**, select **Results, View long-term wind farm results.**

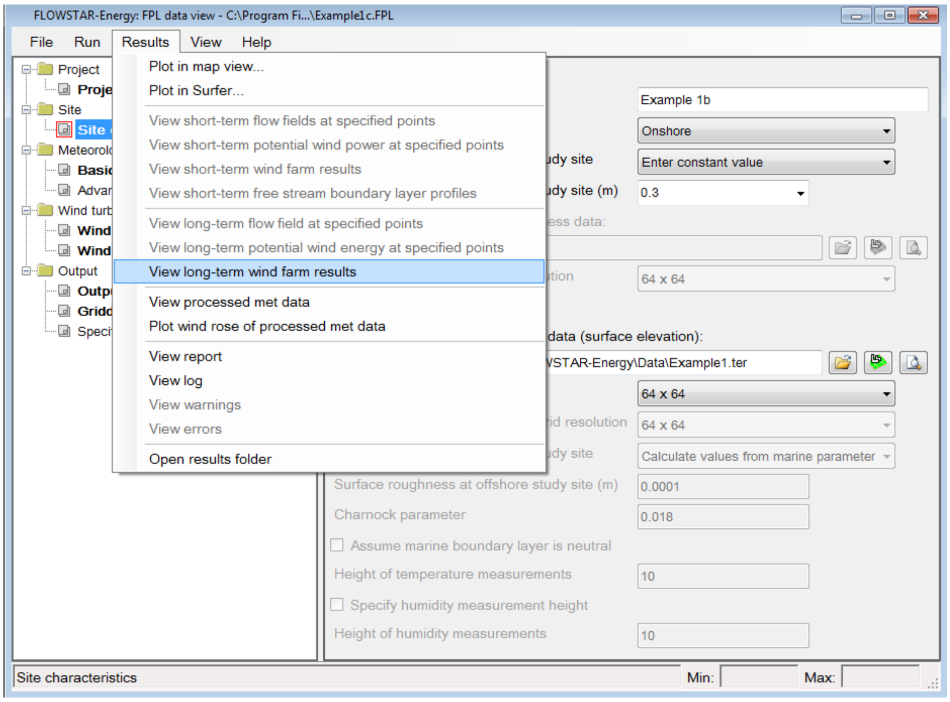



**Figure 6.22** − Example 1c: The **Results** menu.

**Step 8** The file *Example1c.wfl* will open in the default viewing software. You can change the viewing software under **File**, **Preferences**.... **Figure 6.23** shows the results in Microsoft Excel.

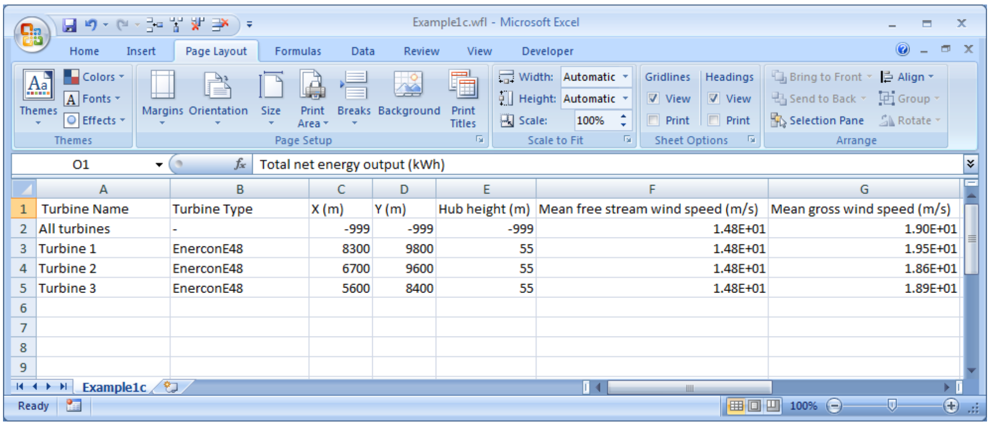



**Figure 6.23** − Example 1c: The wind farm results.

**Step 9** In this example we are interested in the total energy production from the turbines, this is found under the column headed **Total net energy output (kWh)**. It can be seen in this example that the array efficiency for all turbines is 100%, which implies that the three turbines are positioned in such a way as to achieve maximum output for this particular meteorological condition.

# 6.2 Example 2: Offshore calculations

The object of this example is to demonstrate how to set up and run an offshore (marine) modelling study and to look at the different types of output available from FLOWSTAR-Energy.

## 6.2.1 Setting up the run

To model and examine the flow field in an offshore location, proceed as follows.

**Step 1** Open FLOWSTAR-Energy and click on **File**, **New** to create a new input file, i.e. a default input file or "clean sheet" in which to enter data.

**Step 2** Enter a **Name of project** and **Name of site** of your choice on the **Project details** and **Site characteristics** screens.

**Step 3** Set the **Site type** to **Offshore**, and choose to calculate the surface roughness from marine parameters. For this study the other site parameters can be left at the default values. The **Site characteristics** screen should be similar to that shown in **Figure 6.24**.

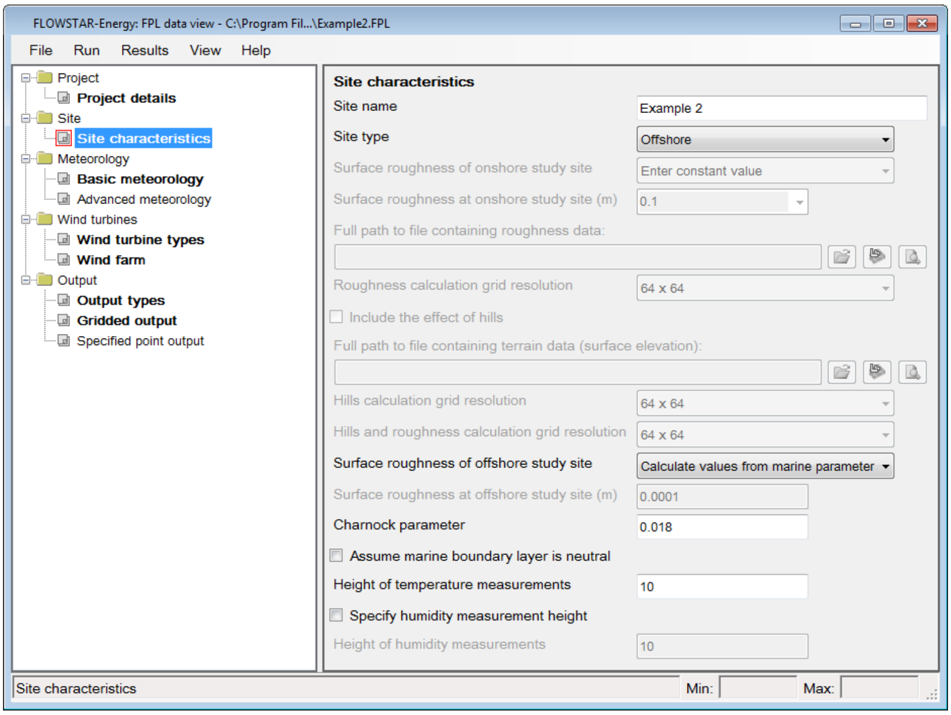



**Figure 6.24** – Example 2: The **Site characteristics** screen.

**Step 4** The met file to use in this example is called *marine.met* and can be found in your *<install path>/Data/Example2* directory. This met file contains one day of hourly sequential data, a wind rose for the day is shown in **Figure 6.25**. Select this file in the **Basic meteorology** screen, as shown in **Figure 6.26**.

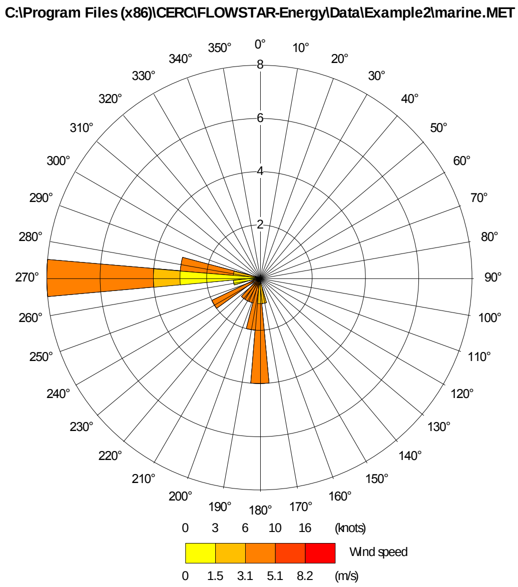



**Figure 6.25** − Example 2: Wind rose created using *marine.met*.

The other meteorological parameters can be left at their default values.

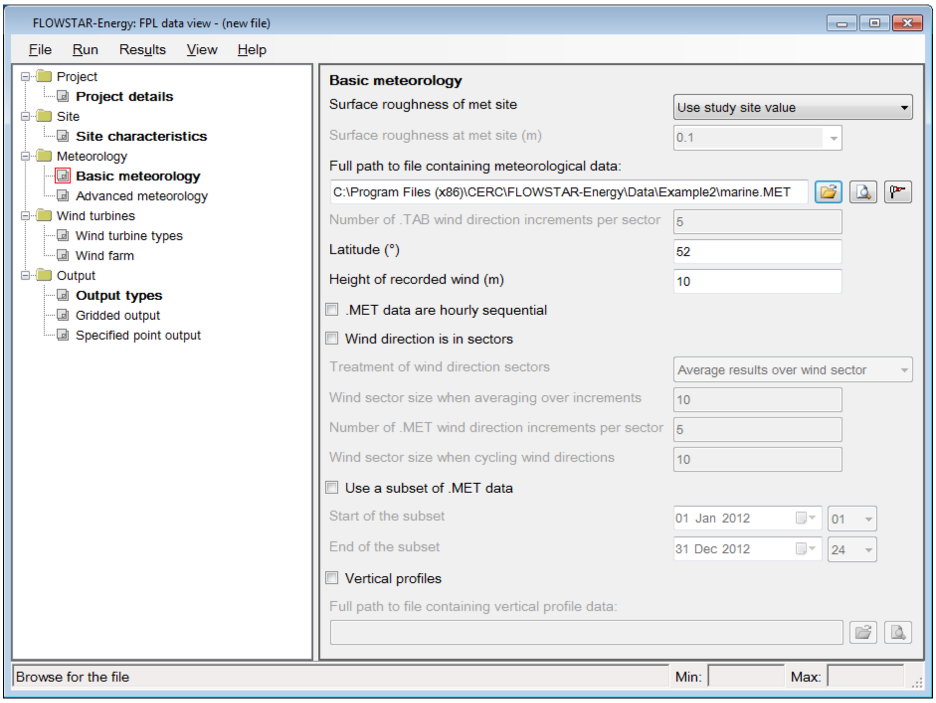



**Figure 6.26** − Example 2: The **Basic meteorology** screen.

**Step 5** The same wind turbine type modelled in example 1 will be used in this example. Enable the **Wind turbine** types screen and enter the necessary data, as shown in **Figure 6.27**. The **Power and thrust data file** *EnerconE48.wtd* can be found in your *<install path>/Data/Example2* directory.

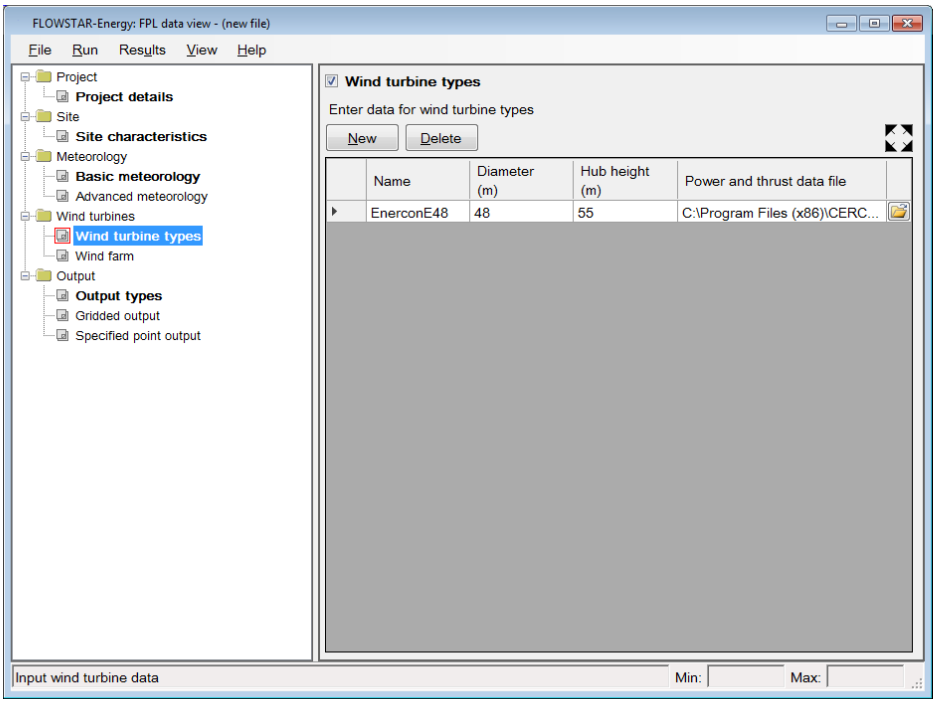



**Figure 6.27** – Example 2: The **Wind turbine types** screen.

**Step 6** Next enable the **Wind farm** screen.

In this example there are nine identical turbines evenly-spaced in a rectangular 3x3 grid with 300m between each turbine, as shown in **Table 6.2**. The steps below show how to copy-and-paste the data from an Excel spreadsheet into the **FPL data view**. If you prefer, you can type the data from **Table 6.2** into the **FPL data view** rather than copying-and-pasting from a spreadsheet package.

| Turbine Name | X-coordinate | Y-coordinate |
|---|---|---|
| 0_0 | 0 | 0 |
| 300_0 | 300 | 0 |
| 600_0 | 600 | 0 |
| 0_300 | 0 | 300 |
| 300_300 | 300 | 300 |
| 600_300 | 600 | 300 |
| 0_600 | 0 | 600 |
| 300_600 | 300 | 600 |
| 600_600 | 600 | 600 |

**Table 6.2** – Wind farm details

**Step 7** The Excel spreadsheet file to use in this example is called *Example2.xlsx* and can be found in your *<install path>/Data/Example2* directory. Open the spreadsheet in your preferred spreadsheet package, select the data and copy it to the clipboard. **Figure 6.28** shows the data selected ready for copying in Microsoft Excel 2007.

