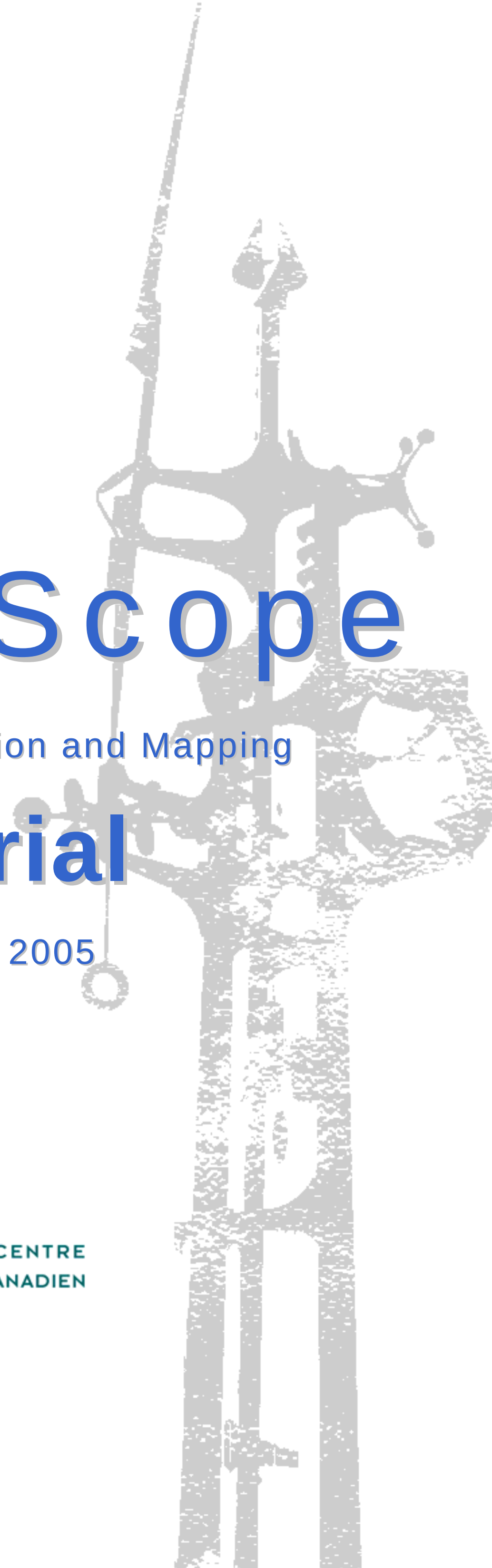
AnemoScope

Wind Energy Simulation and Mapping

Tutorial

November 2005



CHC CANADIAN HYDRAULICS CENTRE
CENTRE D'HYDRAULIQUE CANADIEN

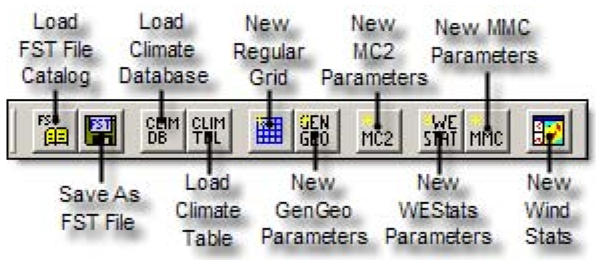
Environment Canada   Environnement Canada

AnemoScope Tutorial

Introduction

In this tutorial, you will perform a complete simulation process and produce wind statistics for a 250 000 km^2 mesoscale domain in the eastern part of Canada and a detailed wind map on a 131.8 km^2 microscale sub-domain focusing on the Montreal region.

This tutorial deals only with AnemoScope functions and does not cover general EnSim capabilities. In this tutorial, you will be using the modules listed below, which correspond to individual steps in the AnemoScope simulation process:



As you complete the tutorial, you will carry out the two main parts of the AnemoScope simulation process:

- **Part 1:** Run mesoscale wind simulations and produce wind statistics for the mesoscale domain in eastern Canada.
- **Part 2:** Run higher resolution microscale simulations and produce a detailed wind map for the microscale sub-domain around Montréal.

Part 1 of the process—producing mesoscale wind statistics—consists of 8 steps:

1. Import a reference map of the region of interest.
2. **Open** a wind climate database.
3. **Select** a climate table.
4. **Define** the mesoscale domain and the mesoscale grid.
5. **Generate** the geophysical terrain data on the mesoscale grid.
6. **Set up** and launch mesoscale simulations for selected climate states.
7. **Compute** the mesoscale wind statistics data.
8. **Check** the mesoscale wind statistics in the Wind Statistics View.

Part 2 of the process—producing a microscale wind map—consists of 3 steps:

1. **Define** a high resolution grid to support the microscale terrain data.
2. **Generate** the high resolution geophysical terrain data for the microscale grid.

3.  **Set up** and perform the microscale simulations and compute the microscale wind statistics data.

Part 2 of this process can be performed by itself if mesoscale wind statistics are already available from other sources. For example, you could use data files downloaded from the Canadian Wind Energy Atlas at http://www.windatlas.ca.

We strongly recommend that you complete Part 1 of this tutorial before trying Part 2. Because many of the skills used in Part 2 are explained in Part 1, doing the sections out of order, or skipping Part 1 entirely, may cause some confusion.

Benchmark Data

All the data needed for this tutorial is provided on the tutorial CD provided. Unzip the archive to a disk having at least 1.5 GB of free space. Ensure that the path does not contain any spaces. If you unzip the archive to **C:\CHC\Tutorial**, the resulting directory tree will be structured as follows:

- **Tutorial\InputData** - contains the GIS data needed as input for the tutorial.
- **Tutorial\MC2** – contains MC2 simulation results for all wind climate states corresponding to Annual wind climate table 55\115_table.ef.
- **Tutorial\MesoGrid** - contains files created by the GenGeo module, and used as input for the MC2 module.
- **Tutorial\MicroGrid** – contains files created by GenGeo, and used as input for the MMC module.
- **Tutorial\WEStats** - contains the mesoscale wind statistics maps produced by MC2, and used in the MMC module.
- **Tutorial\MMC** – contains the final results of the MMC module.
- **Tutorial\Settings** - contains the benchmark settings files. You will have the option to compare your own settings files produced during the tutorial with these reference files. Ideally, the two should be the same, except for some pathnames.
- **Tutorial\OutputData** - contains the benchmark output data. You will have the option to compare your own results with these reference output files.

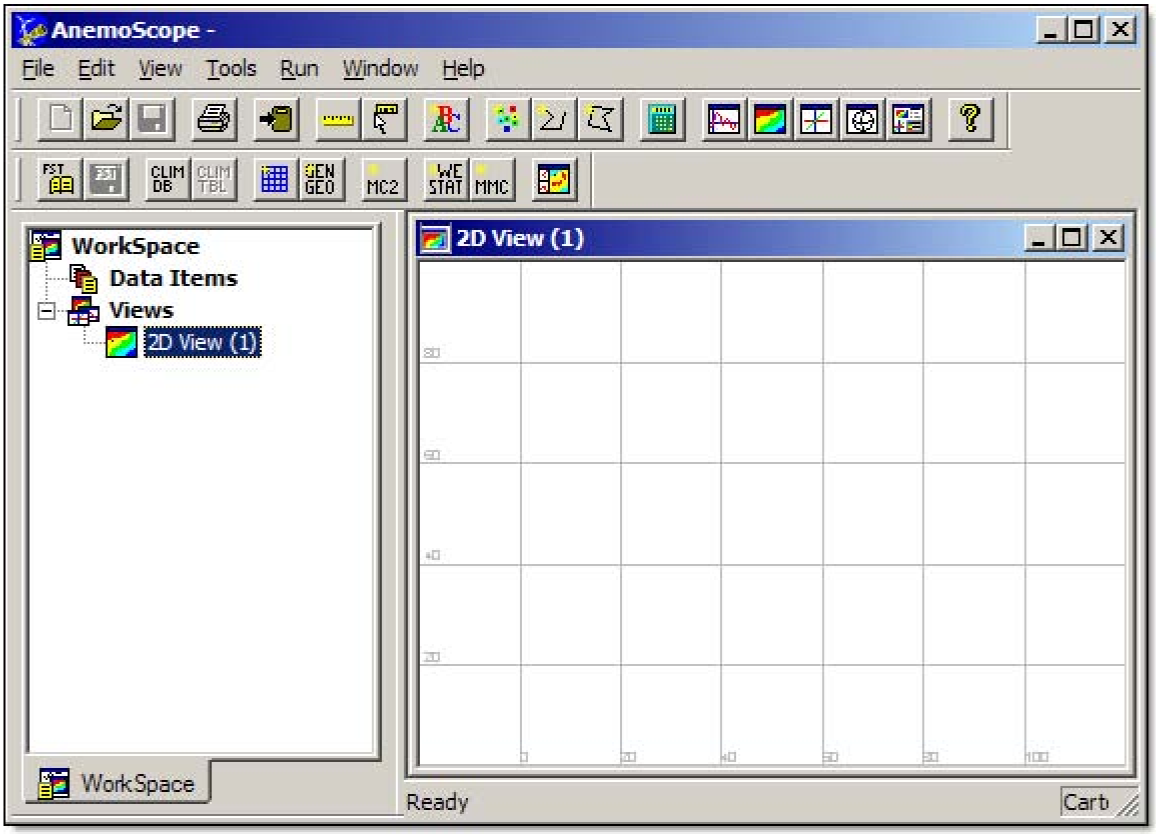
  Note: As you work through the tutorial, the directory **C:\CHC\Tutorial** will be referred to as **[TutorialDir]**.

Before You Begin

1. Start by creating a folder called **MyTutorial** in which all the data you create during the tutorial will be stored. Make sure that the path to this folder, including the name of the folder, doesn't contain any spaces. The software library used by AnemoScope and its subprograms to read and write FST files is not able to deal with pathnames containing spaces. In particular, make sure that the folder isn't

located in your **My Documents** directory. Also, be sure that the disk on which you create the folder has enough space. If you follow every step, slightly less than 350 MB of space will be required.

2. Open the AnemoScope software from your desktop or from the Start menu. The AnemoScope program window should look something like this:



The **WorkSpace** only contains a **2D View**, which is open by default. No data items are loaded. If no 2D View is present, open one using the **New 2D View** button  on the EnSim tool bar.

Note: When you see the notation “**[AnemoScope]**” used in the tutorial, it’s intended to represent the folder into which AnemoScope was installed. The default location is **C:\CHC\AnemoScope**.

Part 1: MC2 - Mesoscale Wind Statistics Production

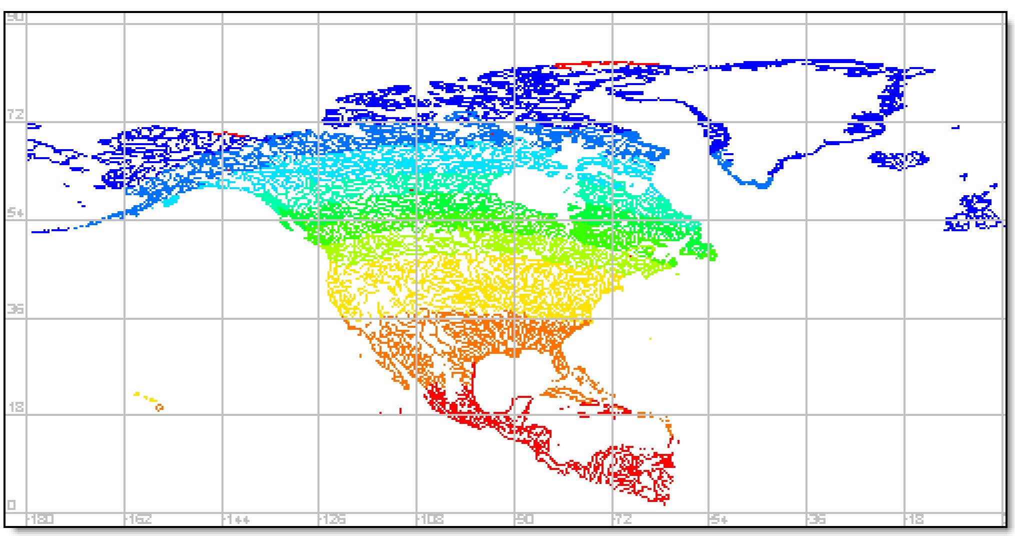
As mentioned in the introduction, this section will teach you how to produce mesoscale wind statistics. By the time you're finished Part 1, you'll know how to use AnemoScope, the Wind Climate database, and the GenGeo database to produce a map containing a wide range of statistical data for a particular region.

You'll need to do the steps in order, since each one builds on the results of the previous step. If you get lost, or you'd like to review a particular part of the process, you can use the benchmark data from the previous lesson, which is included with AnemoScope.

Step 1: Import a Map of the Region of Interest

By the end of this step, you will have imported a map from the MapInfo Interchange File format and displayed it in a 2-dimensional view window. You'll use this map as a reference when choosing how to locate your model grid.

