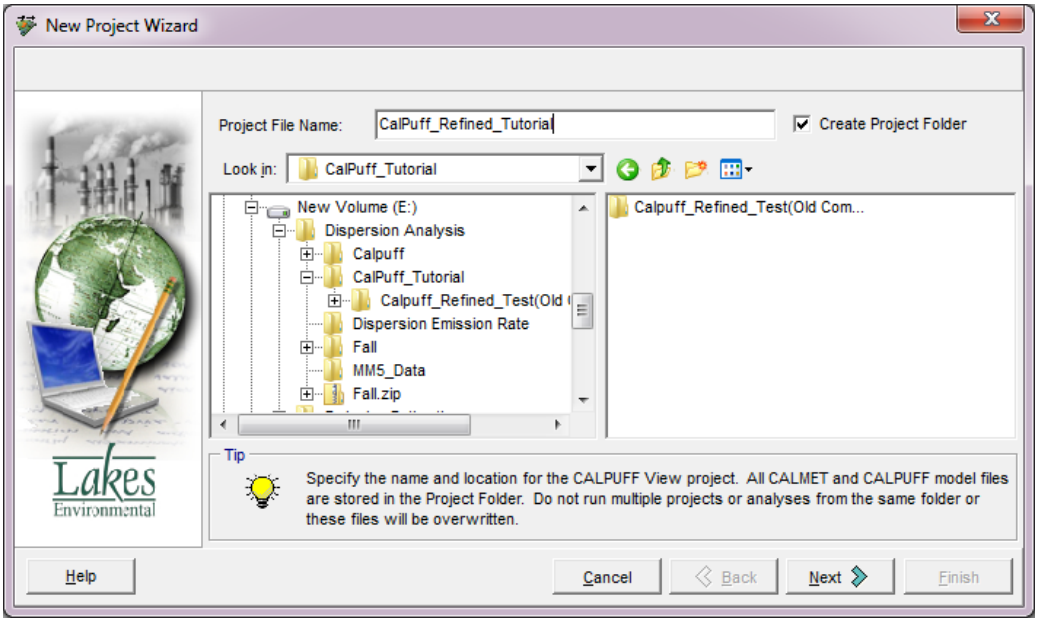
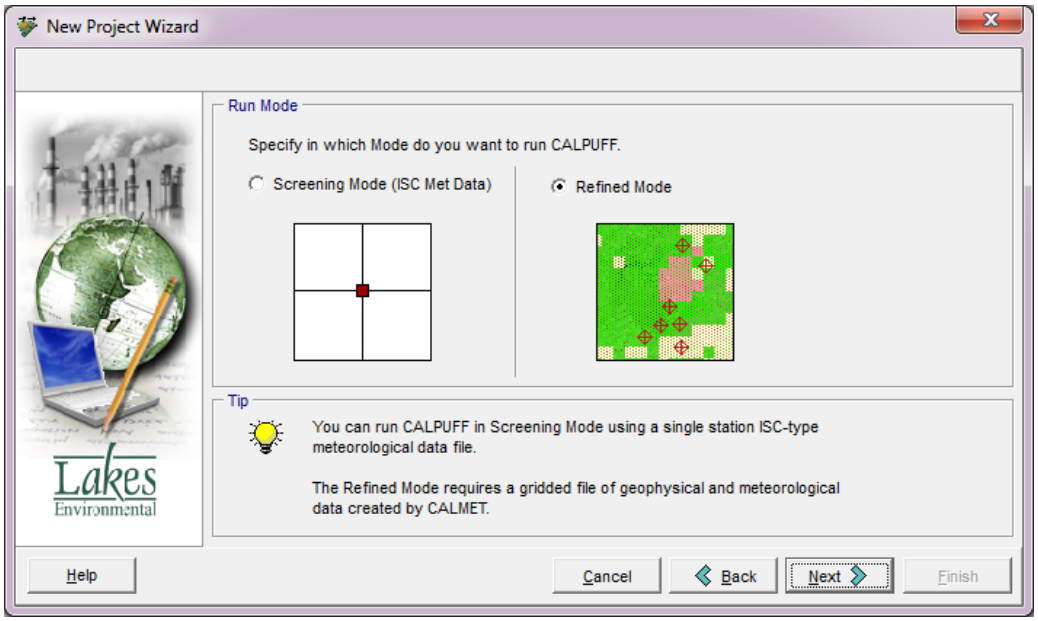
CALPUFF View 7.0.5 Tutorial

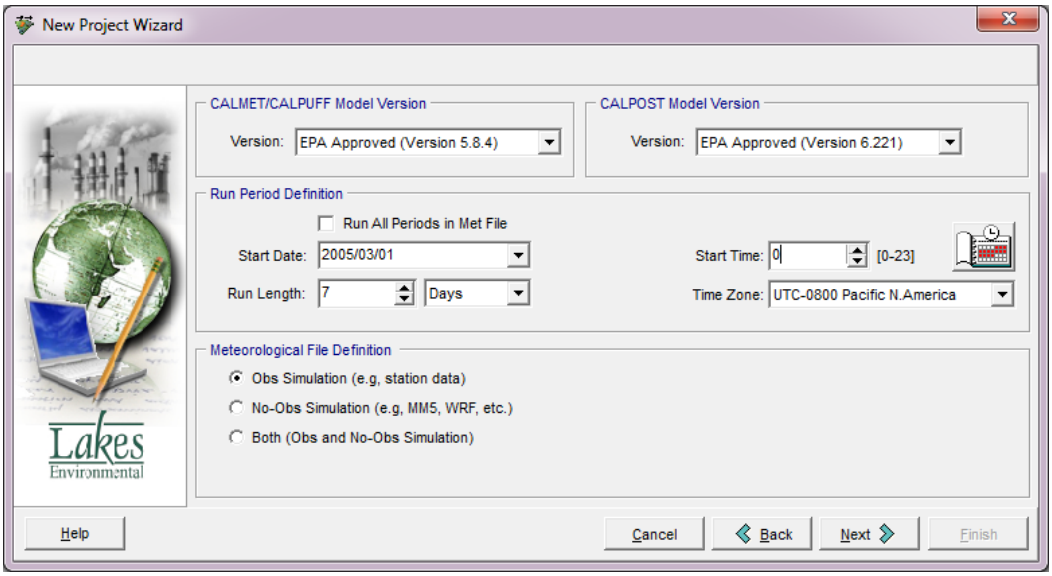
1. Start CALPUFF view application
2. Select **File | New Project** from the menu or click on the **New** menu toolbar button. The **New Project Wizard** is displayed.
3. Specify the project name and location according to your needs. Make sure the **Create Project Folder** box option is checked.