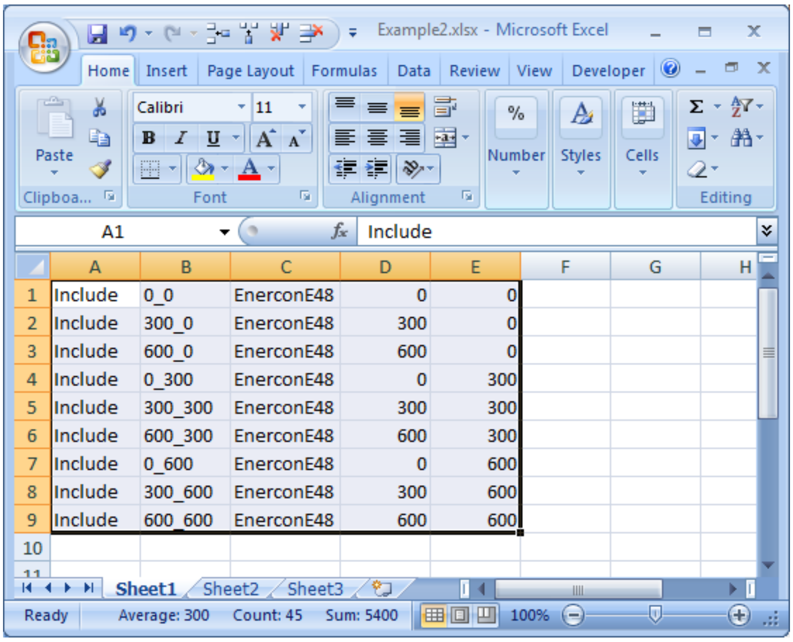



**Figure 6.28** – Example 2: *The Example2.xlsx* spreadsheet in Microsoft Excel 2007, with the data selected ready to be copied into FLOWSTAR-Energy

**Step 8** In the **FPL data view**, right-click in the wind farm table and choose **Paste**. The data will appear in the table.

---

*If you have difficulty pasting the data, this could be because the turbine type name entered on the **Wind turbine types** screen does not match the name in the Excel spreadsheet.*

---

If there are a large number of turbines in your study it can useful to edit the **Wind farm** details using full screen editing; this is accessed by pressing the button . Full screen editing for the **Wind farm** screen is shown in **Figure 6.29**. Click the **Close** button to exit from full-screen editing. You must leave full-screen editing when you wish to use the **map view** or use another screen in the **FPL data view**.

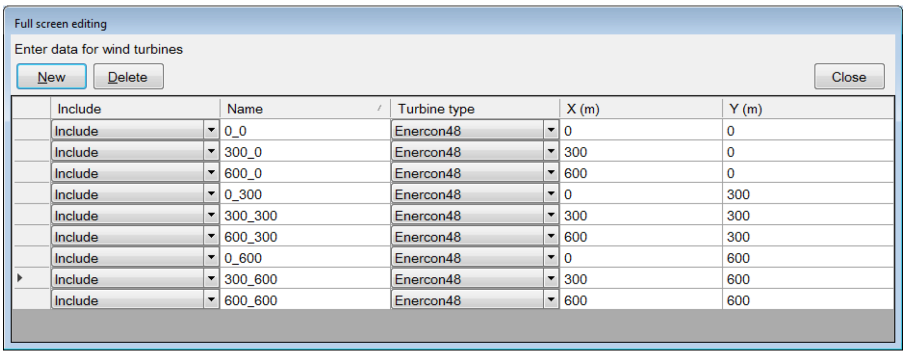



**Figure 6.29** – Example 2: The **Wind farm** screen, full screen view.

**Step 9** On the **Output types** screen, choose **Flow field** output and **Wind farm output** as **Summary results**, as shown in **Figure 6.30**.

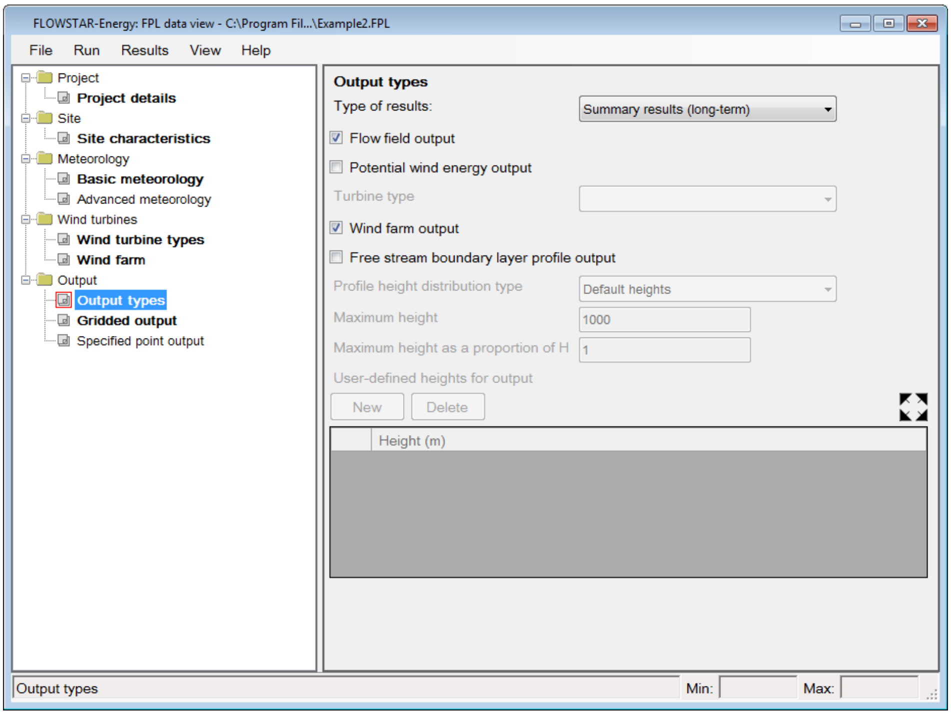



**Figure 6.30** – Example 2: The **Output types** screen.

**Step 10** On the **Gridded output** screen select a regular grid. Set the **x Start** value to -300 and the **Finish** value to 800. Use these same values for the **y Start** and **Finish** values. Set the output height to 55m, as shown in **Figure 6.31**.

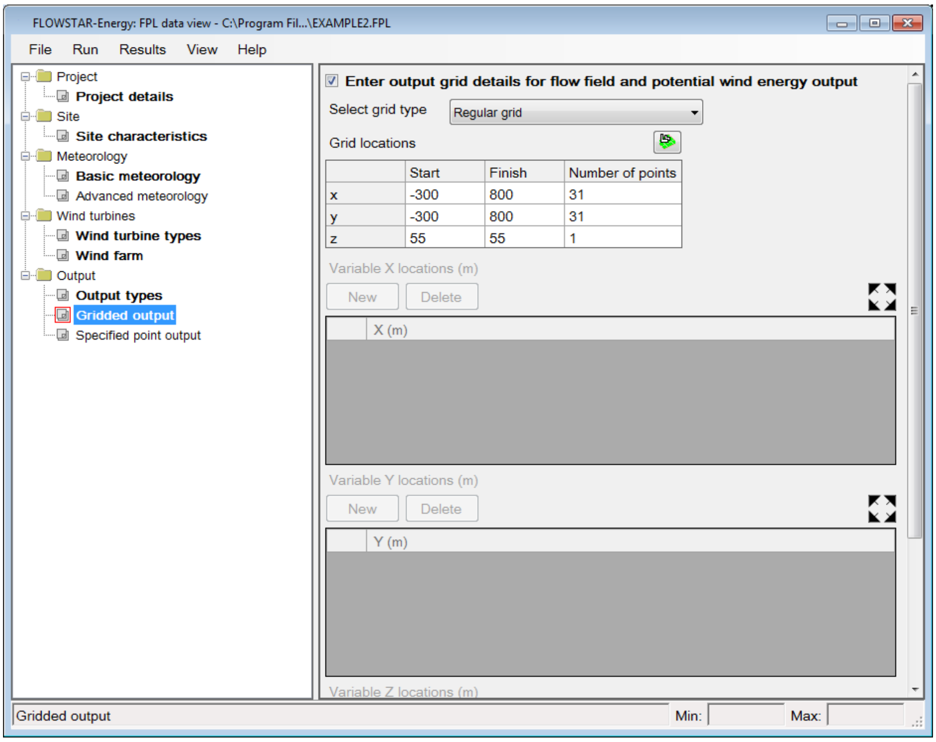



**Figure 6.31** – Example 2: The **Gridded output** screen.

**Step 11** Save the current model input by selecting **File, Save** then run by selecting **Run, Model.** This example will take less than one minute to run.

**Step 12** The next step is to plot a flow field vector plot in the **map view**. Click the flow field button, , to open the **2-D Output Plotter**. In the plotter select the file type **Results averaged over all met conditions**, and highlight the file *Example2.wlt*. Under **Format of graph** choose **Vectors (wind plots only)**, as shown in **Figure 6.32**. Then press the **Plot** button to create the flow field vector plot shown in **Figure 6.33**.