1. Use the Windows Explorer to create a folder named **MyMeso** in your **MyTutorial** directory. Remember, there's no space between **My** and **Meso**.
2. Import the reference map, which is stored in the MapInfo Interchange File format. From the menu bar, select **File→Import→MapInfo Interchange File**.
3. Use the **Import** dialog to select the file **Coasts.mif** located in **[TutorialDir]\InputData\Maps\North_America_Atlas10M**. The **Coasts.mif** object will appear in the Data Items list, with the map itself shown as a 2-dimensional line set child object.
4. Within the WorkSpace, drag the line set object (the one with this icon: ) to the 2D View. A hydrological map of North America in Lat/Long coordinates is displayed:

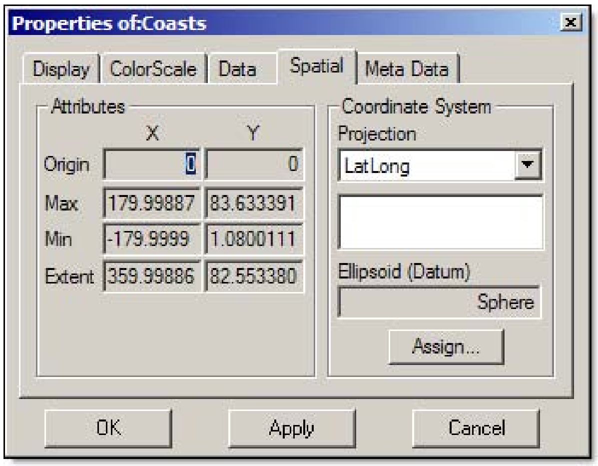


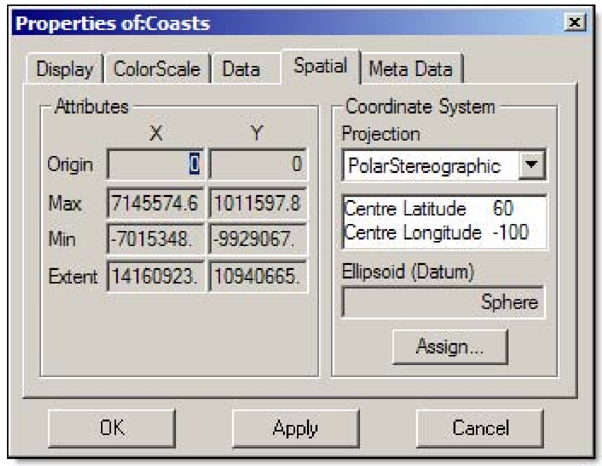
5. Now, transform the data into the Polar Stereographic coordinate system, which is the system used by AnemoScope. Double-click on the 2-dimensional line set

object, or select **Properties** from the map's shortcut menu (which you can open by right-clicking on the object in the WorkSpace) to open the Properties dialog box and select the **Spatial** tab.

a) First, change the Projection by expanding the list of available projections and selecting **Polar Stereographic** on the **Spatial** tab of the Properties dialog.

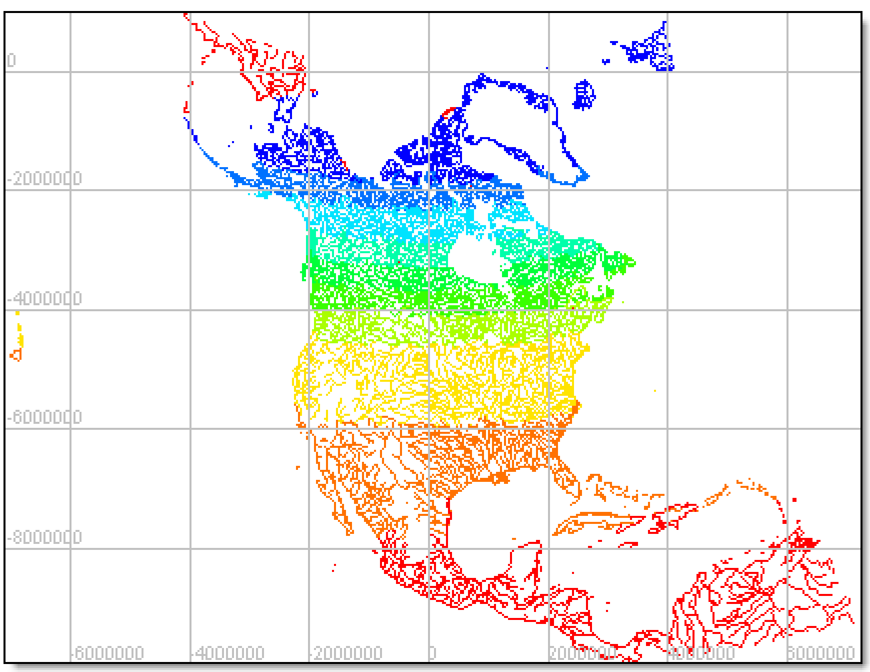
b) Set the Centre Longitude value to **-100** (i.e., 100° W), which is approximately central for Canada. The Centre Latitude should be left as it is (the default value of 60° N). The Ellipsoid (Datum) should also be left as it is. The Properties window should now look as shown below:





c) Click **Apply** to confirm your changes. The map is changed in the 2D View and is displayed in the Polar Stereographic projection.

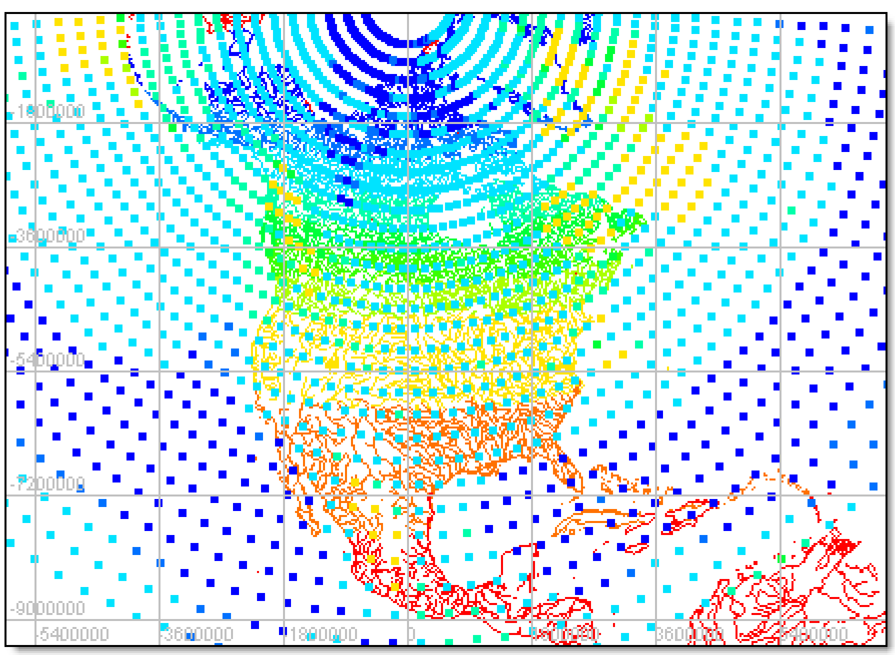
6. Click **OK** to close the Properties window. The map displayed in the 2D window should resemble the one shown below.



Step 2: Open the Wind Climate Database

At the conclusion of this step, you'll know how to open the Wind Climate Database, convert it to Polar Stereographic coordinates, and display the database in a 2-dimensional view window, along with your reference map.

1. Click the **Load Climate Database** button on the AnemoScope tool bar and select the **[AnemoScope]\WindClimateDB_Annual** directory in the **Browse For Folder** box. The Annual wind climate database index file is loaded. Click **OK** when the loading is finished.
2. Within the WorkSpace, drag the new point set object WindClimateDB_Annual to the 2D View. An error message is displayed. This error is shown because the wind climate database is in Lat/Long coordinates, while the reference map is in the Polar Stereographic projection. Click **OK** to close the error message.
3. Double-click the WindClimateDB_Annual point set object within the WorkSpace to open its **Properties** dialog box. Select the **Spatial** tab to modify the **Coordinate System** and convert the wind climate database to the same coordinate system as the reference map.
   - a) First, change the Projection by expanding the list of available projections and selecting **PolarStereographic**.
   - b) Set the **Centre Longitude** value to **-100**. The **Ellipsoid (Datum)** should be left as it is.
   - c) Click **Apply** to confirm your changes. Click **OK** to close the Properties box.
4. Drag the WindClimateDB_Annual object to the **2D View** again. The location of each wind climate table is shown by a square point superimposed on the map of North America.



Step 3: Select a Wind Climate Table

At the end of this section, you will have selected a wind climate table from those in or near your region of interest. The data from this climate table will be used in several of the later steps to provide information on wind conditions that have been observed in the region.

1. **Import** the **Cities.mif** file, and then convert its point set to Polar Stereographic coordinates, as described in Step 1. This file is located in **[TutorialDir]\InputData\Maps\North_America_Atlas10M**.

2. This file contains a set of points with 6 different attributes. We only want to display one of these attributes: city names. Within the WorkSpace, double-click on Cities (UIDENT), or select **Properties** from its shortcut menu (by right-clicking on its name) to open the **Properties** dialog box.

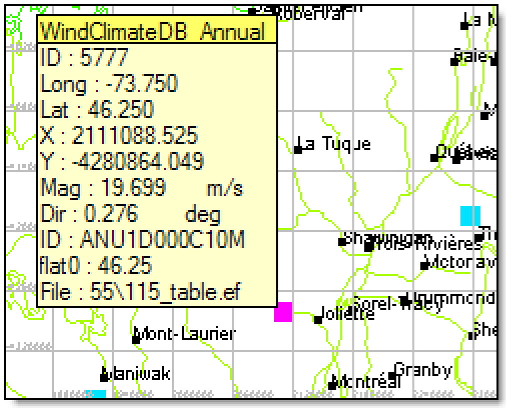
3. Select the **Data** tab. The UIDENT attribute is selected. Click on the NAME attribute. Click **Apply**. Switch to the **Display tab** and check the **Show Node Label** box. Click **OK**.

4. Center the view on the Montreal Region. Zoom in (**<Ctrl>**+left mouse button and drag upwards, or scroll up with your mouse wheel) and move (drag with the left mouse button).

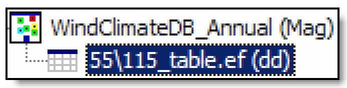
5. Increase the size of the square points representing the climate table locations. Double-click the WindClimateDB_Annual point set object within the WorkSpace to open its **Properties** dialog box. In the **Display** tab, increase the **Point Size** value to **10** using the up arrow. Confirm your changes with **Apply**. Click **OK** to close the Properties box.

6. We will select table **55\115**, which is the closest to the Montreal region. Zoom in until the view looks like the above image. This location of this table is just northwest of Montreal, next to Joliette. It is circled in the picture above. Double-click on the square point.

   If you can't select the point, make sure that the WindClimateDB_Annual object is the currently selected object in the WorkSpace panel. When you've double-clicked on the point, an information box will be displayed. Its last line indicates the name of the table. Make sure you have selected **File: 55\115_table.ef**. If you clicked on the wrong point, make your selection again. You should see the popup display shown below.



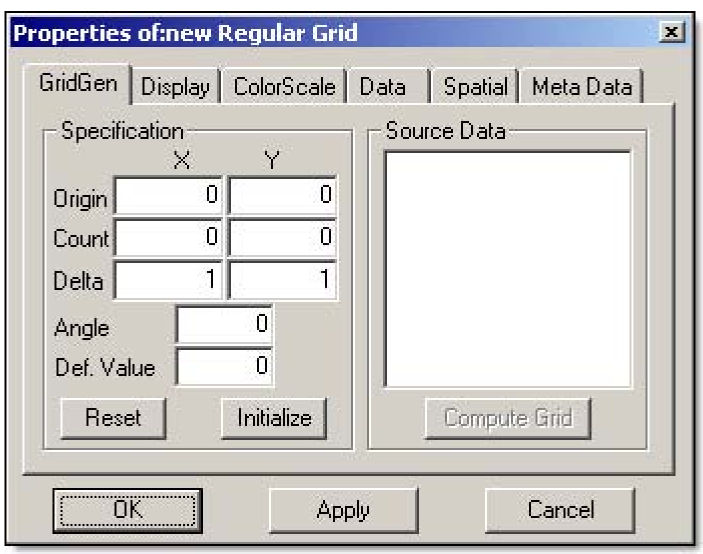
7. When 55\115_table.ef is selected, load the table with the **Load Climate Table** button on the AnemoScope tool bar or right click to show the context menu and select **"Load Wind Climate Table"**. A Table child object will be added in the WorkSpace to the WindClimateDB_Annual point set object:



Step 4: Define a Mesoscale Grid

By the end of this step, you will have created and described a regular grid that will later be used by GenGeo to define the mesoscale terrain features and by MC2 to calculate the mesoscale wind map.

1. Click the **New Regular Grid** button on the AnemoScope tool bar. An empty Regular Grid object, with the icon, is added to the Data Items list, and its Properties dialog pops up. It opens on the GridGen tab, which is initially empty.