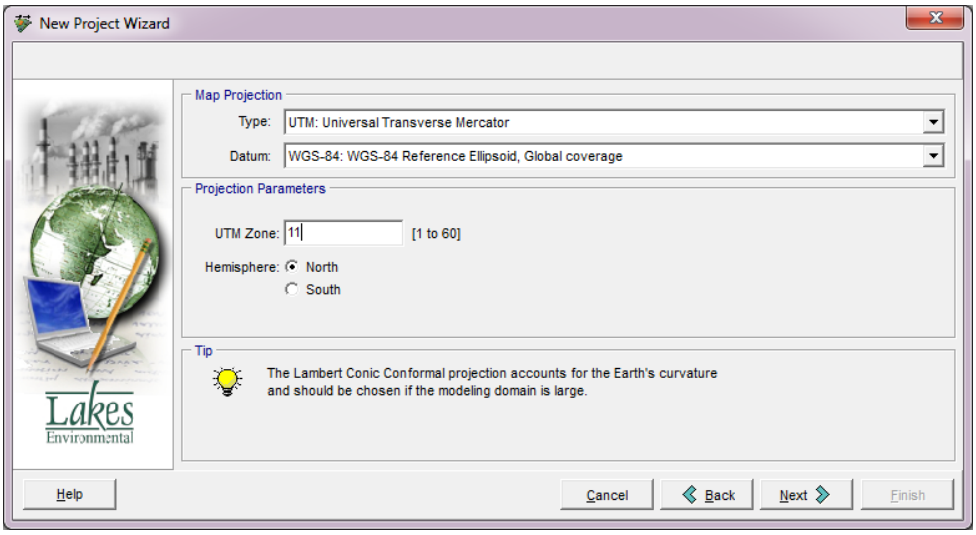
4. Click **Next**. In the next panel select the **Refined Mode** and press **Next**.



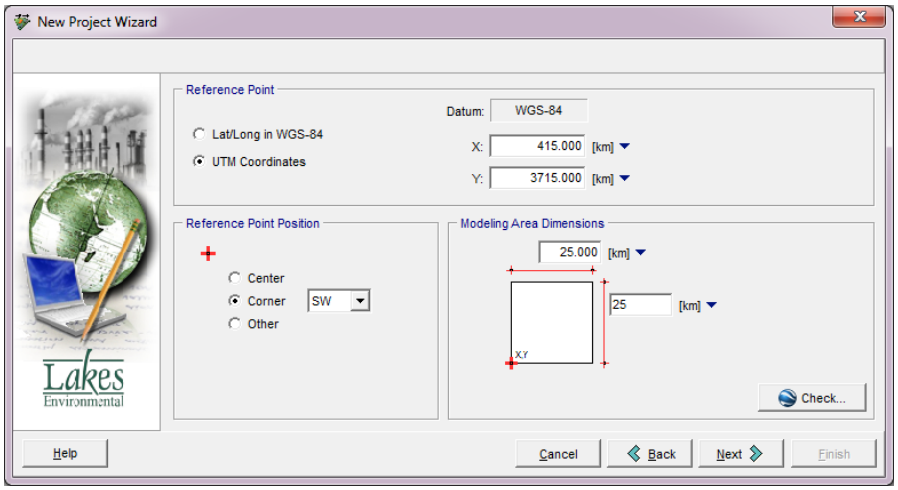
5. Enter **Start Date**, **Run Length**, **Time Zone (= Pacific), Model Version: EPA Approved (Version 5.8.4); Version 6 also available.**