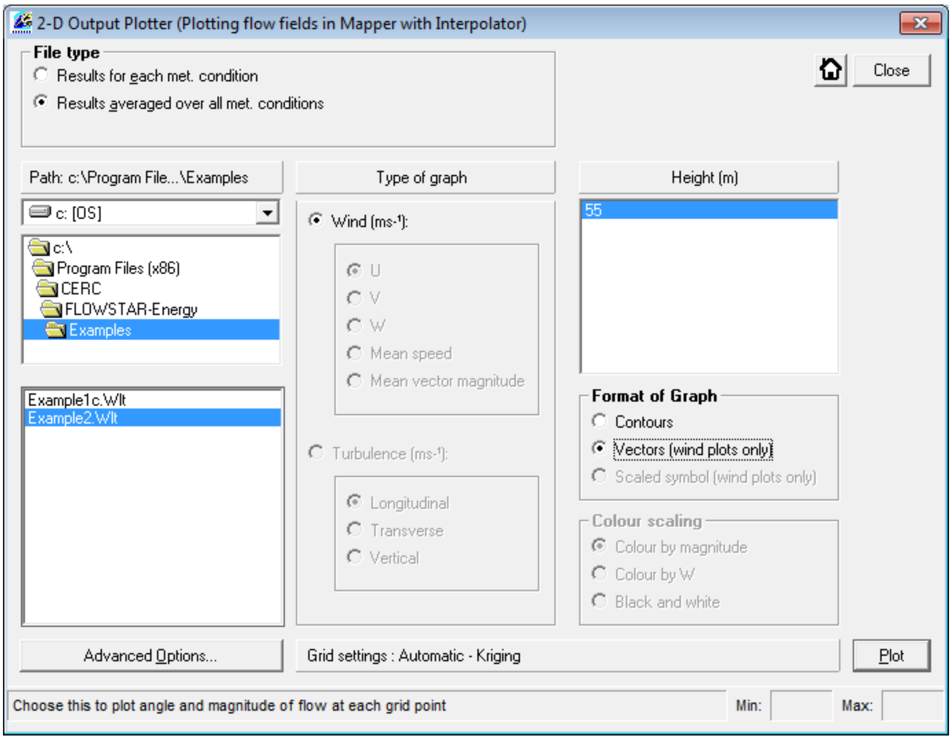



**Figure 6.32** − Example 2: The **2-D Output Plotter** screen.

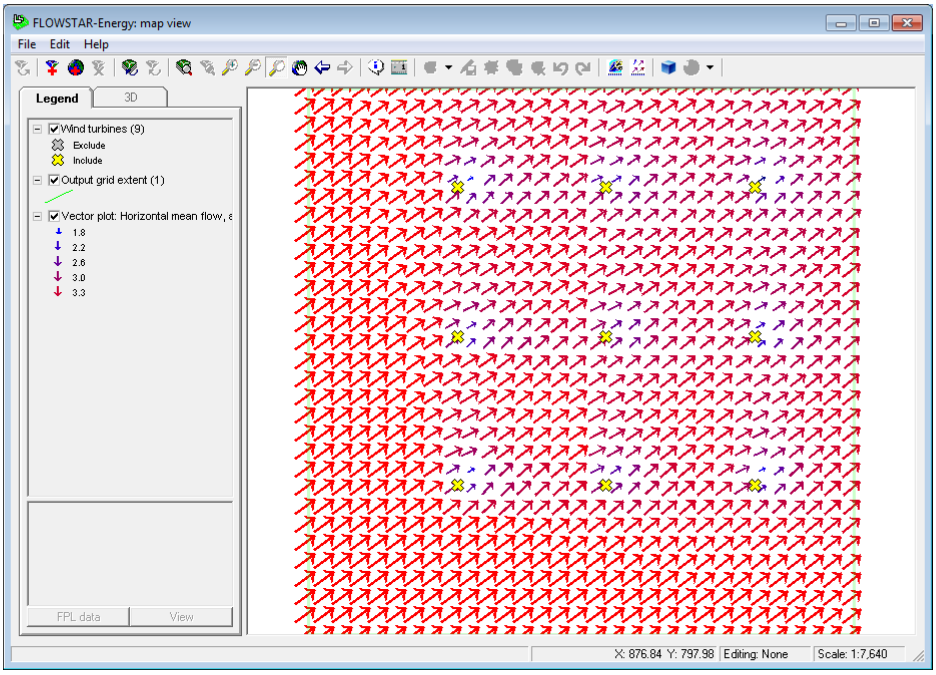



**Figure 6.33** − Example 2: The flow field output vector plot.

Notice how the wind vectors show the average wind direction to be from the south west. The turbines are shown with a yellow cross symbol. The average wind speed decreases in the wake of each wind turbine; the downwind turbines therefore experience a lower mean wind speed than the upstream turbines.

**Step 13** The next step is to look at estimates of the wind farm performance. In the **FPL data view** window select **Results, View long-term wind farm results**. Some of the data are repeated in **Table 6.3** below. The column labelled **Mean net wind speed deficit (%)** shows the percentage by which the wake effects of the other turbines are decreasing the mean wind speed reaching each turbine. The efficiency of each turbine is shown in the column labelled **Array efficiency (%).** This shows that the turbine in the centre of the grid at point (300,300) is the least efficient. The row labelled **All turbines** contains average wind speed and total energy values over all the turbines in the wind farm.

| Turbine Name | Mean net wind speed deficit (%) | Array efficiency (%) |
|---|---|---|
| All turbines | 7.66 | 76.79 |
| 0_0 | 0.00 | 100.00 |
| 300_0 | 6.47 | 80.06 |
| 600_0 | 5.37 | 82.55 |
| 0_300 | 4.77 | 86.79 |
| 300_300 | 12.89 | 62.10 |
| 600_300 | 11.77 | 63.76 |
| 0_600 | 3.93 | 88.89 |
| 300_600 | 12.14 | 63.29 |
| 600_600 | 11.57 | 63.69 |

**Table 6.3** - Example 2: The wind farm results.

# SECTION 7 Technical Summary

This section is intended to provide a summary of the mathematical and physical background to FLOWSTAR-Energy, and is essentially an abbreviated form of the Technical Specification documentation (CERC, 2015), to which readers should refer for full details.

## 7.1 Meteorological input and output

Section 3.3 describes the types of meteorological data that can be input to FLOWSTAR-Energy. For all the data types, each line of input data describes the meteorological conditions for one or more hours, the conditions being assumed to be fixed for any given hour. The meteorology input module reads the data and calculates values of the various meteorological quantities required for running the simulation. Data may be in the form of a chronological record, which is termed 'hourly sequential'. Other data may have a certain weight or frequency associated with each line of meteorological data. Such data are usually non-chronological and are termed 'statistical' as they have been statistically analysed. It is also possible that the data are neither sequential nor statistical, but some more general collection of one or more meteorological conditions.

FLOWSTAR-Energy can be used with meteorological data files that are in one of two formats: *.met* and *.tab*. Section 7.1.1 describes the treatment of *.met* file data; Section 7.1.2 describes the treatment of *.tab* file data. For detailed information about the format of these file types the reader should refer to Section 3.3.

### 7.1.1 Meteorological data input from a *.met* file

The complete list of meteorological parameters that may be input in a *.met* file is shown in **Table 7.1**. The first three columns are alternative forms of the variable names that may be used in creating *.met* files. As FLOWSTAR-Energy matches the name to identify a variable the user may want to use short versions of the variable names (columns 1 and 3) in order to avoid making errors in typing. Names from different lists may be used in the same *.met* file, e.g. using U for wind speed and CLOUD for cloud cover in the same file is acceptable. The fourth column shows the units in which the data should be entered, the fifth and sixth columns show the minimum and maximum values for the data and the seventh column shows whether the variable can be interpolated.

It is likely that the two most common types of *.met* input data will be: hourly sequential data with wind speed, direction, cloud cover, time of day, time of year, temperature and possibly sea surface temperature; or statistical data, comprising wind speed, direction, surface sensible heat flux and boundary layer height. Such data are supplied by the Met Office for the main UK met. stations.

Note that Pasquill-Gifford (P-G) stability categories cannot be directly input into the model (although values of $U$, $\phi$, $L_{mo}$ and $h$ may be), nor are they output. In FLOWSTAR-Energy the boundary layer structure is characterised by the two parameters, $h$ and $L_{MO}$. Values of these parameters corresponding *approximately* to P-G categories are shown in the data file *r91a-g.met* and in **Table 7.2**.

| Short name | Long name | Abbreviated name | Units | Min value | Max Value | Can Interpolate?* |
|---|---|---|---|---|---|---|
| WIND SPEED | WIND SPEED | U | m/s | 0.0 | 100.0 | R |
| UG/USTAR | GEOSTROPHIC WIND SPEED/FRICTION VELOCITY | UGSTAR | - | 5.0 | 1000.0 | N |
| WIND DIRN | WIND DIRECTION (DEGREES) | PHI | ° | 0.0 | 360.0 | R |
| DIRN CHANGE | GEOSTROPHIC MINUS SURFACE WIND DIRECTION (DEGREES) | DELTAPHI | ° | -60.0 | 60.0 | N |
| HEAT FLUX | SENSIBLE HEAT FLUX | FTHETA0 | $W/m^2$ | -100.0 | 1000.0 | N |
| 1/LMO | 1/MONIN-OBUKHOV LENGTH | RECIPLMO | $m^{-1}$ | -10.0 | 10.0 | N |
| BL DEPTH | BOUNDARY LAYER DEPTH | H | m | 40.0 | 4000.0 | N |
| CLOUD | CLOUD AMOUNT (OKTAS) | CL | oktas | 0.0 | 8.0 | R |
| SOLAR RAD | INCOMING SOLAR RADIATION | SOLAR RAD | $W/m^2$ | 0.0 | 1500.0 | N |
| TEMPERATURE | TEMPERATURE (C) | T0C | °C | -100.0 | 60.0 | Y |
| N ABOVE BL | BUOYANCY FREQUENCY ABOVE BOUNDARY LAYER | NU | $s^{-1}$ | 0.0 | 0.1 | Y |
| DELTA THETA | TEMPERATURE JUMP ACROSS BOUNDARY LAYER TOP | DELTATHETA | °C | 0.0 | 25.0 | N |
| PRECIP | PRECIPITATION RATE (MM/HOUR) | P | mm/h | 0.0 | 500.0 | Y |
| SEA TEMP | SEA SURFACE TEMPERATURE (C) | TSEA | °C | -10.0 | 40.0 | Y |
| DELTA T | TEMPERATURE OVER LAND MINUS SEA SURFACE TEMPERATURE | DELTAT | °C | -40.0 | 40.0 | Y |
| SIGMA THETA | SIGMA THETA (DEGREES) | SIGMATHETA | ° | 0.0 | 90.0 | Y |
| S HUMIDITY | SPECIFIC HUMIDITY | S HUMIDITY | kg/kg | 0.0 | 0.1 | Y |
| R HUMIDITY | RELATIVE HUMIDITY (PERCENT) | RHUM | % | 0.0 | 100.0 | Y |
| RH ABOVE BL | RELATIVE HUMIDITY ABOVE BOUNDARY LAYER (PERCENT) | RH ABOVE BL | % | 0.0 | 100.0 | Y |
| DRH/DZ | D(RELATIVE HUMIDITY)/DZ ABOVE BOUNDARY LAYER (PERCENT/M) | DRH/DZ | %/m | -10.0 | 10.0 | N |
| LAT HT FLUX | LATENT HEAT FLUX | LAT HT FLUX | $W/m^2$ | -100.0 | 1000.0 | N |
| Z0 (M) | ROUGHNESS LENGTH (MET SITE) | Z0 (M) | m | $1.0\times10^{-10}$ | 100.0 | Y |
| Z0 (D) | ROUGHNESS LENGTH (DISPERSION AREA) | Z0 (D) | m | $1.0\times10^{-10}$ | 100.0 | Y |
| ALBEDO (M) | ALBEDO (MET SITE) | R | - | 0.0 | 1.0 | Y |
| ALBEDO (D) | ALBEDO (DISPERSION AREA) | ALBEDO (D) | - | 0.0 | 1.0 | Y |
| ALPHA (M) | MODIFIED PRIESTLEY-TAYLOR PARAMETER (MET SITE) | ALPHA | - | 0.0 | 3.0 | Y |
| ALPHA (D) | MODIFIED PRIESTLEY-TAYLOR PARAMETER (DISPERSION AREA) | ALPHA (D) | - | 0.0 | 3.0 | Y |
| HOUR | HOUR | THOUR | - | 0 | 24 | R |
| DAY | DAY | TDAY | - | 1 | 366 | R |
| YEAR | YEAR | YEAR | - | 1900 | 2500 | R |
| FREQUENCY | FREQUENCY | FR | - | 0 | - | N |
| FREQUENCY FOR MONTHS xx TO xx, HOURS xxxxx TO xxxxx | FREQUENCY FOR MONTHS xx TO xx, HOURS xxxxx TO xxxxx (GMT + xxxxx) | MONTHS xx TO xx, HOURS xx TO xx | - | - | - | N |

**Table 7.1 –** Variables that may be input into the meteorological input module. *Key for interpolation indicator: R = Required for interpolation, Y = interpolatable, N = not interpolatable.

There is no exact correspondence between the boundary layer parameters ($h$, $L_{MO}$) and the Pasquill-Gifford categories since many different values of $h$ and $L_{MO}$ may correspond to one Pasquill-Gifford category.

Stability: Stable $h/L_{MO} > 1$
Neutral $-0.3 \leq h/L_{MO} \leq 1$
Convective $h/L_{MO} < -0.3$

| $U$ (m/s) | $L_{MO}$ (m) | $1/L_{MO}$ ($m^{-1}$) | h (m) | $h/L_{MO}$ | P-G Category |
|---|---|---|---|---|---|
| 1 | -2 | -0.5 | 1300 | -650 | A |
| 2 | -10 | -0.1 | 900 | -90 | B |
| 5 | -100 | -0.01 | 850 | -8.5 | C |
| 5 | ∞ | 0 | 800 | 0 | D |
| 3 | 100 | 0.01 | 400 | 4 | E |
| 2 | 20 | 0.05 | 100 | 5 | F |
| 1 | 5 | 0.2 | 100 | 20 | G |

**Table 7.2** − Values of wind speed, Monin-Obukhov length ($L_{MO}$) and boundary layer height ($h$), which may be used to represent Pasquill-Gifford categories A-G.

The minimum requirement for meteorological data (for the model to run) is:

- wind speed,
- wind direction,

plus *one* of the following:

- reciprocal of Monin-Obukhov length,
- surface sensible heat flux,
- cloud cover, time of day and time of year.

With regard to the latter three options, if more than one of these is supplied, they are used in order of preference in the order they are listed above. That is, if the reciprocal of Monin-Obukhov length is specified it will be used, otherwise the surface sensible heat flux will be used. If neither the reciprocal of Monin-Obukhov length nor the surface sensible heat flux are specified then the cloud cover, time of day and time of year will be used.

If cloud cover, time of day and time of year are the *only* data specified, then it is advisable to add temperature and boundary layer height, if a good estimate is known, to the variables in the *.met* file. If the boundary layer height is not known, the estimate for a particular hour will be improved if meteorological data for all the hours from midnight are included, as the FLOWSTAR-Energy scheme uses this information from previous hours to improve the estimate.

It is preferable to include cloud cover data if available. However, if unavailable, incoming solar radiation may be specified instead. Incoming solar radiation is measured at ground level, as measured by a radiometer, although check with the manufacturer exactly what is measured.