2. To create a grid approximately centred on Montreal we have to specify the coordinates of the **Origin** of the grid (the lower left corner), the **Count** (the number

of cells in each direction), and the cell size **Delta**.

In the 2D View, you can see that the Montreal point is roughly located at X = 2159000 and Y = -4355000.

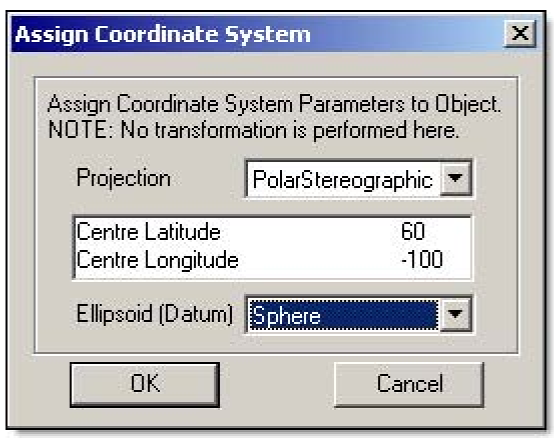
In the Specification frame of this tab, enter the following values:

- **Origin:** X= 1880000.0; Y= -4560000.0
- **Count:** X= 100; Y=100
- **Delta:** X= 5000; Y= 5000 (These values are in metres.)

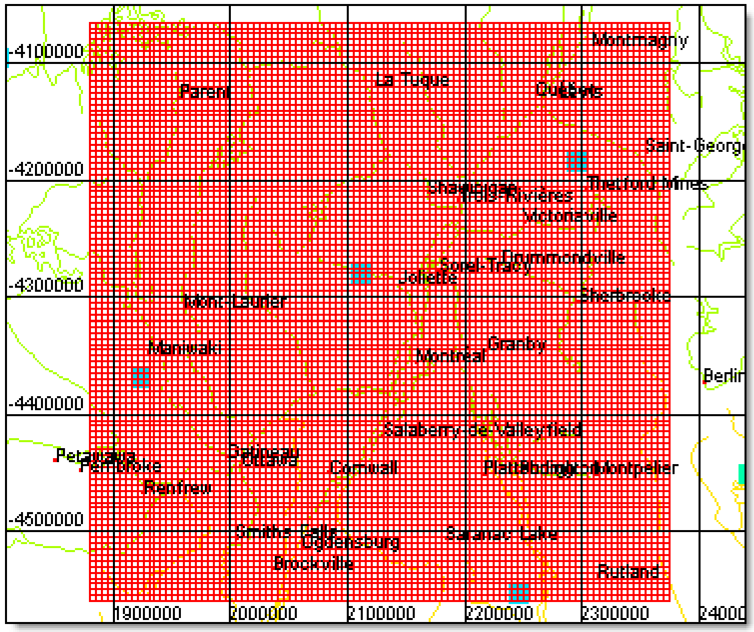
  Note: The Delta X and Delta Y values must always be equal. If you would like to experiment with different Delta values, always make sure that the X and Y values are the same.
- **Angle:** 0 (This angle should always be 0, since the FST format doesn't support rotated grids.)
- **Def. Value:** 0 (This value will be overwritten by GenGeo.)

Click **Initialize** to create the grid. Note that the new grid object's icon changes to  in the Data Items list.

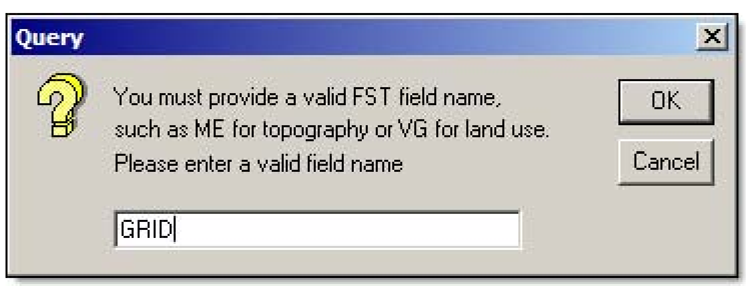
3. Switch to the Spatial tab of the Properties box in order to assign a coordinate system to this new grid. Click the **Assign…** button and select the following parameters:



4. Click **OK** to confirm.
5. Drag the new grid object to the 2D View within the WorkSpace. Zoom in **(<Ctrl>**+left mouse button and drag upwards, or scroll up with your mouse wheel). The 2D view window should now look something like the picture below.



6. If the grid looks good, click **OK** to close the Properties box. If your grid doesn't look like the one pictured, double-check the numbers in the GridGen tab, and make sure that you used the correct Centre Latitude and Longitude values on the Spatial tab.
7. Make sure that the new Regular Grid object is selected in the WorkSpace and click the **Save File** button on the EnSim tool bar to save the grid in EnSim .r2s format. Save the grid as **MyMesoGrid.r2s** in your **MyTutorial\MyMeso** folder.
8. Export the grid to FST format by clicking the **Save As FST** button on the AnemoScope tool bar. Save the grid as **MyMesoGrid.fst** in your **MyTutorial\MyMeso** folder.



After you click **Save**, a message box will appear, asking you to enter a valid FST field name. Enter **GRID**, indicating that this grid will contain no data, but only a grid definition. Click **OK**.

<u>If the export fails:</u>

The export procedure will fail if the path to your grid file contains any spaces.

In this case, check the location and the name of your MyTutorial and MyMeso folders. If their locations contain any spaces in their paths (for example, if they're in your **My**

Documents folder) then you will have to copy the directories to appropriate locations. If you used a space in either of the folder names, you'll have to rename the folders.

If one of the directories was just accessed by AnemoScope, you won't be able to copy the folders right away. AnemoScope keeps a link to the most recent folder used. To deactivate the folder, open a file located outside your **MyTutorial** folder.

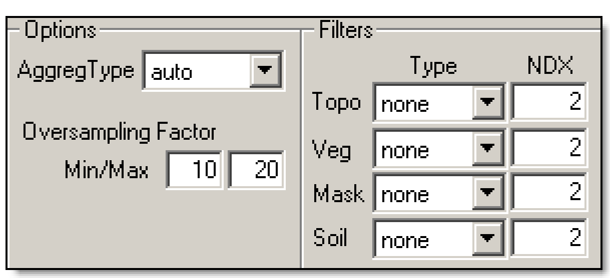
Save Your WorkSpace

At this point, it may be useful to save your WorkSpace by selecting **File→Save WorkSpace…** from the menu bar. Save your WorkSpace as **MyWorkSpace1** in your MyTutorial\MyMeso folder. It will be saved as an EnSim WorkSpace File (*.ews). This way, your AnemoScope session can be ended and restarted at a later time. However, note that any coordinate system settings **Assigned** or **Applied** to data items are not saved and will not be restored. You will have to perform them again when the WorkSpace is reloaded, if you need to repeat any of the earlier steps.

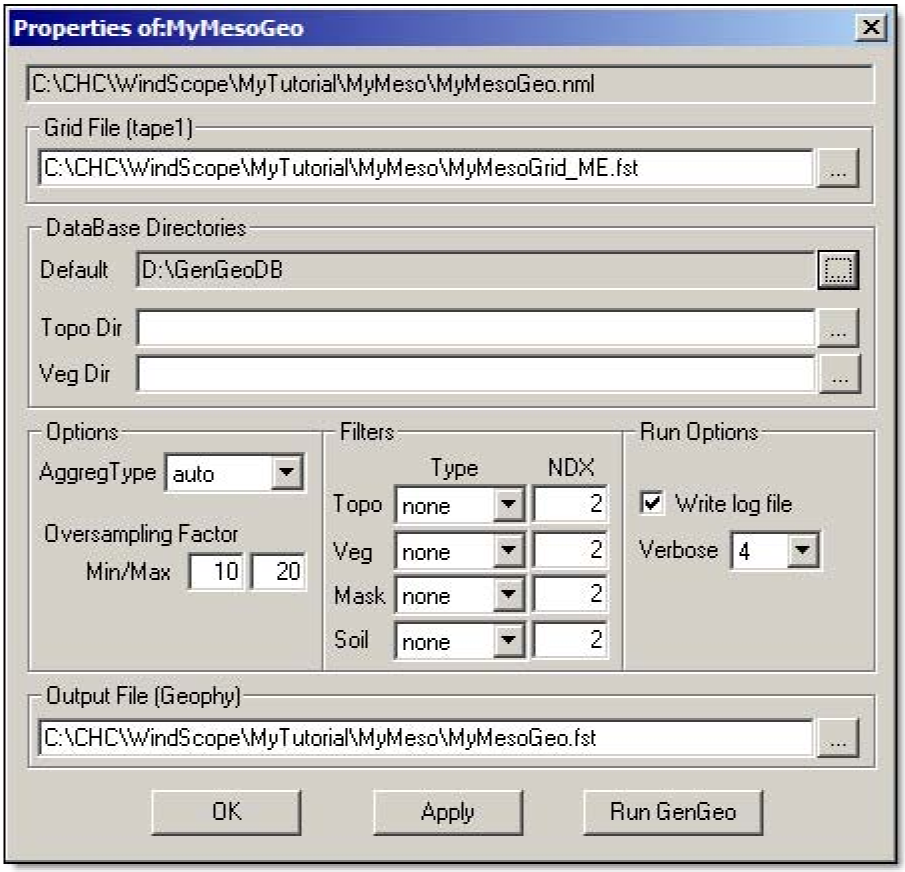
Step 5: Generate Geophysical Terrain Data for the Mesoscale Grid

When you've finished this step, you will have created a mesoscale terrain grid, including topographical and vegetation data, which will be used by MC2 to create the mesoscale wind map.

1. Click the **New GenGeo Parameters** button on the AnemoScope tool bar. A new GenGeo parameter file object is added to the WorkSpace Data Items list and its Editor/Property Page pops up.
2. **Grid File (tape1)**: use the **Browse** button to select the mesoscale FST grid file **MyMesoGrid.fst** from your MyTutorial\MyMeso folder.
3. **DataBase Directories:**
   a) **Default:** use the **Browse** button to select the **GenGeoDB** directory. The GenGeo database is provided with the software distribution as an archive file named **Gengeo.zip**. You'll have to select the **GenGeoDB** directory at the location to which you extracted it from the archive when you installed AnemoScope.
   b) **Topo Dir:** leave this entry blank. This time around, the Default Directory will be used as Topo Dir.
   c) **Veg Dir:** leave this entry blank. The Default Directory will also be used as the Vegetation Directory.



4. **Options and Filters**: keep the default parameters.
5. **Run Options**: check the ☑ **Write log file** box to record a log file. Set the **verbose** level to **4** to get the maximum level of information.
6. Click **Apply** to confirm your settings.
7. **Output File (Geophy)**: use the **Browse** button [...] to go to your **MyTutorial\MyMeso** folder. In the **File Name** box type **MyMesoGeo**. The **Save as type** box should say FST file (*.fst). Click **Save** when you're done.



8. Click the **Run GenGeo** button to launch the GenGeo program execution.
9. You are asked to specify the namelist file in which the input parameters will be stored. Type **MyMesoGeo** in the **File Name** box. The **Save as type** box indicates GenGeo Parameter Set (*.nml). Click **Save** when done.
10. When the execution starts, a DOS Window will open. Success and error messages are displayed in this window. If an error occurs, it may be an indication that your settings are not correct. Make sure you entered the right information in the Namelist File Editor dialog. When the calculations are complete, which may take as long as 90 seconds, depending on your machine, press **<Enter>** to close the window.
11. After a successful execution, click **OK** to close the GenGeo Namelist File editor.
12. Using the Windows Explorer, open the **GenGeo.log** file in your MyTutorial\MyMeso

folder. It can be viewed in Notepad or any other text editor. The file should not contain any error messages and the Data Coverage values should reach 100% for all the fields processed. You can also compare it with the example file located in **TutorialDir]\MesoGrid\GenGeo.log**. Except for the path definitions, they should be similar.