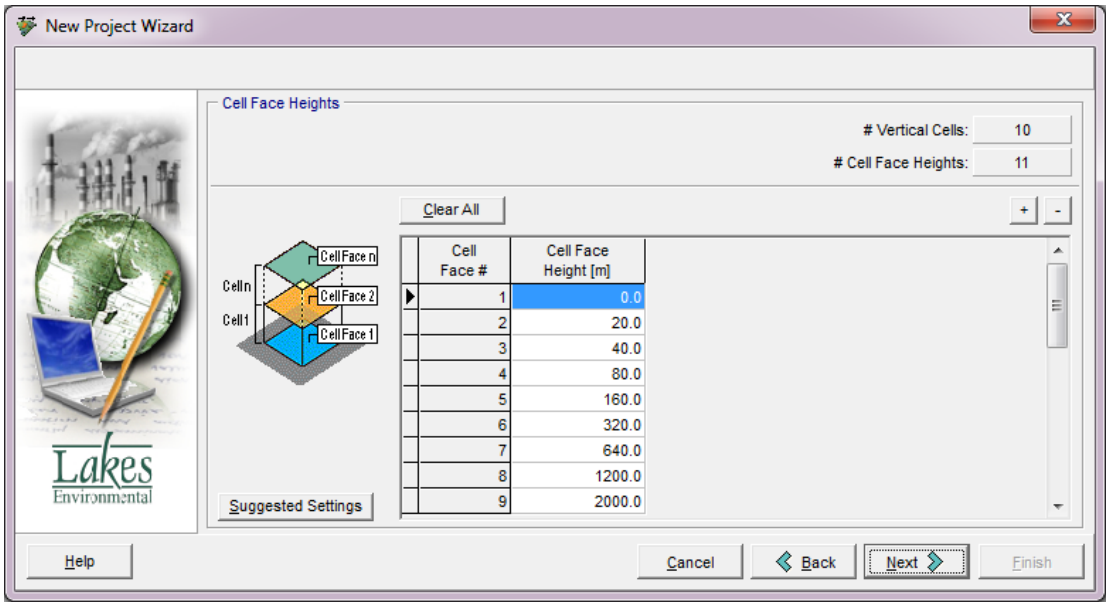
6. Enter **UTM Zone**: **11** for Southern California, click on **Help** for codes for other Zone.



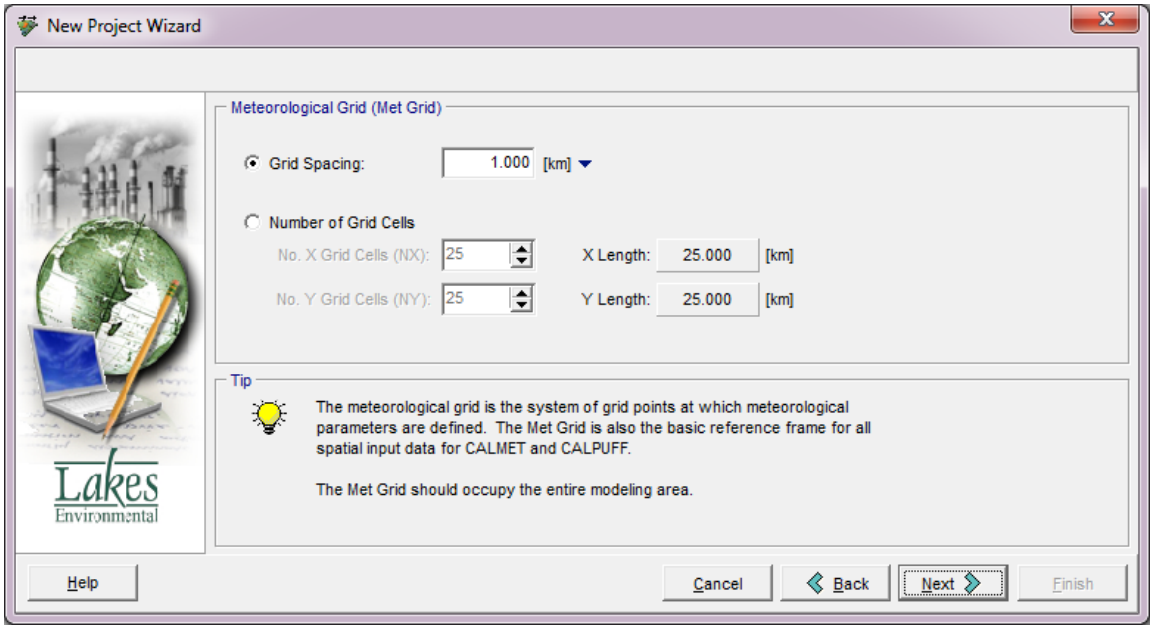
7. Enter **X** and **Y** for modeling location (I did some trial and error to find