In general, the boundary layer height should always be entered if a better estimate is available than that calculated by the meteorological input module. The module will provide a good estimate when the site is in mid latitudes, e.g. UK, and it is either

1. daytime with sequential data stretching back at least to dawn, or
2. night time.

If however, neither 1 nor 2 apply and boundary layer height information is not available, then it is advisable to provide the model with as much information as possible (e.g. all five of the above meteorological parameters). This may help the model to estimate boundary layer height by enabling it to estimate the values of heat flux occurring prior to the current hour under consideration. In addition, specifying temperature may also help to improve the module's estimate of the boundary layer height.

If no cloud cover, surface sensible heat flux or incoming solar radiation data are available then the effects of stratification cannot be modelled by FLOWSTAR-Energy; in this case the surface sensible heat flux should be set to zero to assume neutral atmospheric conditions.

If the meteorological data are statistical then the frequency with which each meteorological condition occurs is also required.

### *Further input parameters*

In addition to the data in the meteorological input data set, the module also requires certain data that are provided via the interface. These are:

- surface roughness,
- height of recorded wind, i.e. the height above ground at which the wind measurements were made,
- whether the meteorological data are hourly sequential,
- whether the wind direction data are in sectors, and if so, the sector size, the sector treatment to apply (the user’s choice: averaging or cycling) and the number of wind sector increments to use. Wind direction data are often recorded to the nearest 10°, 22.5° or 30°. Inputting the appropriate sector size allows the model to account for the approximation in the wind direction. The wind direction cycling methodology is available for hourly sequential data only and uses one of a user-specified number of equally-spaced wind directions within the sector. The wind direction averaging methodology models a user-specified number of equally-spaced wind directions within the sector and the result is the average of these. If the data are not in sectors, the wind direction is used exactly as given in the meteorological data file.

Further optional parameters may also be specified via the interface. These include a minimum value of $L_{MO}$ (for urban areas) and values of Priestley-Taylor parameter and surface albedo.

If the met. site is distant from the study area, the meteorological input module includes the option to modify the wind profile at the study site by taking account of the surface roughness both at the met. site and the study site. Different values of surface albedo, Priestley-Taylor parameter and minimum value of $L_{MO}$ at the met. site and study site may also be entered.

### *User-input vertical profile data*

The user can input vertical profiles for the wind speed, turbulence parameters, temperature and humidity. These data are entered at a sequence of heights for each line of meteorological data. To obtain the value at a particular height, the model will interpolate a value from those entered, using the algorithm outlined below.

The interpolation algorithm is based on a linear interpolation of the input *.prf* data and the standard FLOWSTAR-Energy profiles; essentially a profile similar in shape to the FLOWSTAR-Energy profile is fitted to the input *.prf* data. The interpolation algorithm is detailed in the Technical Specification (CERC, 2015).

Note that the data entered in the *.prf* file override the corresponding values entered in the *.met* file.

## 7.1.2 Meteorological data input from a *.tab* file

The format of a *.tab* file is such that it includes only information relating to wind speed and wind direction; FLOWSTAR-Energy therefore assumes neutral atmospheric conditions when a *.tab* file is selected.

Data entered in this way are by definition statistical and the wind direction is given in sectors; a user-specified number of wind directions within each sector are modelled and the result at each receptor is the average of these.

### *Further input parameters*

In addition to the data in the *.tab* file, the model also requires surface roughness to be provided via the interface.

Further optional parameters may also be specified via the interface. These include a minimum value of $L_{MO}$ (for urban areas) and values of Priestley-Taylor parameter and surface albedo.

If the met. site is distant from the study area, the meteorological input module includes the option to modify the wind profile at the study site by taking account of the surface roughness both at the met. site and the study site. Different values of surface albedo, Priestley-Taylor parameter and minimum value of $L_{MO}$ at the met. site and study site may also be entered.

## 7.1.3 Meteorological data processing

The meteorological input module is called once for each meteorological condition and uses standard algorithms to calculate the boundary layer meteorological parameters required by the dispersion model. Full details can be found in Holtslag and van Ulden (1983) and the Technical Specification (CERC, 2015).

In addition to processing the data, the module checks that the input data are plausible, i.e. that they lie within certain limits. While it is running, messages may be provided giving warnings and notification of errors. Where an error occurs which prevents calculations being done for that particular line, output for that line will be given as -999.

### 7.1.4 Output

The variables output by the meteorological input module are listed in **Table 7.3**.

| Variable | Description |
|---|---|
| $u_*$ | Friction velocity (m/s) |
| $U_g$ | Geostrophic wind speed (m/s) |
| $U_g^*$ | Geostrophic wind speed normalised by the friction velocity |
| $\phi_0$ | Surface wind direction (angle from which wind blows in degrees measured clockwise from north, e.g. 270° is a westerly wind) (°) |
| $\phi_g$ | Geostrophic wind direction (angle from which wind blows in degrees measured clockwise from north) (°) |
| $\Delta\phi$ | Geostrophic wind direction minus surface wind direction (°) |
| $\phi$ | Wind direction (as obtained from the meteorological input data) (°) |
| $\phi_{sec}$ | Wind direction used by the model for this meteorological data line for long-term calculations, which may differ from the input value if the data are in sectors (°) |
| $w_*$ | Convective velocity scale (if $F_{\theta_0} > 0$, $w_* = (g\, F_{\theta_0}\, h/\rho\, c_p\, T_0)^{1/3}$; if $F_{\theta_0} \le 0$, $w_* = 0$) (m/s) |
| $F_{\theta_0}$ | Surface heat flux (W/m$^2$) |
| $K$ | Incoming solar radiation (W/m$^2$) |
| $1/L_{MO}$ | Reciprocal of the Monin-Obukhov length (m$^{-1}$) |
| $h$ | Boundary layer height (m) |
| $N_u$ | Buoyancy frequency above the boundary layer (s$^{-1}$) |
| $\Delta\theta$ | Temperature jump across the boundary layer top (K) |
| $T_0^c$ | Near-surface temperature (°C) |
| $P$ | Precipitation rate (mm/h) |
| $\Delta T$ | Near-surface temperature over land minus sea surface temperature (°C) |
| $\sigma_\theta$ | Standard deviation of mean wind direction (°) |
| $q_0$ | Surface specific humidity (kg/kg) |
| $\lambda_E$ | Surface latent heat flux (W/m$^2$) |
| $RH_u$ | Relative humidity just above the boundary layer (%) |
| $d(RH_u)/dz$ | Relative humidity lapse rate above the boundary layer (%/m) |

**Table 7.3** − Output variables from the meteorological input module.

The variables $P$, $\Delta T$, $q_0$, $\lambda_E$, $RH_u$ and $d(RH_u)/dz$ may be missing if insufficient input data are given. The processed met. module data are available for use by the other modules, in particular the boundary layer structure module.

### 7.1.5 Treatment of low-wind conditions

FLOWSTAR-Energy treats low wind conditions as follows: for any input meteorological data for which the wind speed at 10 m is zero no boundary layer, flow or wake calculations are carried out and the wind speed is assumed to be zero at all heights; for any lines of meteorological data for which the wind speed at 10 m is less than 0.5 m/s no flow or wake calculations are carried out and the wind speed and turbulence everywhere are the free stream values; for any lines of meteorological data for which the wind speed at 10 m is greater than 0.5 m/s the full boundary layer, flow and wake calculations are carried out.

### 7.1.6 Limitations

In calculating the boundary layer parameters, it is assumed that the boundary layer is self-similar for a given value of $h/L_{MO}$. However, users of the model should be aware that there are some situations, such as latitudes near the equator or the poles, where this approximation can lead to significant errors. Full details are given in the Technical Specification (CERC, 2015).

## 7.2 Parameterisation of the boundary layer

In FLOWSTAR-Energy the boundary layer is characterised by the boundary layer height $h$ and the Monin-Obukhov length $L_{MO}$ and **not** by a Pasquill-Gifford stability category.

The Monin-Obukhov length is defined as

(7.1)
$$L_{MO} = \frac{-u_*^3}{\left(\frac{\kappa g F_{\vartheta 0}}{\rho c_p T_0}\right)}$$

in which $u_*$ is the friction velocity at the Earth's surface, $\kappa$ is the von Karman constant (0.4), $g$ is the acceleration due to gravity, $F_{\theta 0}$ is the surface sensible heat flux, $\rho$ and $c_p$ are, respectively, the density and specific heat capacity of air and $T_0$ is the near-surface temperature.

In unstable or convective conditions, the Monin-Obukhov length is negative. The magnitude of the length is then a measure of the height above which convective turbulence, i.e. turbulent motions caused by thermal convection, is more important than mechanical turbulence, i.e. turbulence generated by friction at the Earth's surface.

In stable conditions, the Monin-Obukhov length is positive. It is then a measure of the height above which vertical turbulent motion is significantly inhibited by the stable stratification.

**Figure 7.1** shows the different regions of the boundary layer in terms of the parameters $h/L_{MO}$ and $z/h$ where $z$ is height above the ground. **Figure 7.2** shows the same information but with a dimensional vertical scale, $z$.

In the different regions of the boundary layer different mechanisms are important in generating turbulence. These are:

1. surface heating leading to convectively generated turbulence (the convective eddies increase in energy as they rise through the boundary layer),
2. shear at the surface leading to mechanically generated turbulence,
3. local shear, for instance at the top of the boundary layer, that can be a weak source of turbulence.

This approach to boundary layer stability, whereby the boundary layer structure is defined in terms of two variables ($z/L_{MO}$ and $z/h$), supersedes the Pasquill-Gifford formulation, and differs crucially from the Pasquill formulation in allowing the variation of boundary layer properties with height to be included. However, it is difficult to make exact comparisons between the two schemes.

**Figures 7.1** and **7.2** show Pasquill-Gifford stability categories corresponding *approximately* to ranges of $h/L_{MO}$. Note that, particularly in stable meteorological conditions, the Pasquill class is not a simple function of $h/L_{MO}$.

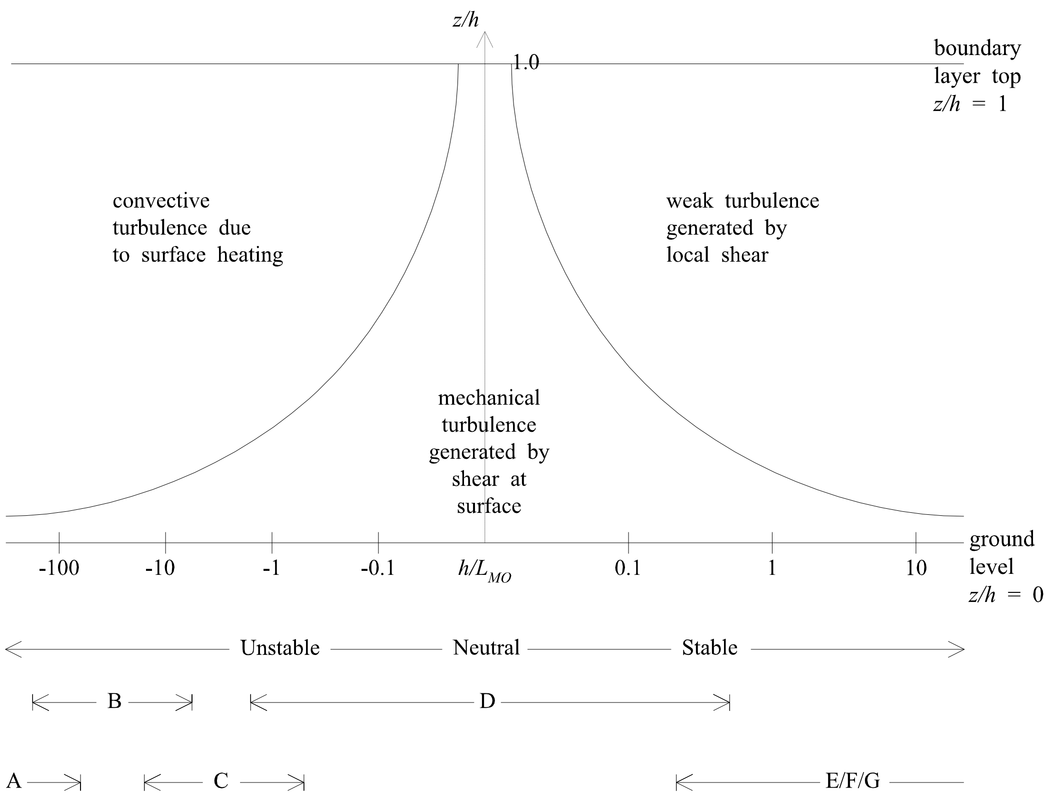



**Figure 7.1** – Non-dimensional schematic representation of variation of Monin-Obukhov length with atmospheric stability.