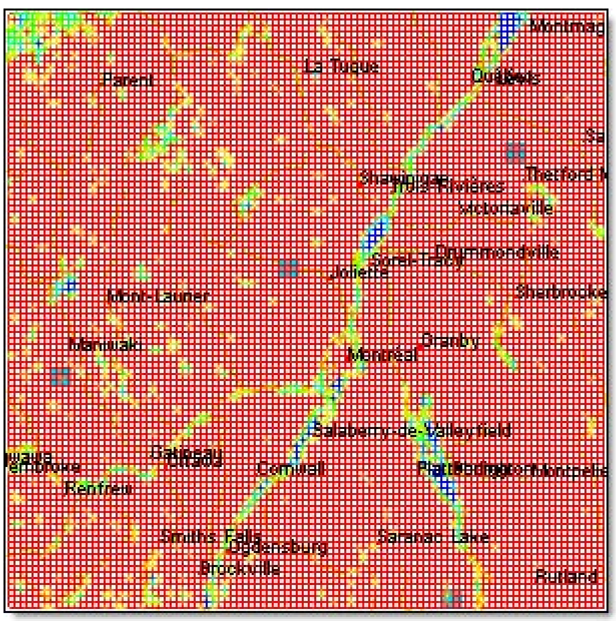
13. Click the **Show FST File Catalog** button on the AnemoScope tool bar to see the catalog of your MyMesoGeo.fst file. Select the file in your **MyTutorial\MyMeso** folder and click **Open** to view it. The contents of the file will be displayed in table format:

Catalog of: C:\CHC\MyTutorial\MyMeso\MyMesoGeo.fst

nomvar	etiket	typvar	grtyp	dateo	deet	npas	NI	NJ	NK	IP1	IP2	IP3	IG1	IG2	IG3	IG4	datyp	nbits
>>	POSX	X	N	0	0	0	100	1	1	464	470	147833	0	0	1000	10	5	32
^^	POSY	X	N	0	0	0	1	100	1	464	470	147833	0	0	1000	10	5	32
MG0		C	Z	0	0	0	100	100	1	0	0	0	464	470	147833	0	1	16
VF		C	Z	0	0	0	100	100	1	1199	0	0	464	470	147833	0	1	16
VF		C	Z	0	0	0	100	100	1	1198	0	0	464	470	147833	0	1	16
VF		C	Z	0	0	0	100	100	1	1197	0	0	464	470	147833	0	1	16
VF		C	Z	0	0	0	100	100	1	1196	0	0	464	470	147833	0	1	16
VF		C	Z	0	0	0	100	100	1	1195	0	0	464	470	147833	0	1	16
VF		C	Z	0	0	0	100	100	1	1194	0	0	464	470	147833	0	1	16
VF		C	Z	0	0	0	100	100	1	1193	0	0	464	470	147833	0	1	16
VF		C	Z	0	0	0	100	100	1	1192	0	0	464	470	147833	0	1	16
VF		C	Z	0	0	0	100	100	1	1191	0	0	464	470	147833	0	1	16
VF		C	Z	0	0	0	100	100	1	1190	0	0	464	470	147833	0	1	16
VF		C	Z	0	0	0	100	100	1	1189	0	0	464	470	147833	0	1	16

14. The GenGeo output can also be visualized in a 2D view. Click the **Open File** button on the EnSim tool bar. Set **Files of type** to FST File (*.fst), and select and **Open** your MyMesoGeo.fst file.

15. Select the **MG** (land/water mask) field (not **MG0**) by checking the box in front of its name. Click **OK** to load the file, and again when the loading is complete.



16. Within the WorkSpace, drag the regular grid object **MG** to the 2D View. MG

appears as a child of the FST object MyMesoGeo. The MG field is displayed in wireframe mode by default. The blue regions are water regions. The St. Lawrence river is clearly visible.

Validation of Your GenGeo Results

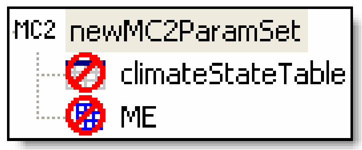
We will now compare your results with the benchmark results.

1. Click the **Open File** button on the EnSim tool bar. Set **Files of type** as FST File (*.fst). Select and **Open** the file **[TutorialDir]\MesoGrid\MesoGeophy.fst**. Select the **MG** field and click **OK**.
2. You have now two MG regular grid objects open. In order to avoid confusion, we'll change the name of the benchmark grid. Double-click the MG object child of MesoGeophy to open its **Properties** dialog. Switch to the **Meta Data** tab.
3. The second line is "**Name MG**". Click on MG and replace the value with **MG-bench**. Click **Apply** and **OK**. The object name will change in the WorkSpace.
4. Within the WorkSpace, drag the regular grid object **MG-bench** to the 2D View. There should be no visible difference between the two grids.
5. We will now compare the two results quantitatively by creating an object that will show any differences between the two grids:
   a) Select the **MG-bench** object in the WorkSpace. Click the **Calculator** button on the EnSim tool bar.
   b) In the **A** box, select **MG**, using the down arrow to expand the list of fields.
   c) In the **B** box, select **MG-bench**.
   d) In the **Expression** box, type **A-B** .
   e) In the **Result** box, type the **Name** of the resulting field, **MG-diff**. Leave the **Units** box blank.
   f) Click **Evaluate**.
6. The regular grid object **MG-diff** is added to the WorkSpace. Drag this object to a 2D View. It is displayed in wireframe mode and should be entirely red, indicating that there is no difference between the fields. Open the **Properties** dialog for the **MG-diff** object. Select the **Data** tab and note that the **Min** and **Max** values should both be 0
7. Click the **Active Cursor** button on the EnSim tool bar. Move the mouse over the grid in the 2D View. The cursor should display a value of 0.000 everywhere on the grid. Deactivate the Active Cursor by pressing **<Esc>**.
8. If your results don't match, check all of your previous steps. In particular, check your GenGeo dialog entries against the example shown earlier. If the entries are the same, remove the MG-bench FST object from the WorkSpace by right-clicking on it and selecting **remove**. Answer **No** to the question "Would you like to save the changes?". Remove the MG-diff regular grid object the same way.

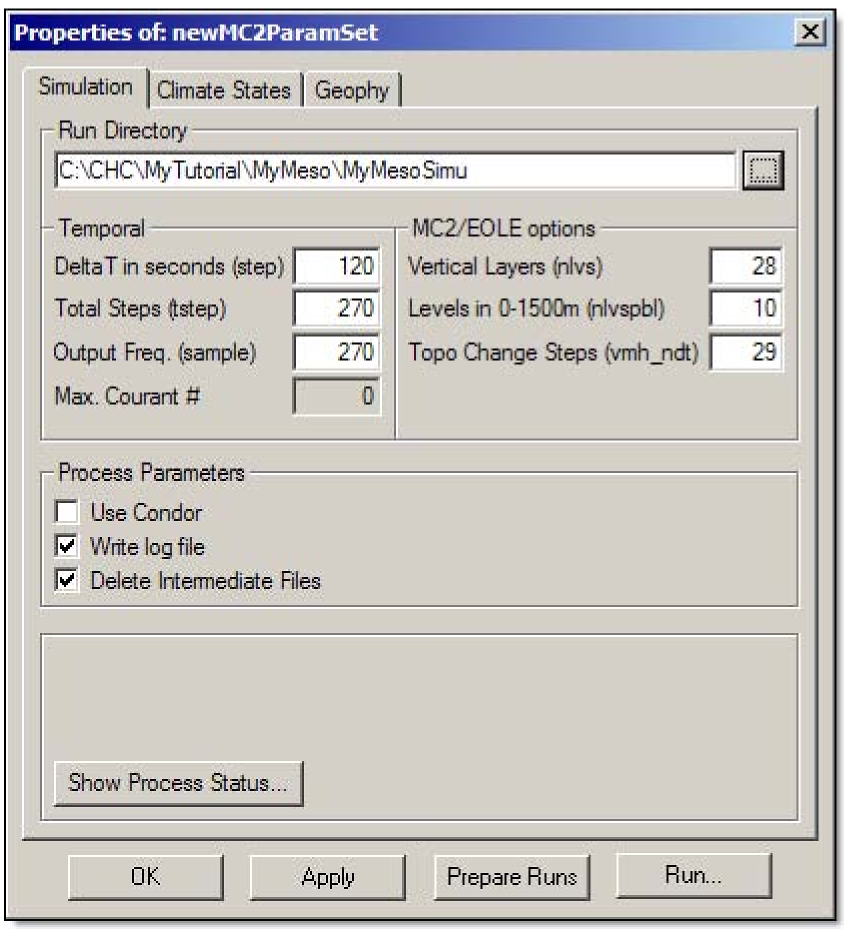
Step 6: Set Up and Perform the Mesoscale Simulations

After this step, you will have configured and run the MC2 module, which combines the mesoscale terrain grid you created in Step 5 with the climate information found in the Wind Climate database to create a series of wind maps for each of the climate states. These maps are then combined by the WEStats module to create the mesoscale wind statistics map.

1. Using the Windows Explorer, create a new folder named **MyMesoSimu** inside your **MyTutorial\MyMeso** folder. Remember, there should be no spaces in the folder name.



2. Click the **New MC2 Parameters** button on the AnemoScope tool bar. The MC2 Parameter File Editor pops up. In addition, a new MC2 parameter file object is added to the Data Items list in the WorkSpace. This object has two child objects which are empty at this stage.



3. First, fill out the **Simulation panel**:

   a) **Run Directory**: Use the **Browse** button to select your MyMesoSimu folder. All the outputs produced by the MC2 simulations will be placed in this folder.

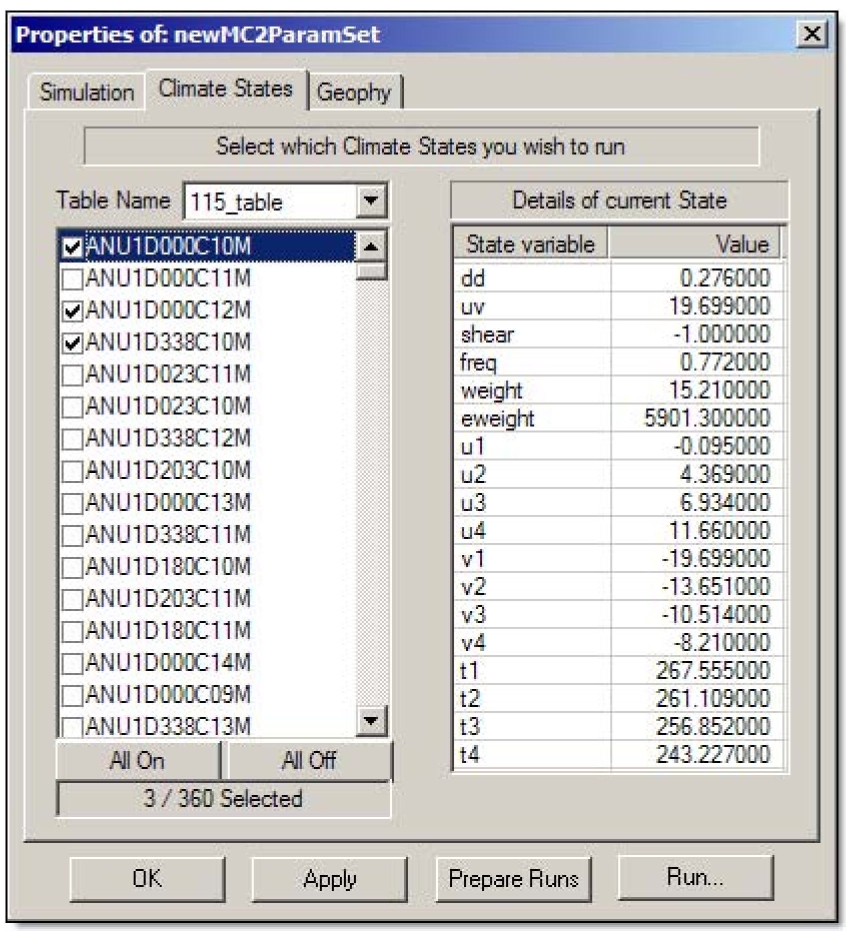
   b) **Temporal**: Keep the default values for the time step **Delta T** (120 s), the number of time steps **Total Steps** (270) and the **Output Freq.** (270). They are well suited for our 5 km resolution map of the Montreal region, which presents only a moderate topography.

   c) **MC2/EOLE options**: Keep the default values for the number of **Vertical Layers** (28), the number of **Levels in 0-1500m** (10), and **Topo Change Steps** (29).

   d) **Process Parameters:** Leave the **Use Condor** box unchecked. The runs will be performed interactively, using only your computer. All the Condor entries will be greyed out. Check the ☑ **Write log file** option, which will store the run information in a log file. Check the ☑ **Cleanup Intermediate Files** option, which will delete the intermediate results.

   e) Click **Apply** to confirm the settings. If you don't apply the settings before switching to another tab, any changes you've made will be lost.

4. Click the **Climate States** tab to move to the Climate States panel.



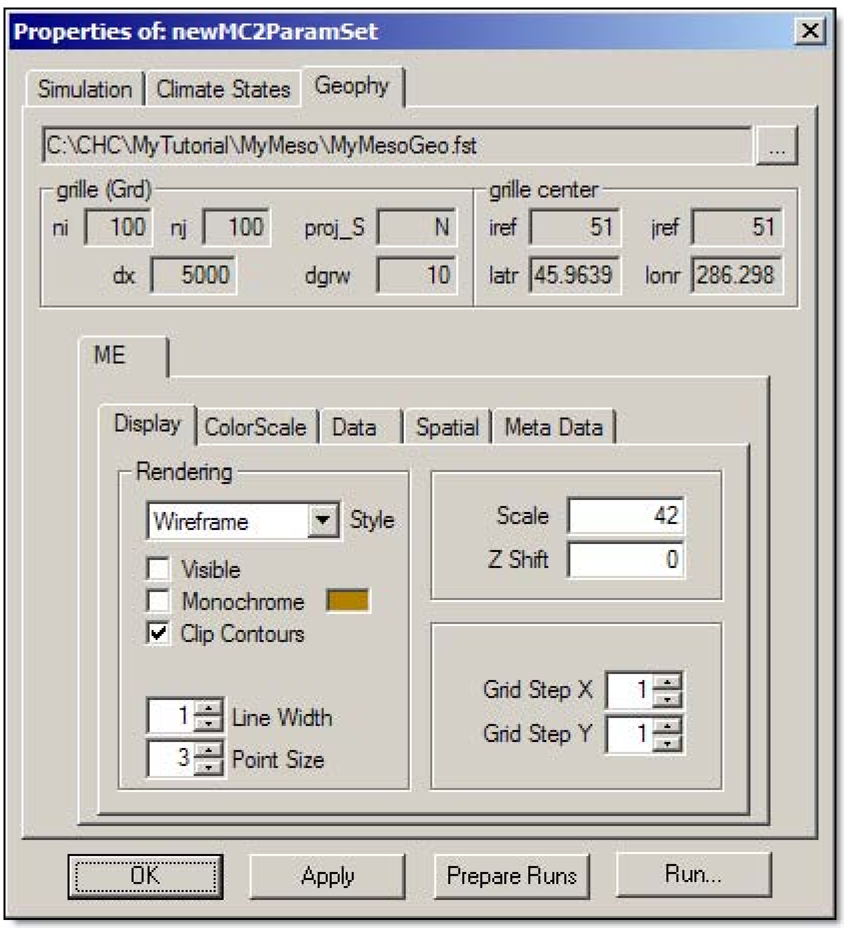
a) **Table name**: Click on the down arrow to expand the list of the tables currently available in the WorkSpace. Only table 55\115 should be available. Select it. If the table is not listed, repeat **Step 3: Select a Wind Climate Table**.

b) The contents of the climate table are displayed in two columns. On the left side, the climate states of the table are listed. On the right side, the parameters of the highlighted climate state are shown. Click on a climate state to highlight it and display its parameters.

c) To generate accurate mesoscale wind statistics, you'll have to perform one mesoscale simulation for each of the climate states. In this case, that would be 360 simulations. For this tutorial, we will perform only three simulations. Select the first climate state (ANU1D000C10M), the second state (ANU1D000C12M), and the fourth state (ANU1D338C10M) by clicking their checkboxes.

5. Click **Apply** again to confirm your selections.

6. Click the **Geophy** tab to switch to the Geophy panel.



a) Use the **Browse** button to select the **MyMesoGeo.fst** file that you created in Step 5. The **grille (Grid)** parameters are shown in the panel.

b) Click **Apply** to confirm. Notice that in the WorkSpace, the two child objects of the MC2 parameter file object are now defined.

7. Go back to the **Simulation** panel and double-check your entries. Notice that the Courant Number field is now filled in.

8. When you're done double-checking, click the **Prepare Runs** button. If any error or warning messages appear, double-check your settings again. You will then be asked to specify the file in which the MC2 input parameters will be stored. Save this file as **MyMesoMC2.mc2** in your **MyTutorial\MyMeso** folder. The file structure required for the MC2 simulations is automatically prepared in your MyMesoSimu folder; three folders (one for each climate state) and a Python script which will launch the simulations are created.

   Note: Running the MC2 modules can take a very long time, depending on the speed of your computer. On a typical Pentium 4 computer, running three simulations will take about 90 minutes. Running the complete set of simulations for the 360 climate states would take about a week. If you would prefer to skip this step, just read the following instructions and continue on with the benchmark data.

9. Click the **Run…** button to launch the MC2 simulations. Answer **Yes** to the question to confirm that you want to launch the simulation. A DOS Window pops up. The progress of the simulation is displayed in this window. At the end of the execution, press **<Enter>** to close this window.

10. After a successful execution, click **OK** to close the MC2 Namelist File Editor. If any errors have occurred, recheck your settings and launch the simulation again after making any corrections.

11. Whether you still have errors or you've successfully run the MC2 module, it's a good idea to check your results.

    a) In the Windows Explorer, go to your **MyMesoSimu** folder. Three subfolders were created, one for each simulated case. Their names are the identifiers of the climate state: ANU1D000C10M, ANU1D000C12M, and ANU1D338C10M. Open the ANU1D000C10M folder.

    b) Check the **status.dot** file by opening it with a text editor, such as Notepad. It should contain the following lines:

    ```
    _status=ABORT;_startstep=0000000000;
    _endstep=0000000270;
    _status=ED;
    ```

    If you don't see all of these lines, turn on Word Wrap under the Format menu in Notepad.

    c) Open the pre-processor log file (ANU1D000C10M _mc2ntr.log) for a quick check. Look for the line

    ```
    MC2NTR ENDED NORMALLY
    ```

    close to the end of the file.

    d) Open the log file of the simulation (ANU1D000C10M _mc2dm.log) for a quick check. Look for the line

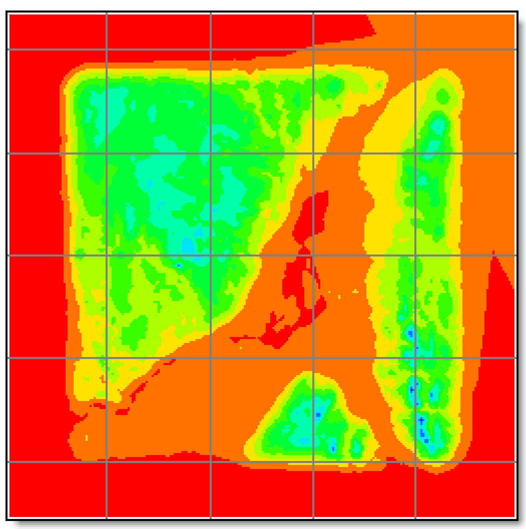
```
MC2DM RUN ENDED NORMALLY
```

close to the end of the file. Also, check all the lines that have a **MAX COURANT NUMBER** entry. The values should not exceed 1, indicating that the simulation is stable. A value of 0.4 is considered ideal. The maximum value reached in this simulation is 0.8618066 , which is fairly high, but within acceptable parameters.

e) The results of the individual MC2 simulations can also be examined to detect possible problems. Click the **Open File** button on the EnSim tool bar. Set **Files of type** to FST File (*.fst), and **Open** the file **dm1998010100-00-00_000270p.fst** in your **MyTutorial\MyMeso\MyMesoSimu\ANU1D000C10M\output** folder. This is the output of your last time step.

f) Select the **P0** (ground pressure) field by checking the box in front of its name. Click **OK**.

g) Within the WorkSpace, drag the regular grid object **P0** (shown as a child of dm1998010100-00-00_000270p) to a 2D View. By default, the P0 field is displayed in wireframe mode. Open its **Properties** dialog, go to the **Display** tab and select **Filled Contours** as the **Rendering Style**. It should look like the example below. Note that the first grid cell row and column are not used by the dynamic fields.



Validation of Your MC2 Results

Now we'll compare your results with the benchmark results.

1. Click the **Open File** button on the EnSim tool bar. Set **Files of type** to FST File (*.fst), and **Open** the **[TutorialDir]\MC2\Output\ ANU1D000C10M\m1998010100-00-00_000270p.fst** file. Select the **P0** field.

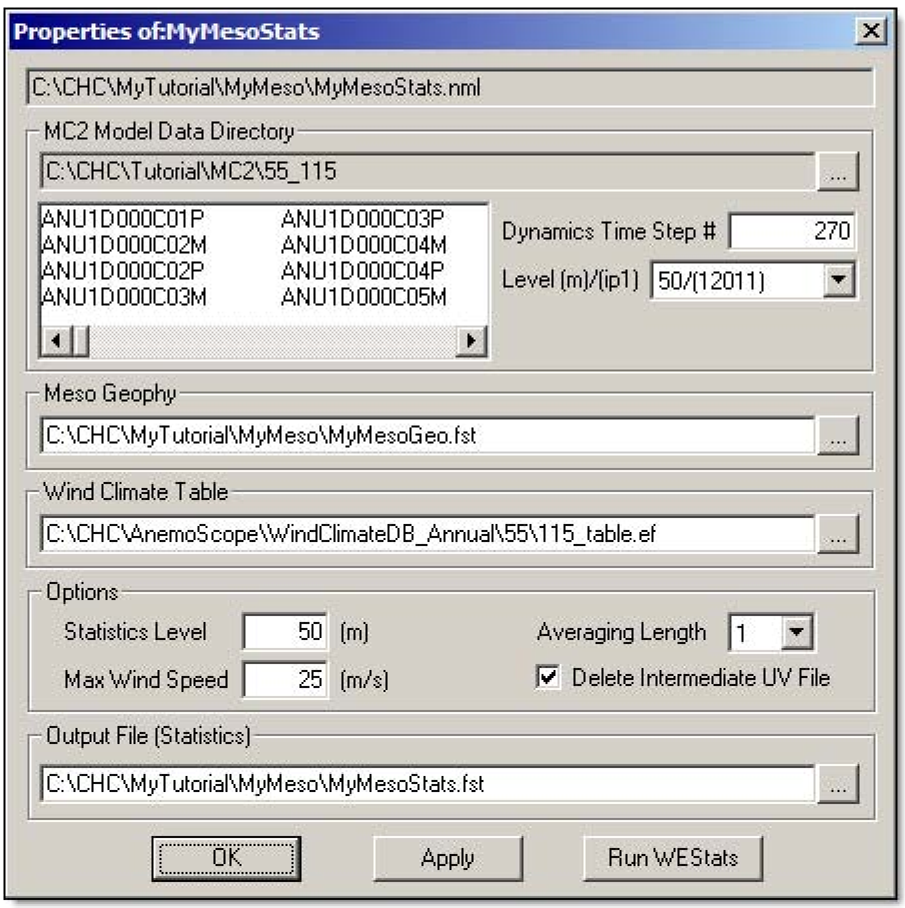
2. You have now two **dm1998010100-00-00_000270p** and two **P0** regular grid objects in your WorkSpace. In order to avoid confusion we will change the names of the benchmark data. Double-click on the second dm1998010100-00-00_000270p object on the Data Items list to open its **Properties** dialog box. In the **Meta Data** tab, click on the name value and replace it with **dm-bench**. Click **Apply** and **OK** when satisfied. Notice that the object name changes in the WorkSpace.
3. Double-click on the P0 object child of dm-bench to open its **Properties** dialog box. Switch to the **Meta Data** tab. Replace “P0” with “P0-bench”. Click **Apply** and **OK** when you’re finished. Note that the object name is changed in the WorkSpace.
4. Within the WorkSpace, drag and drop the regular grid object P0-bench in the 2D View. There should be no difference between the two objects.
5. We will now compare the two results quantitatively.
   a) Click the **Calculator** button on the EnSim tool bar. If the button is not active, select the P0-bench object in the WorkSpace.
   b) As the **A** entry, select **P0**, using the down arrow to expand the list of available fields.
   c) As the **B** entry, select **P0-bench**.
   d) In the **Expression** box, type **A-B** .
   e) In the **Result** box, type the name for the resulting field, **P0-diff**. Do not specify any units.
   f) Click **Evaluate**.
6. The regular grid object **P0-diff** is added to the WorkSpace. Drag this object into the 2D View. It is displayed in wireframe mode and should be completely red (to see it better, remove the P0 object from the 2D View).
7. Click the **Active Cursor** button on the EnSim tool bar. Move the mouse over the grid in the 2D View. The cursor should display a value of 0.000 everywhere. Deactivate the Active Cursor by pressing **<Esc>**.
8. If the two grids don’t match, double-check your settings. In particular, check your MC2 dialog entries against the examples shown earlier. You can also compare ANU1D000C10M_mc2ntr.log and ANU1D000C10M_mc2dm.log with the benchmark log files provided in **[TutorialDir]\MC2\Output\ANU1D000C10M**
9. If you’re finished with the comparisons, remove the dm-bench object from the WorkSpace by opening its **shortcut menu** and selecting **Remove**. Answer **No** to the question “Would you like to save the changes?”. Remove the P0-diff regular grid object the same way.

Step 7: Compute the Mesoscale Wind Statistics Data

When you're finished this step, you'll have created a mesoscale wind energy statistics map from individual climate state wind maps created by the MC2 model.

In the previous steps you performed mesoscale simulations for only three climate states. In order to be representative, the wind statistics must be computed for a broad range of climate states. To save you from having to perform the hundreds of MC2 simulations required, the Tutorial directory **[TutorialDir]\MC2\55_115** contains the MC2 simulation results for the entire set of Climate States corresponding to table 55\115.

1. Click the **New WEStats Parameters** button on the AnemoScope tool bar. The **WEStats Namelist File Editor** appears. In addition, a new WEStats namelist file object is added to the Data Items list in the WorkSpace. Complete the editor as follows:



2. **MC2 Model Data Directory**: Use the **Browse** button to select the **[TutorialDir]\MC2\55_115** directory. All of the simulations (360) available in this directory are listed in the frame below, identified by the name of the climate state. The results of these simulations will be processed to define the mesoscale wind statistics.

   a) **Dynamics Time Step #**: Set this value to **260** and click **Apply**. Note that the

Level [m]/[ip1] box below remains grey and does not list any choices. This is because the time step number 260 is not recorded in the output files.