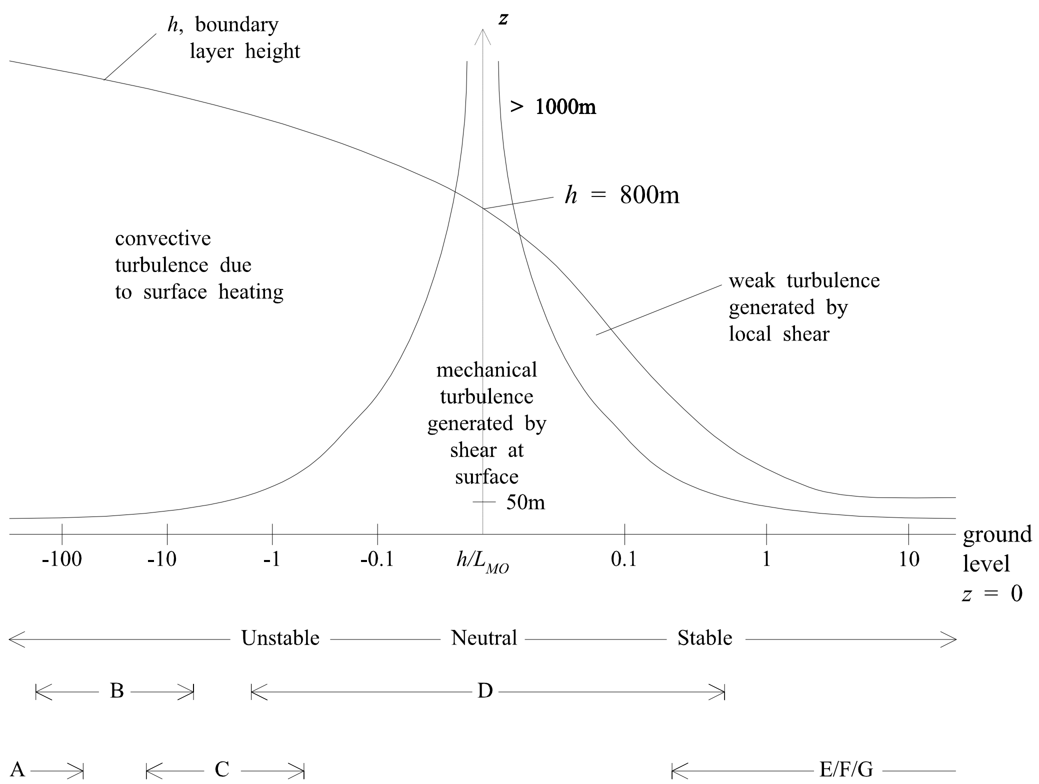



**Figure 7.2** – Dimensional schematic representation of variation of Monin-Obukhov length with atmospheric stability.

### 7.2.1 Boundary layer structure

FLOWSTAR-Energy calculates the boundary layer variables listed in **Table 7.4** at different heights. Vertical profiles are expressed as functions of $z/L_{MO}$ and $z/h$ and have been derived from experimental data (Caughey and Palmer, 1979; van Ulden and Holtslag, 1985; Hunt *et al*., 1988d). These variables are used, in turn, by other modules.

| Variable | Description |
|---|---|
| $U(z), \dfrac{dU}{dz}, \dfrac{d^2U}{dz^2}$ | Mean wind speed (m/s) and its first ($s^{-1}$) and second derivatives ($m^{-1}s^{-1}$) with height |
| $\sigma_u(z), \sigma_v(z), \sigma_w(z)$ | Root-mean-square turbulent velocities (m/s) |
| $\Lambda_v(z), \Lambda_w(z)$ | Turbulent length scales (m) |
| $\varepsilon(z)$ | Energy dissipation rate ($m^2/s^3$) |
| $T_L(z)$ | Lagrangian time scale (s) |
| $N(z)$ | Buoyancy frequency ($s^{-1}$) |
| $T(z)$ | Temperature (K) |
| $\rho(z)$ | Density ($kg/m^3$) |
| $P(z)$ | Pressure (mbar) |

**Table 7.4 –** Boundary layer variables calculated by FLOWSTAR-Energy.

Note that the wake spread parameters $\sigma_y$ and $\sigma_z$ are calculated using these boundary layer variables and hence vary with height. This contrasts with the approach adopted in models using P-G categories, in which values of $\sigma_y$ and $\sigma_z$ are obtained from measurements and are independent of height.

Turbulent velocities $\sigma_u, \sigma_v$ and $\sigma_w$ are subject to a minimum value of turbulence that ranges between 0.01 and 0.2 m/s, depending on the user-input minimum value of $L_{MO}$. This accounts for the fact that in urban areas (high minimum $L_{MO}$), where conditions never become very stable, there will always be some turbulence. The minimum turbulence value to be used can also be specified by the user through the interface.

# 7.3 Marine boundary layer

FLOWSTAR-Energy includes a marine boundary layer scheme for calculating surface roughness and heat fluxes at offshore sites.

## 7.3.1 Wind profile

For surface roughness the formula of Beljaars (1994) has been adopted, used by the European Centre for Medium range Weather Forecasts (ECMWF):

(7.2) $$z_0 = \alpha_m \frac{\nu}{u_*} + \alpha_{Ch} \frac{u_*^2}{g}$$

where $u_*$ (m/s) is the friction velocity, ν ($m^2$/s) is the kinematic viscosity of air, $g$ is the acceleration due to gravity (m/$s^2$), $\alpha_m$ = 0.11 and $\alpha_{Ch}$ is the Charnock parameter, which has a default value of 0.018 but may be entered by the user through the interface.

The velocity profiles used are the same as those used over the land, described in the Technical Specification (CERC, 2015). The equations are solved by iteration.

## 7.3.2 Heat fluxes

Over sea the surface roughnesses for sensible heat ($z_{0H}$) and moisture ($z_{0q}$) are given by Beljaars (1994) as

(7.3) $$z_{0H} = \alpha_H \frac{\nu}{u_*} \quad \text{and} \quad z_{0q} = \alpha_q \frac{\nu}{u_*}$$

where $\alpha_H$ = 0.4 and $\alpha_q$ = 0.62.

Then the sensible heat flux $F_{\theta 0}$ is given by

(7.4) $$F_{\theta_0} = \frac{-c_p \rho \kappa^2 (\theta(z) - \theta_0) u(z)}{\left[ \ln\left( \frac{z + z_{0H}}{z_{0H}} \right) - \psi_H\left( \frac{z + z_{0H}}{L_{MO}} \right) \right] \left[ \ln\left( \frac{z + z_0}{z_0} \right) - \psi\left( \frac{z + z_0}{L_{MO}} \right) \right]}$$

where $c_p$ is the specific heat capacity of air (J/kg/K), $\rho$ is the density of air (kg/$m^3$), $\kappa$ is von Karman's constant (0.4), $\theta$ is potential temperature (K), $\theta_0$ is the potential temperature corresponding to the temperature of the sea surface, and $L_{MO}$ is the Monin-Obukhov length (Panofsky and Dutton, 1984).

For stable and neutral conditions ($1/L_{MO} \geq 0$), $\psi_H = \psi$ (as defined in the Technical Specification (CERC, 2015)), and for convective conditions ($1/L_{MO} < 0$) $\psi_H$ is given by

(7.5)
$$\psi_H = \ln\left[\frac{(1+y)^2}{(1+y_{surface})^2}\right]$$

where

(7.6)
$$y = \sqrt{1-16\left(\frac{z+z_{0_H}}{L_{MO}}\right)} \quad \text{and} \quad y_{surface} = \sqrt{1-16\left(\frac{z_{0_H}}{L_{MO}}\right)}$$

The latent heat flux $\lambda E$ is given similarly by

(7.7)
$$\lambda E = \frac{-\lambda\rho\kappa^2(q(z)-q_{sat0})u(z)}{\left[\ln\left(\frac{z+z_{0q}}{z_{0q}}\right)-\psi_q\left(\frac{z+z_{0q}}{L_{MO}}\right)\right]\left[\ln\left(\frac{z+z_0}{z_0}\right)-\psi\left(\frac{z+z_0}{L_{MO}}\right)\right]}$$

where $\lambda$ is the specific latent heat of vaporization of water (J/kg), $q$ is the specific humidity, $q_{sat0}$ is the saturation specific humidity at the sea surface, and $\psi_q = \psi_H$. Again the equations are solved by iteration (Panofsky and Dutton, 1984).

### 7.3.3 Input data requirements

Some additional data are required to model offshore sites, all of which are important in determining the structure of the marine boundary layer:

- wind speed and direction,
- temperature (°C),
- sea surface temperature (°C),

***Users are strongly recommended to include temperature data in offshore site runs.*** *However, if no temperature data are available, then users can select an option to assume that the marine boundary layer is neutral, in which case no heat fluxes are calculated.*

- Charnock parameter,

  This is a constant used in the heat flux calculations. Typical values range from 0.018 to 0.08, but in the marine boundary layer this may depend on, for example, distance from the coast. Validation suggests that a value of 0.08 is most appropriate in FLOWSTAR-Energy for regions not altogether remote from land (e.g. North Sea).

- height above sea level of temperature measurements (in metres) (if temperature data are included in run),
- height above sea level of humidity measurements (in metres) (if humidity data are included in run).

# 7.4 Complex terrain module

The complex terrain module calculates a three-dimensional flow and turbulence field for onshore sites when hills and/or variable surface roughness are modelled.

When variable terrain height is modelled there are two possible flow field solutions. The flow field solution used depends on the meteorological conditions. The critical parameter is the maximum height of the dividing surface, $H_c$; refer to Section 7.4.2 for more details. The critical dividing surface height is 20 m.

**Table 7.5** summarises the dependence of the model behaviour on the maximum dividing surface height. Note that when modelling variable terrain height for very stable flows, the stable flow solution does not account for any spatial variation of surface roughness.

| Complex terrain modelling option | $H_c$>20m | $H_c$≤20m |
|---|---|---|
| Variable terrain height, with or without variable surface roughness | Very stable flow solution (assuming a constant roughness value) | FLOWSTAR-D solution |
| Variable surface roughness only | FLOWSTAR-D solution | FLOWSTAR-D solution |

**Table 7.5** – Summary of model behaviour when running the complex terrain option.

### 7.4.1 FLOWSTAR-D solution

In this case the flow field and turbulence values used in the dispersion modelling calculations are those output from the FLOWSTAR-D (F*) model. FLOWSTAR models air flow and dispersion over hills with roughness changes, including the effects of stratification. The model is described in some detail in Carruthers *et al*. (1988) while the basis of the theory is presented in Hunt *et al*. (1988b, 1988c) and Hunt (1985).

The airflow model uses linearised analytical solutions of the momentum and continuity equations, and includes the effects of stratification on the flow. If modelled, ideally hills should have moderate slopes (say less than 1 in 3) but the model is useful even when this criterion is not met. The terrain height is specified at up to 66,000 points which are interpolated by the model onto a regular grid of up to 256×256 points. The best results are achieved if the points at which the elevation (and surface roughness, if it varies) is specified are regularly spaced. The effects of roughness and terrain can be modelled separately. The airflow model has been extensively tested with laboratory and field data.

### 7.4.2 Very stable flow conditions (variable terrain height, Hc>20m)

When very stable flows approach an isolated hill, the flow may divide, with the air above a certain height $h_c$ (the *dividing surface*) flowing over the hill in a terrain-following manner and the air below the dividing surface flowing around the hill in a 2-dimensional flow. This is shown in **Figure 7.3**.

The maximum height of the dividing surface, $H_c$, is defined by an energy balance equation, which locates the lowest height at which the kinetic energy of an air parcel in the flow approaching the hill is equal to the potential energy attained by elevating an equivalent fluid parcel from this height to the top of the hill:

$$\frac{1}{2}U^2(H_c) = \int_{h_c}^{h_{\max}} N^2(z) \times (h_{max} - z)\, dz \,. \qquad (7.8)$$

The flow below the dividing surface is two-dimensional potential flow apart from upstream flow in a thin layer close to the terrain. Above the dividing surface, the FLOWSTAR-D solution is used except close to the ground where a weighted average of the FLOWSTAR-D and terrain following flow fields is used.

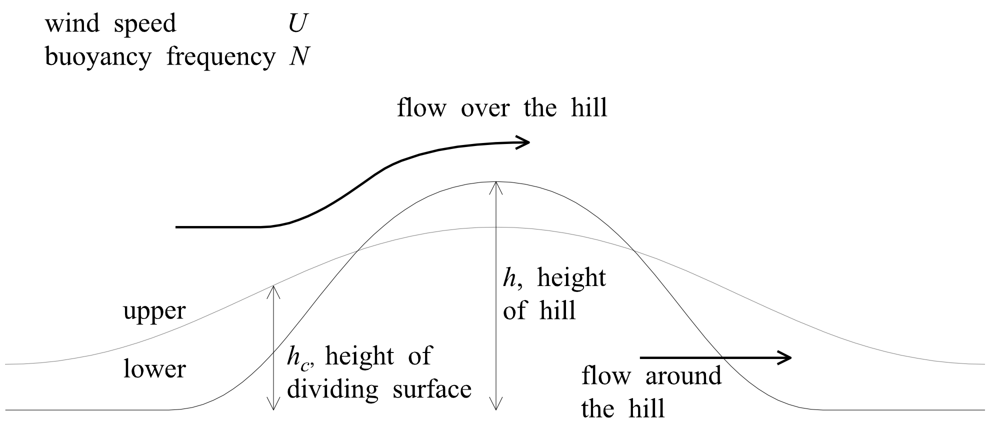



**Figure 7.3** – Effect of a hill on the flow in very stable conditions. $H_c$ is the maximum height of the dividing surface.

### 7.4.3 Running the complex terrain module

During a model run the following calculations take place:

- interpolation of terrain and roughness data (if variable) onto the local grid,
- calculation of mean air flow,
- calculation of shear stress and turbulence,
- calculation of mean streamlines through wind turbine (if included),
- calculation of wind turbine wake dispersion (if included).

It is not recommended that the complex terrain module be used unless hill slopes are greater than about 1 in 10 or there are large surface roughness changes.

The model can be used for very large changes in surface roughness. An order of magnitude variation in the surface roughness is allowable with the approximations made in the calculation.

# 7.5 Wind farm module

The wind farm module calculates the effect of one or more wind turbines on local wind speed and turbulence. The scientific methodology behind the model is described in this section. In particular, the FLOWSTAR-Energy model of a wind turbine wake is discussed. The wake is a region downstream of the wind turbine where the flow field is affected by the wind turbine.

If wind turbine effects are being modelled then a calculation grid is created with a spatial extent that covers all output points and wind turbine locations. This domain is then extended by 10% in each direction, so that the flow field can properly be resolved over the full modelling region. The resolution of the calculation grid is set by the user through the interface.

An overview of the FLOWSTAR-Energy model of a wind turbine wake is as follows. The wind speed downstream of a wind turbine in operation is reduced owing to the action of the wind turbine on the flow field and so a wake exists downwind of the wind turbine. Similarities between the decay of the wake behind a wind turbine and the dispersion of a plume of passive gas emitted from an elevated source are exploited in the model of wind turbine wakes. For example, a plume is characterised by an amount of gas emitted continuously, which is advected downwind whilst mixing with the ambient air, at a rate determined by the amount of turbulence in the atmosphere. A wind turbine wake is characterised by a wind speed deficit produced by the wind turbine, which is also advected downwind and gradually mixes with the ambient air, in a similar manner to a plume, but with some special characteristics.

The basic method is that the fully-expanded wind turbine wake immediately downwind of a wind turbine is characterised as a thin volume source, with source strength and dimensions calculated to give a maximum wind speed deficit determined by conservation of momentum according to the properties of the wind turbine. The wind speed deficit is the quantity which is then dispersed downwind.

The dispersion of the wind speed deficit, or wake, is modelled using CERC's ADMS dispersion model used for pollutants, with some modifications to account for the special dispersion characteristics of wind turbine wakes. These modifications include the additional shear-induced turbulence in the wake and the lack of large-scale wind direction meandering in wind farm environments.

Wind farms are modelled by considering the wake effects from individual wind turbines in downstream order, so that wakes from upstream wind turbines affect the flow field used when characterising the downstream wind turbine effective sources and when dispersing their wakes. The wake model has been validated against measured data from Tjæreborg Enge, Nysted wind farm and Noordzee wind farm.

Section 7.5.1 describes how each wind turbine is characterised as an effective volume source; Section 7.5.2 describes the effective volume source dispersion; Section 7.5.3 describes the treatment of the additional shear-induced turbulence in a wind turbine wake; and Section 7.5.4 describes the FLOWSTAR-Energy treatment of wind turbine wake interaction.

### 7.5.1 Representing a wind turbine as an effective volume source

As described above, the fully expanded wind turbine wake immediately downstream of the wind turbine is characterised as an effective volume source, with a source strength that gives a maximum wind speed deficit determined by conservation of momentum. The wake effects are modelled by calculating the dispersion of this initial wind speed deficit downstream of the wind turbine.

Classical 1-D momentum theory for a wind turbine states that the rate of change of momentum across a wind turbine rotor disc (i.e. the thrust) is caused solely by the pressure gradients across the disc, and that since the pressure recovers at some distance downstream, the momentum in the 'stream-tube' upstream of this pressure gradient is equal to the momentum in the stream-tube downstream of the pressure gradient, i.e. the stream-tube expands as the flow speed reduces.

The quantity used to describe the strength of the wind speed deficit in the wake relative to the upstream flow is called the axial induction factor $a$, which is defined as follows:

(7.9) $$U_d = (1-a)U$$

where $U_d$ is the axial flow speed through the rotor disc (m/s) and $U$ is the upstream wind speed (m/s).

The 1-D momentum theory together with (7.9) leads to a relationship between $U$ and the axial flow speed in the fully expanded wake $U_w$, as given by Hansen (2008):

(7.10) $$U_w = (1-2a)U$$

which yields a maximum wind speed deficit in the wake $\Delta U_{\max}$ of $2aU$. This is the initial value of the wind speed deficit immediately downwind of the wind turbine.

The wake is modelled in FLOWSTAR-Energy as an effective volume source with square cross-section representing the fully-expanded wake where the axial flow speed is $U_w$. The cross-sectional area of the source is equal to the cross-sectional area of the expanded wake. This type of source has an initial 'top-hat' vertical and horizontal profile, which decays to a Gaussian-shape profile as it evolves downstream. **Figure 7.4** shows a schematic of the expanding stream-tube and the effective volume source.

The justification for using a square cross-section volume source rather than a circular one is mainly pragmatic; a square cross-section volume source is simpler to model than a circular volume source and a circular volume source is unlikely to yield much benefit due to the evolution of the profile to a Gaussian shape within a few wind turbine diameters.

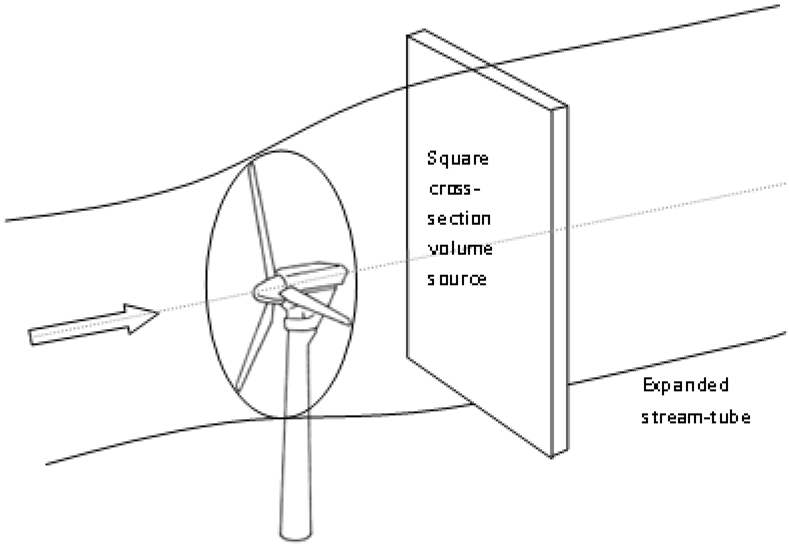



**Figure 7.4** – Schematic of the expanding stream-tube of air flowing through the wind turbine rotor, and the effective volume source representing that wind turbine.

To calculate the dimensions of the volume source we use the classical 1-D momentum theory to find:

(7.11) $$dy = dz = \frac{D}{2}\sqrt{\frac{\pi(1-a)}{(1-2a)}}$$

where $dy$ is the crosswind extent of the volume source, $dz$ is the depth of the volume source, and $D$ is the diameter of the rotor. As most modern wind turbines have efficient mechanisms to bring the wind turbine rotor disc perpendicular to the inflow wind as quickly as possible the model makes an assumption of zero yaw, i.e. it assumes that the wind turbine rotor disc is always perpendicular to the upstream wind and does not take account of any rotations. If the wind direction changes between meteorological conditions then the model assumes that there is an instantaneous re-orientation of the wind turbine. The volume source is therefore always perpendicular to the upstream flow. The along-wind extent of the volume source is set to ten percent of the crosswind extent *dy*.

The wind turbine effective source strength is calculated inversely from the known maximum wind speed deficit $\Delta U_{\max} = 2aU_{\infty}$ where $U_{\infty}$ is the upstream wind speed. In model terms, with concentration as a surrogate for ΔU, $\Delta U_{\max}$ can be expressed as

(7.12) $$\Delta U_{\max} = \frac{QV_{src}}{\dot{V}}$$

$Q$ is the source strength (in units of $ms^{-2}$), $V_{src}$ is the source volume, and $\dot{V}$ is the volume flow rate through the source. Therefore

(7.13) $$Q = 2aU^2$$

These parameters are all functions of the axial induction factor $a$, which must in turn be calculated from information about the wind turbine itself.

***Calculation of the Axial Induction Factor***

FLOWSTAR-Energy calculates the axial induction factor from the thrust coefficient $C_T$ (input to the model as a function of wind speed using the *.wtd* file described in Section 3.4).

The relationship between thrust coefficient $C_T$ and axial induction factor $a$ is, following Hansen (2008),

(7.14)
$$C_T = \begin{cases} 4a(1-a)F, & a \le a_c \\ 4\left(a_c^{\,2} + (1-2a_c)a\right)F, & a > a_c \end{cases}$$

where $a_c \approx 0.2$.

This becomes an equation for $a$ in terms of $C_T$:

(7.15)
$$a = \begin{cases} \frac{1}{2}\left(1-\sqrt{1-C_T}\right) & C_T \le 0.64 \\ \frac{C_T - 4a_c^{\,2}}{4(1-2a_c} & C_T > 0.64 \end{cases}$$

## 7.5.2 Effective volume source dispersion

In FLOWSTAR-Energy, the effective volume source representing the initial wind speed deficit in the wake is decomposed into a maximum of 10 source elements, the decomposition being a function of the receptor location. The difference in streamwise distance between source elements and each receptor is constrained not to vary too rapidly, subject to the maximum number of 10 source elements. The wake deficit is then calculated by summing the contributions from each element.

The contribution from each element is approximated by a crosswind vertical slice of finite length and height. The expression for the wake deficit $\overline{\overline{C}}(x,y,z)$ from a crosswind vertical slice of length $L_s$ and height $L_1$ is given by equation (7.16). The source strength $\overline{\overline{Q}}_s$ is in units of $\mathrm{ms}^{-2}$.

(7.16)
$$\overline{\overline{C}}(x,y,z) = \frac{\overline{\overline{Q}}_s}{4U}\left[\mathrm{erf}\left(\frac{y+L_s/2}{\sqrt{2}\sigma_y}\right) - \mathrm{erf}\left(\frac{y-L_s/2}{\sqrt{2}\sigma_y}\right)\right] \times \left[\mathrm{erf}\left(\frac{z+L_1/2-z_s}{\sqrt{2}\sigma_z}\right) - \mathrm{erf}\left(\frac{z-L_1/2-z_s}{\sqrt{2}\sigma_z}\right)\right] + \text{ reflection terms}$$

### *Dispersion parameters*

Field experiments and research have shown that the dispersion parameters $\sigma_y$ and $\sigma_z$ vary with downwind distance from a source of airborne emitted material in a way that depends on the atmospheric boundary layer height ($h$), the height of the source ($z_s$) and the height of the plume as it travels downwind. For reviews of this subject, see Hunt *et al*. (1988a), Hanna and Paine (1989) and Weil (1985). This approach is in contrast to older methods, described in the NRPB report R91 (Clarke, 1979) and used in the model ISC (U.S. E.P.A., 1995), in which the effect of the source height is not taken into account when calculating the width and depth of the plume.

There is no general theory or even generally accepted semi-empirical expression that describes the dispersion from a source at all heights within the boundary layer ($0 < z_s < h$), in all conditions of atmospheric stability, and over the complete range of distances from the source to about 30 km downwind. In developing FLOWSTAR-Energy the approach adopted has been first to use formulae that have been developed and broadly accepted for specific ranges of the parameters $z_s/h$, $h/L_{MO}$ (stability) and $x/h$ (downwind distance). Interpolation formulae have then been constructed to cover the complete parameter range. The basis for these formulae is set out at length in the report by Hunt *et al*. (1988a).

### *Stable and neutral boundary layers*

All the turbulence in the stable boundary layer is mechanically generated, i.e. there is no generation of turbulence due to convective motions. Usually, the level of turbulence decreases with height, as the relative importance of stratification increases, although it can be enhanced by wave motions at the top of the boundary layer (the effect of such wave motions is not considered by FLOWSTAR-Energy).

The vertical dispersion parameter ($\sigma_z$) at the mean height of the wake ($z_m$) is linked directly to the vertical component of turbulence ($\sigma_w$) and the travel time from the wind turbine ($t$) by the relationship

(7.17)
$$\sigma_z = \sigma_w t \left( \frac{1}{b^2} + \frac{N^2 t^2}{1 + 2Nt} \right)^{-1/2}$$

where $N$ and $\sigma_w$ are the buoyancy frequency and root-mean-square vertical turbulent velocity at $z_m$, respectively (Weil, 1985; Hunt, 1985). The factor $b$ is a function of $z_s/h$ and ensures a smooth transition between the solutions for wind turbines of different heights.