Change the Dynamics Time Step # to **270**, which was the final time step of the mesoscale simulations, and notice that the Level [m]/[ip1] box becomes active. The statistics will be computed using the results at this time step.

b) **Level [m]/[ip1]**: Select the **50/(12011)** Statistics Level in the list of available levels. This grid level will be used to interpolate the wind fields at the Statistics level.

3. **Meso Geophy**: Use the **Browse** button to select the mesoscale geophysical data file you generated with GenGeo (MyMesoGeo.fst).
4. **Wind Climate Table**: Use the **Browse** button to select the climate table used for the mesoscale simulation, **55\115_table.ef**, which is located at **[AnemoScope]\WindClimateDB_Annual\55\115_table.ef**.
5. Fill in the Options area as follows:
   a) **Statistics level**: Leave this as **50**. This is the height above ground level for which the statistics will be computed, in metres.
   b) **Max Wind Speed**: Do not change the default value of **25** (m/s). This is the wind speed above which all the values are considered to be in a single wind speed class.
   c) **Averaging length**: Do not change the default value of **1,** which means that some quantities will be smoothed using a local average over a 3×3 cell area.
   d) Leave the **Delete intermediate UV File** option checked in order to remove all intermediate output files.
6. **Output File (Statistics)**: Name your output file **MyMesoStats.fst** and place it in **MyTutorial\MyMeso**.
7. Click **RunWEStats** to launch the computations. Before the execution starts, specify the namelist file in which the WEStats input parameters will be stored. Save this file as **MyMesoStats.nml** in your **MyTutorial\MyMeso** folder.
8. A **Running WEStats…** window pops up in which information on the execution is displayed. Click **OK** when it finishes. This process can take as long as several minutes to complete.
9. When the computation is completed, close the WEStats Namelist File Editor dialog by clicking **OK**.

Step 8: Check the Mesoscale Wind Statistics in the Wind Statistics View.

When you've closed the Namelist File Editor dialog, use the Wind Statistics View to examine your results. The Wind Statistics View is specifically designed to allow you to examine any point on any field of the wind statistics file, while generating a wind rose and a wind speed histogram for the location.

1. Open a new **Wind Statistics View** by clicking the **New Wind Statistics View**

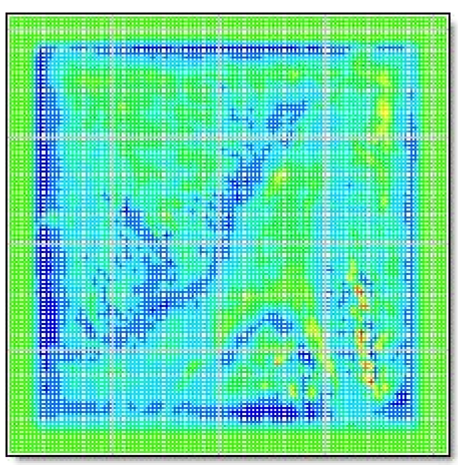
button on the AnemoScope tool bar. This view is specifically designed to visualize the wind statistics data.

2. Click the **Open File** button on the EnSim tool bar. Set **Files of type** to FST File (*.fst), and open your **MyMesoStats.fst** statistics file. Select all of the fields by clicking **All On**. Click **OK**.
3. Within the WorkSpace, drag the regular grid object **ER** (a child of MyMesoStats) into the Wind Statistics View. You should see a 2D map displayed on the right-hand side of the view. Double-click on any point on the map. This will cause a wind rose to appear on the left-hand side of the view. The variable ER contains the frequency distribution of wind direction by sector. By default, sector 1, centered around 0°, is active, so the frequency of wind in this sector is what's being shown on the map. You can select another sector by using the Data tab of the ER object's Properties dialog. Note that the wind rose shows information for all 12 sectors at once.
4. Within the WorkSpace, drag the regular grid object **UH** (also a child of MyMesoStats) into the Wind Statistics View. The new map is displayed on the right-hand side of the view. Double-click on any point on the map; in addition to the wind rose, a wind speed histogram will appear on the left-hand side of the view. The variable UH is the frequency distribution of wind speed by class. By default, class 1, 0 to 0.2 m/s, is active but you can select another class on the Data tab of the UH object's Properties dialog. The wind speed histogram show values for all 27 categories at once.
5. You can now drag and drop any other field from MyMesoStats into the Wind Statistics View. It will be displayed in the right-hand frame. Double-clicking anywhere on the map will display wind roses and wind speed histograms based on the ER and UH fields. Try it out by displaying the **EU** field (which shows the mean wind speed) and selecting a few locations.

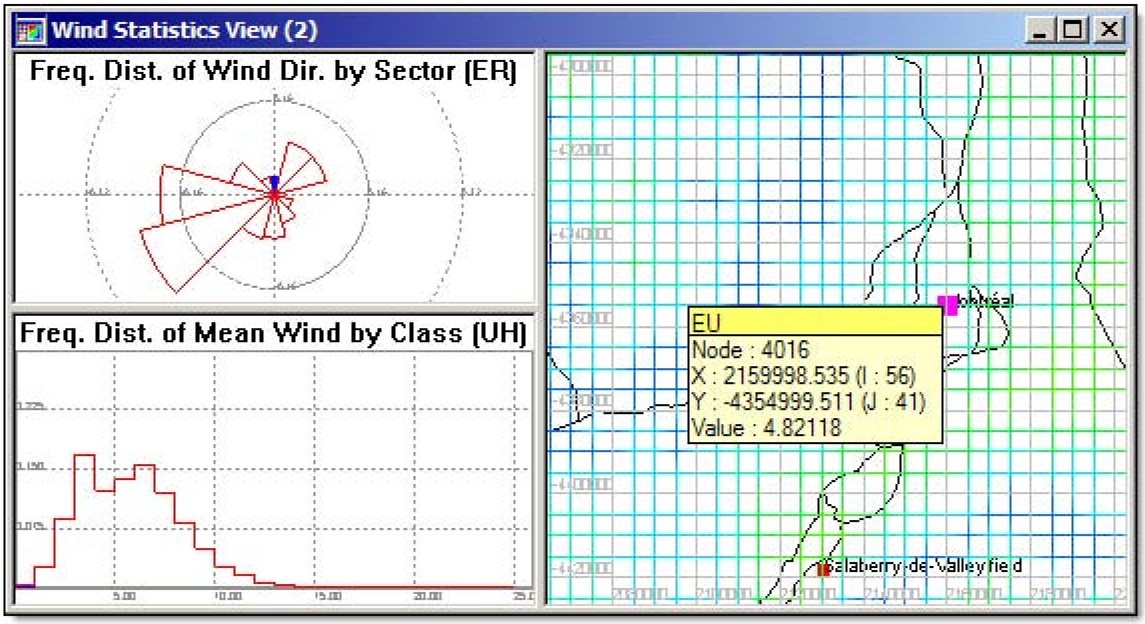
Validation of Your WEStats Results

Now we'll compare your results with the benchmark results.

1. First, compare your map of the EU field with the example below.



2. Then drag the **Cities** point set object to the Wind Statistics View. Zoom in and center the view on Montreal. Double-click on the grid point located just northeast of Montreal (Node 4116, located at I=57, J=42). The wind rose and the wind speed histogram should look like the following example (each frame of the Wind Statistics View has it own zooming capability):



3. For a quantitative comparison, load the reference WEStats output file **MesoStats.fst** located in **[TutorialDir]\WEStats** and compare them with the calculator, like you did with your GenGeo and MC2 results.

Part 2: MMC - Microscale Wind Statistics Production

Note: You should not try this part of the tutorial until you're familiar with the topics covered in Part 1.

When exploring the MMC module, we'll use the MsMicro microscale wind model to compute wind flow in the lower part of the atmospheric boundary layer, between 10 m and 150 m above ground. The calculation is done on a fine grid; here we'll use a grid with a resolution of 200 m. As was the case for the mesoscale model, the coordinate system used is Polar Stereographic, with a spherical ellipsoid.

The target microscale area is Montreal and its surroundings.

MsMicro needs three inputs, in addition to the parameter settings:

- **Wind data**: We get this from the mesoscale gridded statistics generated by WEStats on the basis of the MC2 results.
- **Topographic elevation.** This should be at approximately the same resolution as the microscale grid, about 200 m. In most cases, you'll need to supply this information, since the GenGeo database, although covering the entire surface of the Earth, is limited in resolution to about 1 km.
- **Detailed land cover.** This should also be at about the same resolution as the microscale grid. Like the topographical data, you'll need to supply this data.

For the Tutorial MMC exercise, we provide the wind data and a high resolution topographic elevation database, but not the detailed land cover database. So, for land cover, we'll use the GenGeo database, which isn't really accurate enough for the best possible results. This will cause minor patchwork patterns, with a 1-km footprint in some parts of the solution, as well as slight inaccuracies in the land-water boundaries that you'll be able to see in the microscale wind pattern obtained. With more precise data, these problems can be avoided.

The MMC tutorial data is organized as follows:

- **[TutorialDir]\InputData\Maps\Montreal\Water.mif** - contains a high resolution (1:250k) coastline for the Montreal area in MIF format, in Lat/Long coordinates with a Spherical datum.
- **[TutorialDir]\InputData\Dem\Montreal.arc** - contains a high resolution (1:250k) terrain model for the Montreal area, in ArcInfo ASCII grid format.
- **[TutorialDir]\MicroGrid** - contains settings and results of the microscale grid preparation process.
- **[TutorialDir]\MMC** - contains the reference settings and results for the microscale model.

Before you begin, use the Windows Explorer to create a new folder called **MyMicro** inside your **MyTutorial** directory. Remember, there should be no spaces in this folder names.

There are two more things that you'll have to do before you begin to create the microscale terrain map. Because the **GenGeoDB** only has a resolution of about 1 km, more precise geographical data must be used. For GenGeo to understand the data, it first has to be converted into FST format.

Importing High Resolution Topography

AnemoScope includes support for various DEM (Digital Elevation Map) data formats including ArcINFO ASCII Grid, CDED/DTED and Surfer. The high resolution DEM supplied with this tutorial is in ArcINFO ASCII Grid format.

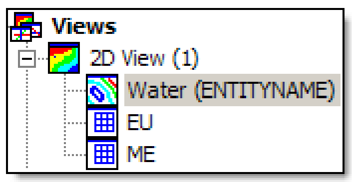
1. Using the Windows Explorer, create a new folder called **MyTopodb** inside your **MyTutorial\MyMicro** folder. Remember, there are no spaces in the folder name.
2. Launch AnemoScope from your desktop or from the Start menu, if it's not already running.
3. Let's take a look at our DEM data. From the menu bar, select **File→Import→ArcINFO Grid (ASCII)** to import the **Montreal.arc** file from **[TutorialDir]\ InputData\Dem**. The DEM characteristics are displayed in the pop up window:
   - d) ncols, nrows: 2402 3602
   - e) xllcorner, yllcorner: -75.000583 43.998936
   - f) cellsize: 0.000833 degrees.
4. This DEM was defined in the Lat/Long reference system, with a spherical ellipsoid but since this information is not stored in **ArcINFO ASCII Grid** files you must use the **Spatial** tab of its **Properties** dialog to **Assign** the Lat/Long coordinate system, with a spherical ellipsoid. Drag it to your 2D view.
5. **Export** the DEM to FST format by clicking the **Save as FST** button  on the AnemoScope tool bar. Save it as MyTOPODB_ME.fst under your MyTutorial\MyMicro\MyTopodb folder.
6. Once you've clicked **Save**, you'll be asked to provide a valid FST field name. Enter **ME** and click **OK**. GenGeo reads data from FST files, so this translation will make the grid usable as a topographical database.

   Note: Because this file is used only as a source of information, and it isn't altered by AnemoScope when it's accessed, it's not necessary to convert the file to the Polar Stereographic reference system.

High Resolution Coastlines

Next we will load a high resolution map for our area of interest.

1. From the menu, select **File→Import→MapInfo Interchange File** and select the file **Water.mif** from **[TutorialDir]\InputData\Maps\Montreal**.
2. This data is also defined with Lat/Long coordinates on a spherical ellipsoid so you must change the coordinate system to **Polar Stereo** (60,-100).
3. In the WorkSpace, drag the coastline object to the 2D view. Locate Montreal Island on the map.
4. Click the **Open File** button on the EnSim tool bar. Set **Files of type** to FST File (*.fst), select your MyTutorial\MyMeso folder, and open your **MyMesoStats.fst** mesoscale wind statistics file. Select the **EU** field only. Drag the EU object in the 2D View.
5. Open your **MyTOPODB_ME.fst** topographic file. Using the Spatial tab on its Property Dialog, change the projection to **Polar Stereographic** (60,-100). In the WorkSpace, drag the **ME** object to the 2D View.
6. Reorder the objects in the 2D View. Within the WorkSpace, drag the **EU** object and the **Water** objects to the 2D view so that they're listed in the following order:



You are now ready to perform the three steps of the process to compute microscale wind statistics.

Step 1: Define a High-resolution Grid to Support the Microscale Terrain Data

When you've finished this step, you will have created a second terrain grid with GenGeo, although at a considerably finer resolution than your mesoscale grid. This terrain grid will be used by MMC, along with your mesoscale wind statistics map, to create the microscale wind map.

1. We have to create a grid in Polar Stereographic coordinates that GenGeo can use as a target grid to prepare the MsMicro topography and Land Use data. To make best use of our DEM data we have to generate the largest grid that fits inside the DEM. In this case, the grid will have a resolution of 200 m. The ideal size of this grid is shown by the **MicroScaleGridPerimeter.i2s** file available in the **[TutorialDir]\InputData\Maps** folder. Open it and overlay it in the view. This is a native EnSim **i2s** file and is already in Polar Stereographic coordinates.
2. On the **Display** tab of the **Properties** dialog, increase the **Line Width** of the **MicroScaleGridPerimeter** object to **3**.
3. Using the **MicroScaleGridPerimeter** as a boundary, create a 200 m grid that fits to the nearest kilometre.

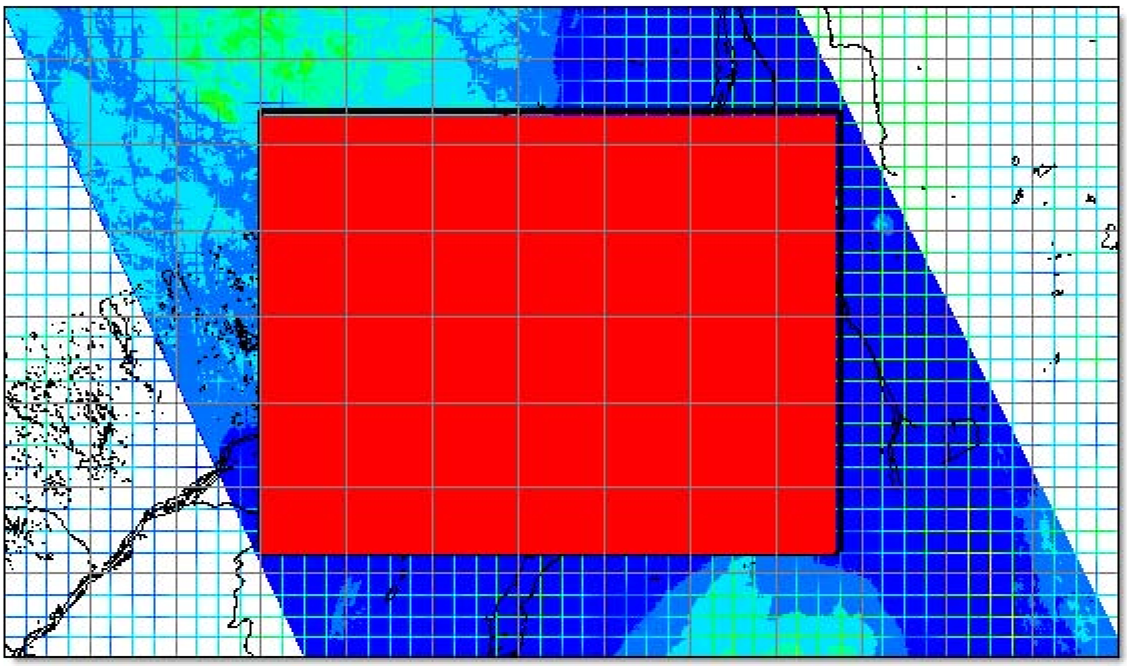
a) Click the **New Regular Grid** button on the AnemoScope tool bar. An empty Regular Grid object, with the icon , is added to the Data Items list, and its Properties dialog pops up. It opens on the **GridGen tab**, which is initially empty.

b) In the WorkSpace, drag the **MicroScaleGridPerimeter** line object and drop it onto the Regular Grid object. The GridGen tab is automatically filled. **Initialize** the grid.

c) Switch to the **Spatial tab** and **Assign** the Polar Stereographic (60,-100) coordinate system to this new grid. Go back to the GridGen tab.

d) In the WorkSpace, drag the new Regular Grid object to the 2D View. It fits within the **MicroScaleGridPerimeter** outline, but the Delta values are much too large. Set Delta X= 200 and Y= 200 and adjust the X and Y counts to Count X= 670 and Y= 510. Click **Initialize** to update the grid computations.

Note that with the new Delta and Count values, the grid is slightly smaller than the perimeter file along the top and right-hand edges.



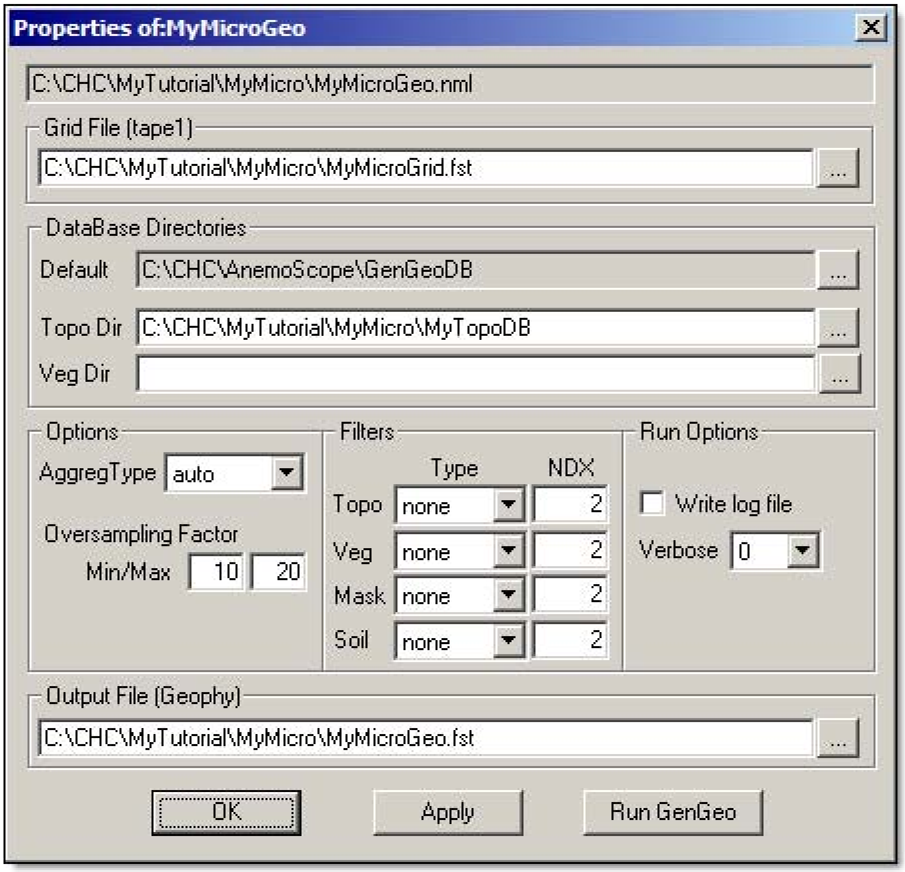
4. Use the **Save** button to save your grid as **MyMicroGrid.r2s** in your **MyTutorial\MyMicro** folder.

5. Click the **Save as FST File** button to export it as **MyMicroGrid.fst** in the same folder. Enter **GRID** when you're asked for the FST field name.

You now have an FST file that covers the entire area that will be examined on the microscale level, at a fine resolution.

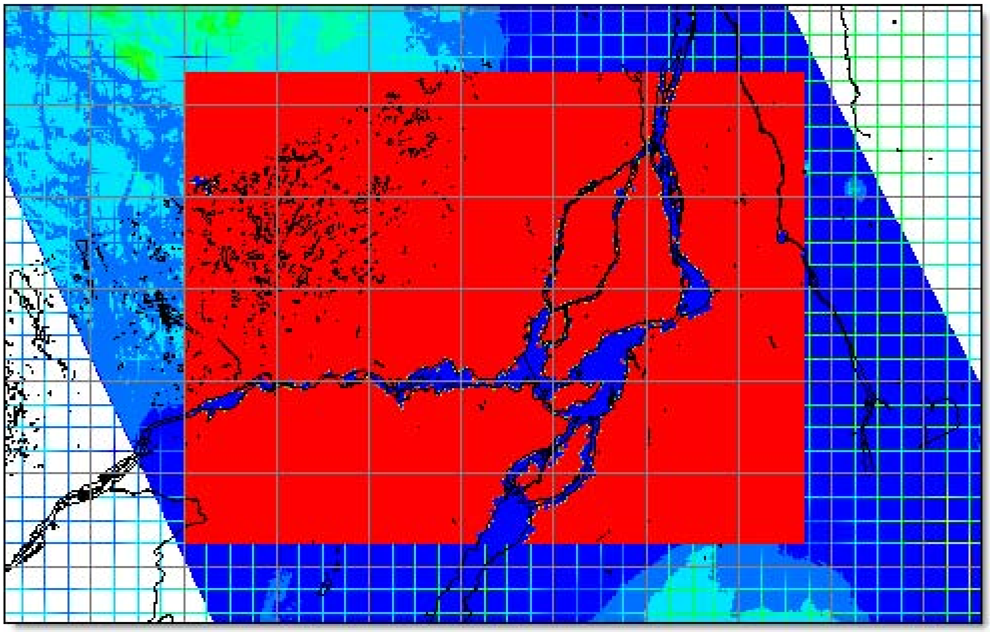
Step 2: Generate the High-resolution Geophysical Terrain Data for the Microscale Grid.

1. On the AnemoScope tool bar, click the **New GenGeo Parameters** button to create a new GenGeo parameter file object.



2. **Grid File (tape1)**: select your **MyMicroGrid.fst** microscale grid file.
3. Fill in the **DataBase Directories** parameters.
   a) **Default**: select the **[GenGeoDB]** directory, which was previously extracted from Gengeo.zip.
   b) **Topo Dir**: select the **MyTutorial\MyMicro\MyTopodb** folder that contains the **MyTOPODB_ME.fst** high resolution topographic database you created earlier.
   c) **Veg Dir**: leave this entry blank. GenGeo will use the land cover information from the default database.
4. **Options and Filters**: do not modify these values. Keep the default parameters.
5. Check the **Write log file** box and keep the verbose value at **0** (least feedback). Click **Apply** to confirm your settings.
6. **Output File (Geophy)**: save your output file in **MyTutorial\MyMicro** and name it **MyMicroGEO.fst**

7. Click **Run GenGeo**. When you're prompted, call the namelist file **MyMicroGEO.nml**.
8. When the module is finished, which should take a few minutes, open your **MyMicroGEO.fst** file and select the **ME** and **MG** fields. Drag **MG** into the view.
9. Move the coastline **Water** to the top of the 2D View list. Compare your results with the following:



10. The DEM object **MyTOPODB_ME** is no longer necessary. Open its **shortcut menu** and **Remove** it from the WorkSpace.

Save Your WorkSpace

Before you go on, save your WorkSpace by selecting **File→Save WorkSpace…** from the menu bar. Save your WorkSpace as **MyWorkSpace2** in your MyTutorial\MyMicro folder.

Step 3: Set Up and Perform the Microscale Simulations and Compute the Microscale Wind Statistics Data.

1. In the WorkSpace, drag **EU** to the top of the 2D View. To make it easier to see on top of the other maps, make **EU** a black monochrome wireframe. Check **Monochrome** in the **Display tab** of the **EU** object's **Properties** dialog and select black from the available colours.
2. Click the **New MMC Parameters** button on the AnemoScope tool bar. A newMMC object appears in the WorkSpace, and the MMC Namelist File Editor dialog opens.
3. The two **Input Files** are **MyMicroGEO.fst** (MicroGeophy) and the

MyMesoStats.fst (MesoStats). Select them for your MyMicro and MyMeso options, respectively. The MesoDX and Stats Elevation boxes are automatically filled with the data found in the mesoscale statistics file.

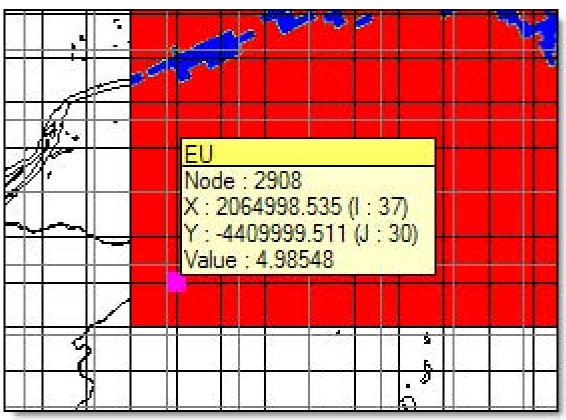
4. In the **Sweep Options**, keep the **Stride** parameter at its default value (1). We'll come back to the (I1,J1) and (I2,J2) entries in a moment.