The transverse dispersion parameter ($\sigma_y$) is given by

(7.18)
$$\sigma_y = \sigma_v \, t \left( 1 + \sqrt[3]{15.6} \, \frac{u_* t}{h} \right)^{-1/2}$$

with $u_*$ the friction velocity at the Earth's surface and $\sigma_v$ the root-mean-square crosswind turbulent velocity.

In neutral conditions the wake is effectively confined within the boundary layer, because turbulence at the top of the layer is inhibited. Sufficiently far from the wind turbine, after parts of the wake have been reflected at the ground and at the top of the boundary layer, the vertical variation in wind speed deficit is so small as to be negligible. This occurs approximately at the downwind distance where $\sigma_z \approx h$ (in fact, where $\sigma_z = 1.5\ h$). Downwind of this point the wake is considered to grow horizontally as a vertical wedge as if from a uniform vertical line source extending from 0 to $h$, rather than as a cone, so the variation with $z$ in equation (7.16) is ignored.

### *Convective boundary layer*

Field experiments of dispersion from elevated sources in the convective boundary layer (Briggs, 1985) have confirmed earlier laboratory and computational studies (Lamb, 1982) that the form of the vertical profiles of concentration are skewed and significantly non-Gaussian, thus changing the profile of concentration compared with an assumed Gaussian distribution. Ignoring the non-Gaussian distribution could lead to under-estimates of the effect of the wind turbine wake near the ground. Dispersion models like FLOWSTAR-Energy that use non-Gaussian profiles are referred to as "new generation" models. New generation models include the Hybrid Plume Diffusion Model (HPDM) of Hanna and Paine (1989), the Almanac code of National Power (Moore and Lee, 1982) and the more recent CTDM code of the U.S. E.P.A. (Perry, 1991) and AERMOD (Cimorelli *et al*., 2004).

The transverse dispersion parameter ($\sigma_y$) is calculated in two parts, the first for dispersion due to convection ($\sigma_{yc}$) and the second due to mechanically driven turbulence ($\sigma_{yn}$):

(7.19) $$\sigma_{yc} = \sigma_{vc}\, t\left(1 + \frac{t}{h}\sqrt[3]{0.75 w_*}\right)^{-1/2}$$

(7.20) $$\sigma_{yn} = \sigma_{vn}\, t\left(1 + \frac{t}{h}\sqrt[3]{15.6 u_*}\right)^{-1/2}$$

$\sigma_{vc}$ and $\sigma_{vn}$ are the root-mean-square horizontal turbulent velocities due to convection and mechanically driven turbulence, respectively.

The total transverse spread is therefore given by

(7.21) $$\sigma_y^2 = \sigma_{yc}^2 + \sigma_{yn}^2$$

For the vertical spread $\sigma_z$, the skewed nature of the probability distribution function for the vertical turbulent velocity $\sigma_w$ leads to the definition of $\sigma_{w+}$ and $\sigma_{w-}$ for the upwards and downwards velocities, respectively. Then $\sigma_{z+} = \sigma_{w+}\ t$ and $\sigma_{z-} = \sigma_{w-}\ t$ are defined and used in the calculation of $C_{CBL}$. A more detailed description is given in the Technical Specification (CERC, 2015).

The formulae above (equations (7.19) and (7.20)) reduce to forms *suitable* for neutral conditions as the meteorological conditions tend towards neutral conditions, but the actual forms are not the same as the formulae for the neutral boundary layer. A smooth transition from the convective boundary layer solution to the neutral boundary layer solution is used.

The transition to a far-field model, where the effective volume source representing the initial wind speed deficit in the wind turbine wake is equivalent to a vertical line source extending from 0 to *h* and of uniform strength, is assumed to occur when $\sigma_z = 1.5\,h$.

### 7.5.3 Shear-induced turbulence

At the edge of a wind turbine wake, there is a gradient in the speed of the air flow, which generates extra turbulence. An extra component of turbulence, $\sigma_{shear}$, has been added to account for this; $\sigma_{shear}$ is calculated as

(7.22)
$$\sigma_{shear}[i+1] = \begin{cases} 0.4|\Delta U| \dfrac{x}{X_{crit}}, & x \le X_{crit} \\ \sigma_{shear}[i] \times e^{-dt/t} & x > X_{crit} \end{cases}$$

based on Bevilaqua and Lykoudis (1978) where $D$ is the diameter of the wind turbine, $x$ is the downwind distance from the (effective) source and $|\Delta U|$ is the local wind speed deficit.

Here, *i* and *i*+1 represent consecutive points in a downstream direction along the wake centreline; *dt* represents the time taken by the wake centreline to travel from the *i*th to the (*i*+1)th point. *t* represents the time taken by the plume to spread the width (depth) of the wake:

(7.23)
$$t = \frac{2R}{\sigma_{tot}}$$

*R* is the wake radius, defined as the effective volume source half-width plus the local plume spread, calculated at a given point as a weighted average over all upstream wakes; $\sigma_{tot}$ is the combination of the upstream turbulent velocity and the local shear-induced turbulence:

(7.24)
$$\sigma_{tot}[i+1] = \sqrt{\sigma_{shear}[i]^2 + \sigma[i+1]^2}$$

$X_{crit}$ is a critical distance, which is dependent on the inflow turbulence and is the distance downstream from the effective source at which $\sigma_{shear}$ starts to decay. For inflows with low turbulence, the shear-induced turbulence increases downstream from zero at the effective source to $X_{crit}$, in proportion with the local wind speed deficit, and then decays; for turbulent inflows, the initial value of $\sigma_{shear}$ is $0.4|\Delta U|$ at the wind turbine and it decays immediately. $X_{crit}$ is defined as

(7.25)
$$X_{crit} = \begin{cases} 4D & TI \leq TI_{lower} \\ 4D\left[1 - \dfrac{(TI - TI_{lower})}{(TI_{upper} - TI_{lower})}\right] & TI_{lower} < TI < TI_{upper} \\ 0 & TI \geq TI_{upper} \end{cases}$$

The inflow turbulence is characterised by the turbulence intensity $TI$, which is expressed as a percentage and represents the ratio of the horizontal turbulence to the horizontal mean flow. $TI$ is defined as

(7.26)
$$TI = 100 \times \frac{\sqrt{{\sigma_u}^2 + {\sigma_v}^2}}{\sqrt{u^2 + v^2}}$$

$TI_{lower}$ and $TI_{upper}$ are threshold values determined during validation of the model; these are set to 12% and 18% respectively.

### 7.5.4 Treatment of wind turbine interaction

FLOWSTAR-Energy accounts for the effect of the change in the wind field due to upstream wind turbines on downstream wind turbines. This section describes these changes.

#### *Ordering of wind turbines*

Before the wake calculations are carried out for each input meteorological condition all the input wind turbines are re-ordered according to their downwind position; the most upstream wind turbine is modelled first, the most downstream wind turbine is modelled last. The assumption is made that wind turbines are only affected by the wakes from upstream wind turbines.

#### *Modification to flow field*

The flow field is modified by each wind turbine wake and includes the effect of the wakes from all upstream wind turbines. This is made possible by the ordering of wind turbines described above.

The flow field is perturbed by each wind turbine by adjusting the values of the velocity components from the current flow field to include the wind speed deficit. This is done over a grid of points covering the modelling region and the wind turbine locations at a range of heights within the atmospheric boundary layer. This process is repeated iteratively covering the effect of each wind turbine in downstream order. This change in the flow field for each wind turbine in a wind farm affects the characterisation of the effective volume source and the dispersion of the wake.

## 7.6 Long term averages

This section describes the calculation of the wind and turbulence fields averaged over all the input met conditions.

Let us consider the long term wind field at a particular point. If we take ($u_i$,$v_i$,$w_i$) to be the wind vector at a particular point for met. condition *i*, with $u_i$, $v_i$ and $w_i$ being the wind components in the x-, y- and z- directions respectively, then

(7.27) $$(U_{LT}, V_{LT}, W_{LT}) = \frac{1}{N}\sum_{i=1}^{N} f_i (u_i, v_i, w_i)$$

is the mean wind vector, with $f_i$ being the frequency of occurrence of met. condition *i* and *N* being the number of hours represented by the valid met. lines. (Note that if the met. data are not statistical, all the frequencies $f_i$ are equal to 1.)

The mean horizontal wind speed at a point is given by

(7.28) $$\frac{1}{N}\sum_{i=1}^{N} f_i \sqrt{u_i^2 + v_i^2}$$

In addition, the magnitude of the mean horizontal wind vector is calculated. This is given by

(7.29) $$\sqrt{\left(U_{LT}^2 + V_{LT}^2\right)}$$

The mean turbulence field is defined in a similar way to the mean flow field. That is if we take ($\sigma_{ui}$,$\sigma_{vi}$,$\sigma_{wi}$) to be the turbulence components in the x-, y- and z- directions respectively at a particular point for hour *i*, then the mean turbulence field, $(\Sigma_{uLT}, \Sigma_{vLT}, \Sigma_{wLT})$, is defined by

(7.30) $$(\Sigma_{uLT}, \Sigma_{vLT}, \Sigma_{wLT}) = \frac{1}{N}\sum_{i=1}^{N} f_i (\sigma_{ui}, \sigma_{vi}, \sigma_{wi})$$

# APPENDIX A References

Documents or presentations containing results of CERC model validation can be found on the CERC website cerc.co.uk.

FLOWSTAR-Energy is based upon several of the ADMS 5 modules. The full ADMS 5 Technical Specification is available from the CERC website at the address cerc.co.uk/environmental-software/technical-specifications.html.

***Published technical papers***

The following publications provide technical information on the ADMS and FLOWSTAR-Energy models including their validation and application.


Carruthers, D.J., Edmunds, H.A., Lester, A.E., McHugh, C.A. and Singles, R.J., 2000: Use and validation of ADMS-Urban in contrasting urban and industrial locations. In *Int. J. Environment and Pollution*, **14**, pp. 364-374.

Carruthers, D.J. and Choularton, T.W. 1982: Airflow over hills of moderate slope. In *Quart. J. R. Met. Soc.* **108**, 603-624.

Carruthers, D.J., Dixon, P., McHugh, C.A., Nixon, S.G. and Oates, W., 2001: Determination of Compliance with UK and EU Air Quality Objectives From High-Resolution Pollutant Concentration Maps Calculated Using ADMS-Urban. In *Int. J. of Environment and Pollution,* **16**, pp. 460-471.

Carruthers, D.J., Dyster, S. and McHugh, C.A., 2000: Contrasting methods for validating ADMS using the Indianapolis dataset. In *Int. J. Environment and Pollution*, **14**, pp. 115-121.

Carruthers, D.J., Dyster, S. and McHugh, C.A., 2003: Factors affecting interannual variability of $NO_X$ and $NO_2$ concentrations from single point sources. In *Clean Air and Environmental Protection*, **33**, 1, pp. 15-20.

Carruthers, D.J., Edmunds, H.A., Bennett, M., Woods, P.T., Milton, M.J.T., Robinson, R., Underwood, B.Y. and Franklyn, C.J., 1995: Validation of the UK-ADMS Dispersion Model and Assessment of its Performance Relative to R91 and ISC using Archived LIDAR Data. Study commissioned by Her Majesty's Inspectorate of Pollution, published by DoE. DoE/HMP/RR/95/022.

Carruthers, D.J., Holroyd, R.J., Hunt, J.C.R., Weng, W-S., Robins, A.G., Apsley, D.D., Thomson, D.J. and Smith, F.B., 1994: UK-ADMS: a new approach to modelling dispersion in the Earth's atmospheric boundary layer. In *Journal of Wind Engineering and Industrial Aerodynamics,* **52**, pp. 139-153.

Carruthers, D.J., Holroyd, R.J., Hunt, J.C.R., Weng, W-S., Robins, A.G., Apsley, D.D., Smith, F.B., Thomson, D.J. and Hudson, B., 1991: UK Atmospheric Dispersion Modelling System. In *Proceedings of the 19th NATO/CCMS International Technical Meeting on Air Pollution Modelling and its Application, September 1991, Crete, Greece*. Eds. Han van Dop and George Kallos, Plenum Publishing Corporation, New York.

Carruthers, D.J., McKeown, A.M. and McHugh, C.A., 1997: Air dispersion study of options for abatement of $SO_2$ emissions at Castle Cement, Clitheroe. In *Environment Agency*

*Report NW-9/97-25/B-BADQ*.

Carruthers, D.J., McKeown, A.M., Ellis, K.L. and McHugh, C.A., 1997: Air dispersion climate from the Castle Cement Works at Ribblesdale. In *Environment Agency Report NW-7/97-20-BAZSQ*.

Carruthers, D.J., McKeown, A.M., Hall, D.J. and Porter, S., 1999: Validation of ADMS against wind tunnel data of dispersion from chemical warehouse fires. In *Atmospheric Environment*, **33**, pp. 1937-1953.