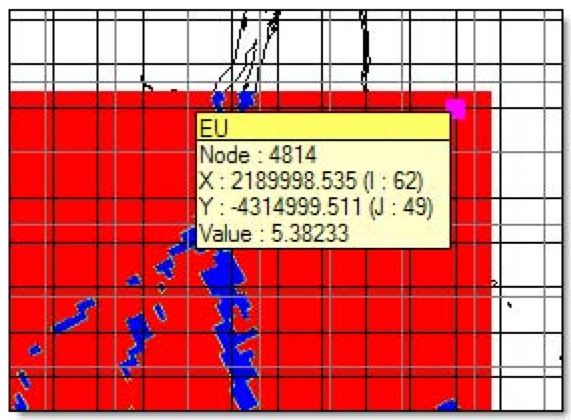
5. Start setting the **MSMicro Grid** options.

   a) Keep the default **Tile Size** (or **nu**) value at 128. This value defines the number of grid points used for each microscale tile.

   b) Increase the **Tile Overlap** value to **0.6**. Click **Apply**. Notice that the **Micro DX**, the resolution of the microscale grid, is modified. The **Micro DX** should now be 192.308 m, which is fairly close to the 200 m objective.

6. Returning to the Sweep Options, the next things to determine are the sweep indices: I1, I2, J1, and J2. These options determine the area of the mesoscale grid on which MsMicro will center its individual microgrid domains. The rule here is to keep a "microframe border" around the chosen I and J values to accommodate the outermost microgrid domains.





This microframe border is **nu**×**0.8**×**MicroDx** wide on each side, which in this case is 128×0.8×192.308 = 19,692 m. This is 3.94 times the resolution of the Mesoscale grid, as shown in the **Meso DX** box. So, we need to keep a 4 Meso Delta microframe border around the (I1, J1)/(I2, J2) domain.

Within the view window, click on the mesoscale grid node closest to the lower left corner of the microscale geophysical data, but still within the microscale grid, to get the mesoscale node indices. Do the same for the upper right corner. We find that the corners are at (37, 30) and (62,49) respectively.

So, the smallest choice possible for (I1,J1) is

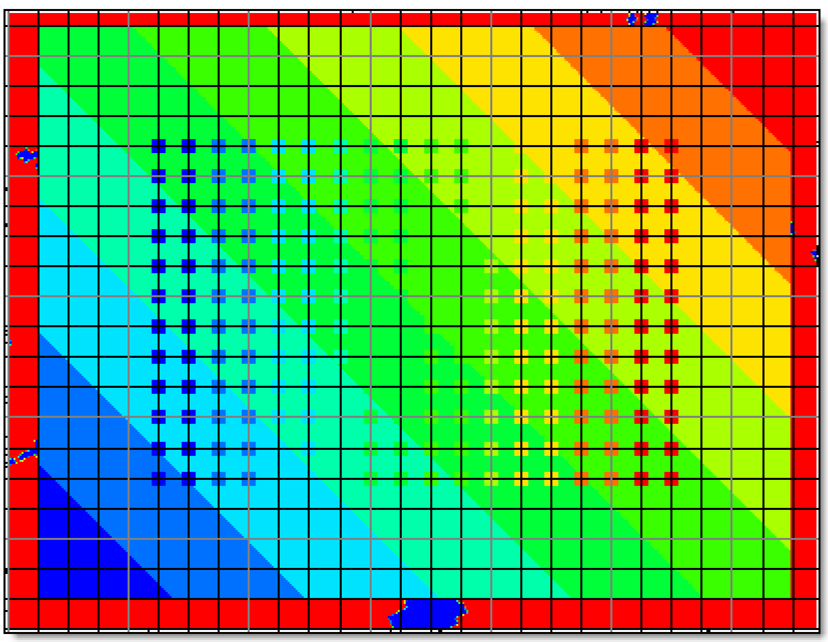
I1 = 37 + 4 = **41** and J1 = 30 + 4 = **34**

and the largest choice possible for (I2,J2) is

I2 = 62 – 4 = **58** and J2 = 49 – 4 = **45**.

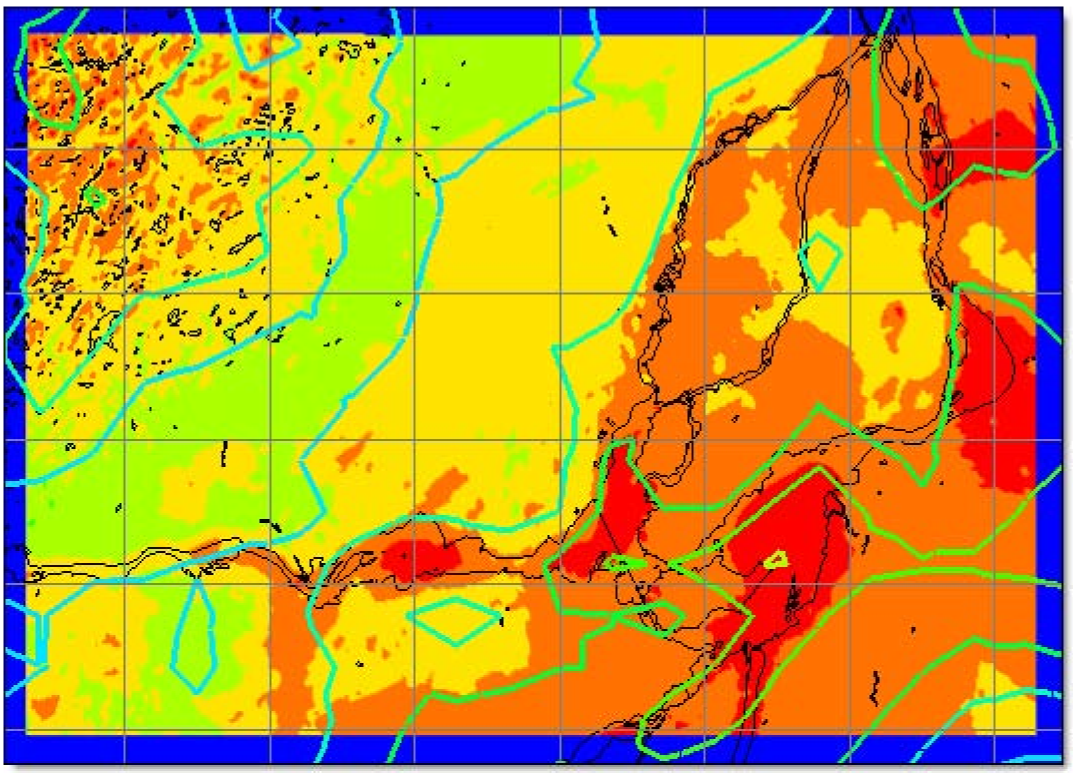
Enter the final values: (41, 34) and (58, 45).

7. Leave the **Output Variables** (**'EUMI', 'E1MI'**) and **Output Elevation** (**50**) as their default values. With these settings, MsMicro will run at 50 m above ground and produce all the available microscale statistics, such as EUMI (mean wind speed) and E1MI (mean wind power).
8. Using the Windows Explorer, create a new folder called **TILES** in your **MyTutorial\MyMicro** directory. The microscale model outputs will be written there.
9. For the **MS Micro Output Tiles Directory** parameter, use the **Browse** button to select the **TILES** directory.
10. The **Output File (Merged Results)** entry should be named **MyMicroResults.fst**, located in your **MyTutorial\MyMicro** directory.
11. **Uncheck** the **Run MsMicro** box. With this option turned off, MsMicro won't be run by MMC, so we'll be able to double-check the micrograid area before we run the model. Click **Apply** to confirm the new settings.
12. Click **RunMMC**. Reply **OK** to the question. The **TILES** directory is empty at the moment, so there's no data to be lost. Save the MMC Name List File as **MyMMC.nml** in your **MyMicro** folder.
13. Let's check our layout before running MsMicro. Once the first execution is done, open the **microGrid.fst** file (it's located in the **TILES\MMC_WORKING_DIR** directory). Load the **ZERO** and **TI_I** records.
14. Drag **ZERO** to the 2D View. **ZERO** must fit entirely within the MyMicroGEO MG domain.
15. Since **TI_I** is a point set object defined in Lat/Long coordinates, change it to Polar Stereographic (60,-100) coordinates with the **Apply** button. Drag it to the same view. The **TI_I** points fall on the I1,I2 x J1,J2 mesoscale nodes. The grid settings



look satisfactory.

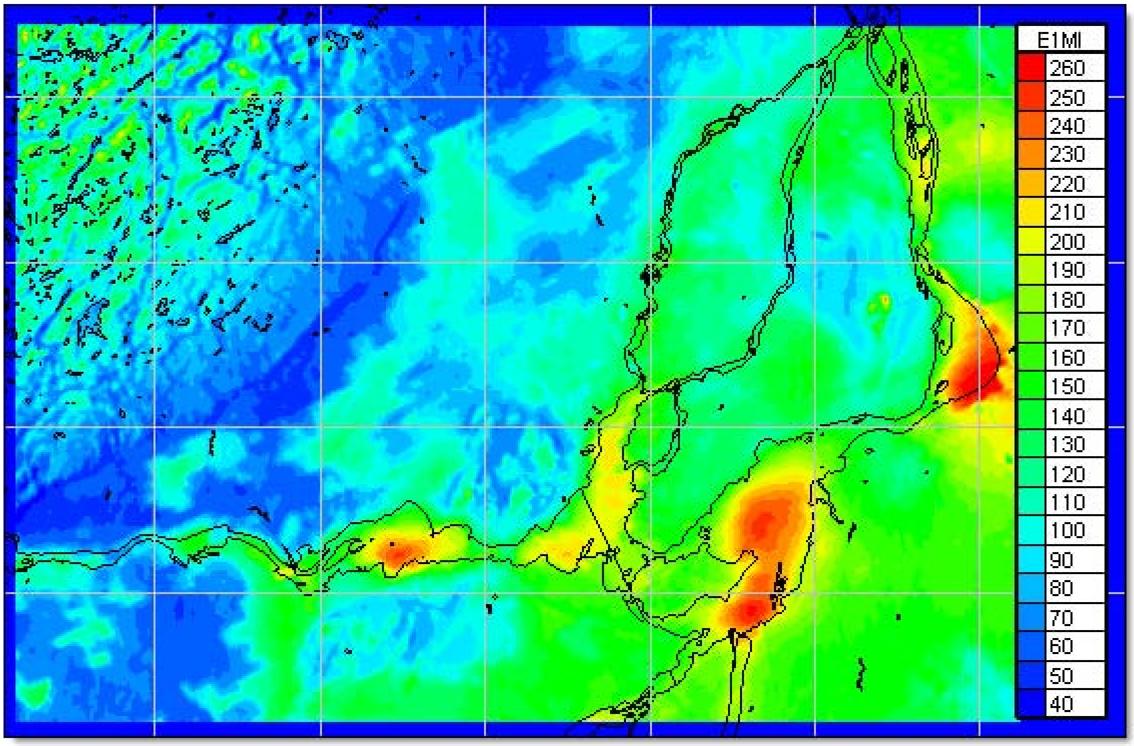
16. The only thing left now is to run MsMicro. Check the **Run MSMicro** box in the MMC dialog. This enables various fields on the dialog.
17. Set the **Output File (Merged Results)** to **MyMicroResults.fst**, located in your **MyTutorial\MyMicro** directory.
18. Click **Run MMC** again. The **TILES** directory will be overwritten. Answer **OK** to this question again.
19. The execution will take between 5 and 10 minutes. Once it's done, open the output file **MyTutorial\MyMicro\MyMicroResults.fst** and load **EUMI** and **E1MI**. Drag **EUMI** to the 2D View. You can remove the **TI_I**, **ZERO** and **MyMicroGrid** objects from the View, if you'd like, but leave **EU** there.
20. Change the **Display** parameters of **EU** in its **Properties** dialog. Select **Isolines** as the **Rendering Style**, uncheck **Monochrome** and increase the **Line Width** to 3.



21. Drag **EU** to the top of the 2D View. Drag the **Water** object above the **EUMI** object as well. Display your result as a surface (using the **Display** tab of the **Properties** dialog) and compare it with the screenshot shown above.

    The visual result can be improved by changing iso-levels on the **Properties Dialog (ColourScale tab)** of the **EU** and **EUMI** objects. Try adjusting the settings until all of the information is clearly visible.
22. Drag the **E1MI** object to a 2D View. The map should look approximately as shown below, depending on your ColorScale selections. The image below is shown with

its legend displayed.



Validation of Your MMC Results

By now, you should be able to compare your results, settings file, and log files with the reference files provided in **[TutorialDir]\MMC**.

If any of the results from MMC don't appear to match the examples, double-check your parameters with the examples shown. If that doesn't clear up the problem, take a look at your WEStats results and your microscale terrain grid.

Congratulations!

You've completed the AnemoScope tutorial. If you've followed all of the steps, you should now know how to scale down the supplied data to create a microscale wind map for any region in the world.