CERC, 1998: Turbulence, Fluctuations and Averaging Times. CERC Technical Note, 22 December 1998.

CERC, 2015: ADMS 5 Technical Specification.

DETR, 2000: The Air Quality Strategy for England, Scotland, Wales and Northern Ireland. *Cm 4548, SE2000/3, NIA7*. Published by DETR in partnership with the Scottish Executive, the National Assembly for Wales and the Department of the Environment for Northern Ireland.

Dyster, S.J., Thomson, D.J., McHugh, C.A. and Carruthers, D.J., 2001: Turbulent Fluctuations and Their Use in Estimating Compliance Standards and in Model Evaluation. In *Int. J. of Environment and Pollution*, **16**, pp. 57-68.

Fackerell, J.E. and Robins, A.G., 1982: Concentration fluctuations and fluxes in plumes from point sources in a turbulent boundary layer. In *J. Fluid Mech.*, **117**, pp. 1-26.

Hunt, J.C.R., Holroyd, R.J., Carruthers, D.J., Robins, A.G., Apsley, D.D., Smith, F.B. and Thomson, D.J., 1990: Developments in Modelling Air Pollution for Regulatory Purposes. In *Proceedings of the 18th NATO/CCMS International Technical Meeting on Air Pollution Modelling and its Application*, Vancouver, Canada.

Irwin, J.S., Carruthers, D.J., Stocker, J., and Paumier, J., 2003: Application of ASTM D6589 to evaluate dispersion model performance. In *Int. J. Environment and Pollution*, **20**, pp. 4-10.

Jackson P.S. and Hunt J.C.R., 1975: Turbulent flow over a low hill. In *Quart. J. R. Met. Soc.* **101**, 929-955.

McHugh, C.A., Carruthers, D.J., Higson, H. and Dyster, S.J., 1999: Comparison of Model Evaluation Methodologies with Application to ADMS 3 and US Models. Paper 58 in *Proceedings of the 6$^{th}$ International Conference on Harmonisation within Atmospheric Dispersion Modelling for Regulatory Purposes, Rouen, 11-14 October 1999*.

Pasquill, F. and Smith, F.B., 1983: Atmospheric Diffusion - Study of the dispersion of windborne material from industrial and other sources. Third edition. Ellis Horwood Ltd, Chichester.

Robins, A. and McHugh, C.A., 2001: Development and Evaluation of the ADMS Buildings Effects Module. In *Int. J. of Environment and Pollution*, **16**, pp. 161-174.

Thomson, D.J., 1990: A stochastic model for the motion of particle pairs in isotropic high-Reynolds number turbulence and its application to the problem of concentration variance. In *J. Fluid Mech.*, **210**, pp. 113-153.

Thomson, D.J., 1997: The 'inertial-meander' sub-range of a dispersing plume. In *J. Applied Meteorology*, **36,** pp. 1046-1049.

Timmis, R., Wilkinson, S., Carruthers, D.J. and McHugh, C.A., 2000: Recent Studies to

Validate and Compare Atmospheric Dispersion Models for Regulatory Purposes in the UK. In *Int. J. of Environment and Pollution*, **14**, pp. 431-442.

### *Others*

The following publications are references to papers and reports on dispersion relevant to the technical content of ADMS 5.

Azzi, M., Johnson, G. and Cope, M.E., 1992: An Introduction to the Generic Reaction Set Photochemical Smog Mechanism. In *Proceedings of the 11th International Conference of the Clean Air Society of Australia and New Zealand (Brisbane, Australia, July 1992*), **2**, pp. 451-462. Eds. P. Best, N. Bofinger, D. Cliff.

Belcher S.E, Coceal, O., Hunt, J.C.R., Carruthers, D.J., and Robins, A.G., 2013: A review of urban dispersion modelling. ADMLC report, available at http://webarchive.nationalarchives.gov.uk/20131103234051/http://www.admlc.org.uk/ADMLCReport7.htm (accessed May 2015)

Beljaars, A. C. M., 1994: The parameterisation of surface fluxes in large-scale models under free convection. In *Quart. J. R. Met. Soc.*, **121**, pp. 255-270.

Bevilaqua, P.M. and Lykoudis, P.S., 1978: Turbulence memory in self-preserving wakes. In *J. Fluid Mech.*, **89**, pp. 589-606.

Briggs, G.A., 1969: *Plume Rise*. U.S. Atomic Energy Commission Div. Tech. Inf.

Briggs, G.A., 1985: Analytical parameterizations of diffusion - the convective boundary layer. In *J. Clim. Appl. Met.*, **14**, pp. 1167-1186.

Carruthers, D.J., Hunt, J.C.R. and Weng, W.-S., 1988: A computational model of stratified turbulent airflow over hills - FLOWSTAR I. In *Computer Techniques in Environmental Studies*, pp. 481-492. Ed. P. Zanetti, Springer-Verlag.

Caughey, S.J. and Palmer, S.G., 1979: Some aspects of turbulence structure through the depth of the convective boundary layer. In *Quart. J. R. Met. Soc.*, **105**, pp. 811-827.

Cimorelli, A.J, Perry, S.G., Venkatram, A., Weil, J.C., Paine, R.J., Wilson, R.B., Lee, R.F., Peters, W.D., Brode, R.W. and Paumier, J.O., 2004: AERMOD Description of Model Formulation. United States Environmental Protection Agency, Office of Air Quality Planning and Standards, Emissions Monitoring and Analysis Division, Research Triangle Park, EPA-454/R-03-004.

CLAG, 1994: Critical Load of Acidity in the United Kingdom. Summary Report of the Critical Loads Advisory Group, Institute of Terrestrial Ecology, Penicuik, Midlothian.

Clarke, R.H., 1979: A model for short and medium range dispersion of radionuclides released into the atmosphere. NRPB Report NRPB-R91.

Davies, B.M., Jones, C.D., Manning, A.J. and Thomson, D.J., 2000: Some field experiments on the interaction of plumes from two sources. In *Quart. J. R. Met. Soc.*, **126**, pp. 1343-1366.

DiNenno, P.J., Beyler, C.L., Walton, W.D., Custer, R.L.P. and Watts, J.M. (ed.), 1988: SFPE Handbook of Fire Protection Engineering. Society of Fire Protection Engineers, Boston, MA.

Drysdale. D., 1999: An Introduction to Fire Dynamics. Second edition. John Wiley & Sons.

Dzhelepov, B.S. and Peker, L.K., 1961: Decay schemes of radioactive nuclei. Pergamon.

EN 13725:2003: Air Quality - Determination of odour concentration by dynamic olfactometry.

Fannelöp, T.K., 1994: Fluid Mechanics for Industrial Safety and Environmental Protection. Elsevier, Industrial Safety Series, Vol. 3.

Fischer, H.B., List, E.J., Koh, R.C.Y., Imberger, J. and Brooks, N.H., 1979: Mixing in Inland and Coastal Waters. Academic Press.

Gray, D.E. (ed.), 1972: American Institute of Physics Handbook (third edition). McGraw-Hill, New York.

Griffiths, R.F., 1991: The use of probit expressions in the assessment of acute population impact of toxic releases. In *J. Loss Prev. Process Ind.*, **4**, pp. 49-57.

Gultepe I., Milbrandt J., Belair S., (2006) Visibility paramaterisation from microphysical observations for warm fog conditions and its application to the Canadian MC2 model, AMS meeting, Atlanta, January 2006

Hanna, S.R., Briggs, G.A. and Hosker, R.P., 1982: Handbook of Atmospheric Diffusion. US Dept of Energy office of Scientific and Technical Information Publication DOE/TIC-22800.

Hanna, S.R. and Paine, R.J., 1989: Hybrid plume dispersion model (HPDM) development and evaluation. In *J. Appl. Meteorol.*, **28**, pp. 206-224.

Hanna, S.R., Egan, B.A., Purdum, J. and Wagler, J., 1999: Evaluation of ISC3, AERMOD and ADMS dispersion models with observations from five field sites. Hanna Consultants Report P020. Fairfax, VA.

Hansen, M.O.L., 2008: Aerodynamics of wind turbines. Second edition. Earthscan.

HMIP, 1993: Guidelines on Discharge Stack Heights for Polluting Emissions. Environmental Protection Act 1990, Technical Guidance Note (Dispersion) D1, June 1993. HMSO.

Holtslag, A.A.M. and van Ulden, A.P., 1983: A simple scheme for daytime estimates of the surface fluxes from routine weather data. In *J. Clim. Appl. Met.*, **22**, pp. 517-529.

Hunt, J.C.R., 1985: Turbulent diffusion from sources in complex flows. In *Ann. Rev. Fluid Mech.*, **17**, pp. 447-458.

Hunt, J.C.R., Holroyd, R.J and Carruthers, D.J., 1988a: Preparatory studies for a complex dispersion model. CERC Report HB9/88.

Hunt JCR, Leibovich S and Lumley JL, 1981: Prediction method for the dispersal of atmospheric pollutant in complex terrain. Technical Report P85-81-04, Flow Analysis Associates, Ithaca, NY.

Hunt, J.C.R., Leibovich, S. and Richards, K.J., 1988b: Turbulent shear flow over hills. In *Quart. J. R. Met. Soc.*, **114**, pp. 1435-1470.

Hunt, J.C.R., Richards, K.J. and Brighton, P.W.M., 1988c: Stably stratified flow over low hills. *Quart. J. R. Met. Soc.*, **114**, pp. 859-886.

Hunt, J.C.R., Stretch, D.D. and Britter, R.E., 1988d: Length scales in stably stratified turbulent flows and their use in turbulence models. In *Proceedings IMA Conference on Stably Stratified Flow and Dense Gas Dispersion*, pp. 285-322. Ed. J.S. Puttock, Clarendon Press.

Lakes Environmental Consultants Inc., 2003: Flare sources. In *Proposed guidance for air dispersion modelling*, p. 22. Ontario Ministry of the environment.

Lamb, R.G., 1982: Diffusion in the convective boundary layer. In *Atmospheric Turbulence and Air Pollution Modelling*, pp. 159-230. Eds. F.T.M. Nieuwstadt and H. van Dop, D. Reidel, Dordrecht.

Lawson, R.E., Snyder, W.H. and Thompson, R.S., 1989: Estimation of maximum surface concentrations from sources near complex terrain in neutral flow. In *Atmospheric Environment*, **23**, pp. 321-331.

Lederer, C.M., Hollander, J.M. and Perlman, I., 1968: Table of isotopes. Sixth edition. New York, Wiley.

Macdonald, R.W, Griffiths, R.F. and Hall, D.J., 1998: An improved method for estimation of surface roughness of obstacle arrays. In *Atmospheric Environment*, **32**, pp. 1857–1864

Moore, D.J. and Lee, B.Y., 1982: An asymmetrical Gaussian plume model. Central Electricity Generating Board Report RD/L/2224N81.

Nuffield Science Foundation, 1984: Nuffield Advanced Science - Book of Data. First revised edition. Longman.

Oke, T.R., 1987: Boundary layer climates. Second edition. Methuen.

Panofsky, H.A. and J.A. Dutton, 1984: Atmospheric Turbulence, Models and Methods for Engineers and Scientists. John Wiley and Sons, New York.

Perry, S.G., 1991: CTDMPLUS: A dispersion model for sources near complex topography. Part I: Technical formulations. In *J. Appl. Meteor.*, **31**, pp. 633-645.

Sehmel, G.A. 1980: Particle and gas dry deposition: a review. In *Atmospheric Environment*, **14**, pp. 983-1011.

Snyder, W.H. and Lawson, R.E., Jr., 1991: Fluid modelling simulation of stack-tip downwash for neutrally buoyant plumes. In *Atmospheric Environment*, **25A**, pp. 2837-2850.

U.S. E.P.A., 1995: User's Guide for the Industrial Source Complex (ISC3) Models. Volume I: User Instructions (Report EPA-454/B-95-003a) ; Volume II: Description of model algorithms (Report EPA-454/B-95-003b), September 1995. U.S. E.P.A., Office of Air Quality Planning and Standards, Emissions Monitoring and Analysis Division, Research Triangle Park, NC 27711.

van Ulden, A.P. and Holtslag, A.A.M., 1985: Estimation of atmospheric boundary layer parameters for diffusion applications. In *J. Clim. Appl. Met.*, **24**, pp. 1194-1207.

Venkatram, A., 1977: Internal boundary-layer development and fumigation. In *Atmospheric Environment*, **11**, pp. 479-482.

Venkatram, A., Karamchandani, P., Pai, P. and Goldstein, R., 1994: The Development and Application of a Simplified Ozone Modelling System (SOMS). In *Atmospheric Environment*, **28**, pp. 3665-3678.

Weil, J.C., 1985: Updating applied diffusion models. In *J. Clim. Appl. Met.*, **24**, pp. 1111-1130.

CERC

Cambridge Environmental Research Consultants Ltd
3 King's Parade, Cambridge, CB2 1SJ, UK
Tel: +44 (0)1223 357 773, Fax: +44 (0)1223 357 492
Email: help@cerc.co.uk
Website: www.cerc.co.uk