X and Y using Google earth **Check**), Select **Corner SW** and enter **Modeling Area Dimensions**



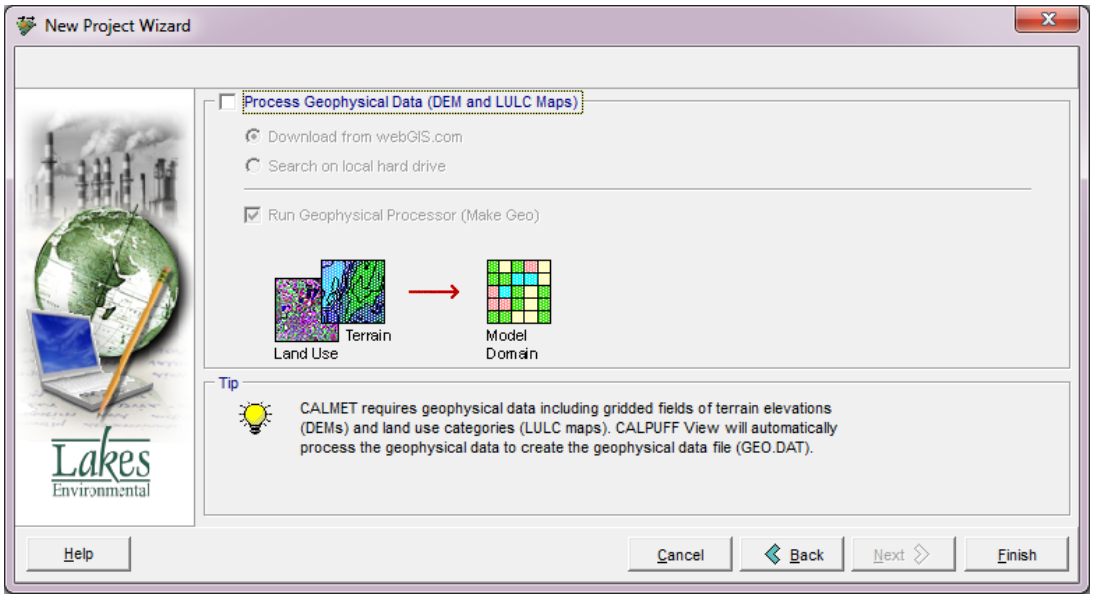
Cell Face Heights: Default



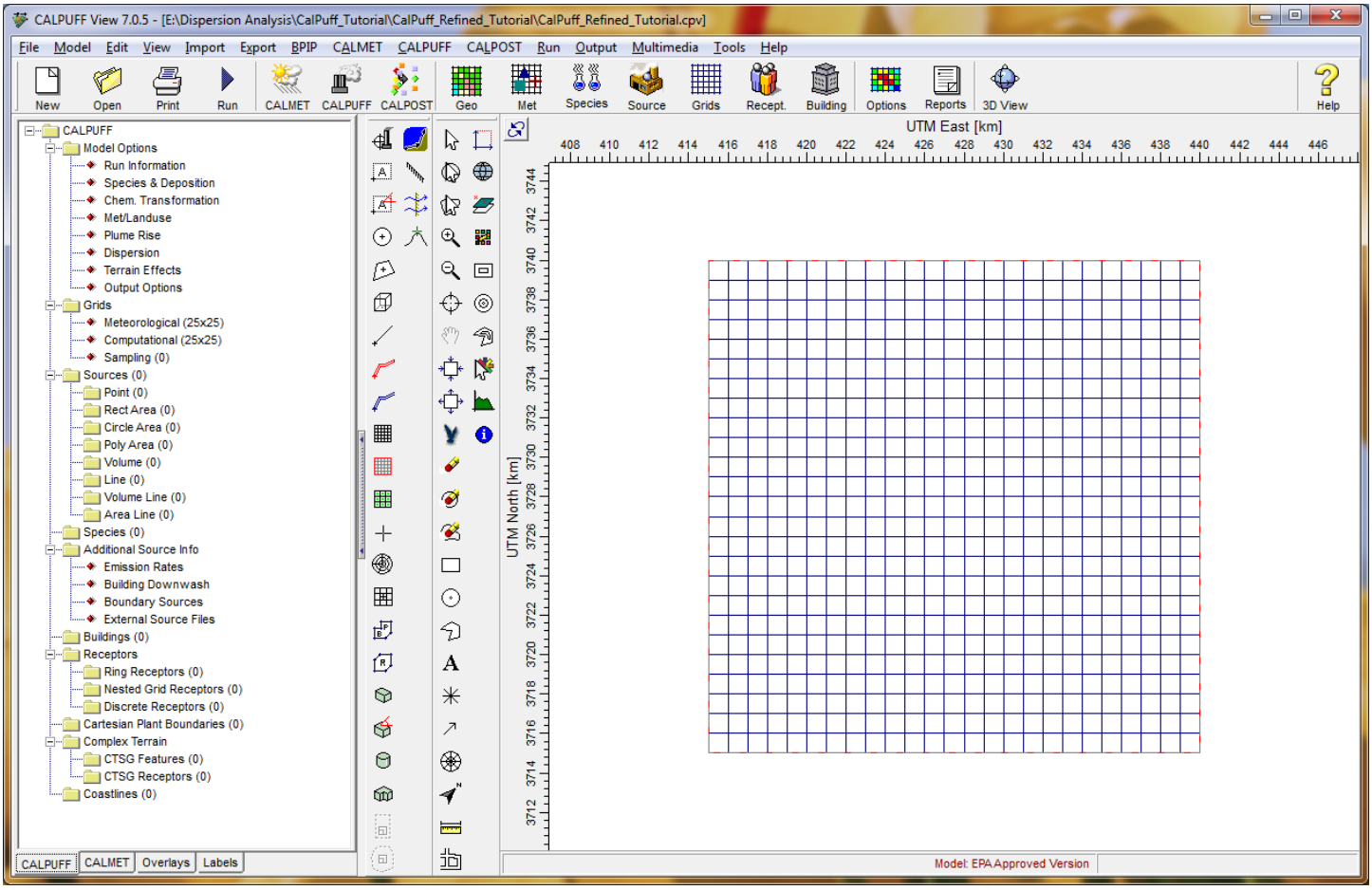
8. Enter **Grid Spacing** as per project's needs.



9. Click **Next**. In the next panel check box for **Process Geophysical Data** (Terrain, Land Use data) and Click **Finish.** **Check Box => If checked, Calpuff View automatically downloads all necessary geophysical data. In this tutorial, box will not be checked.**

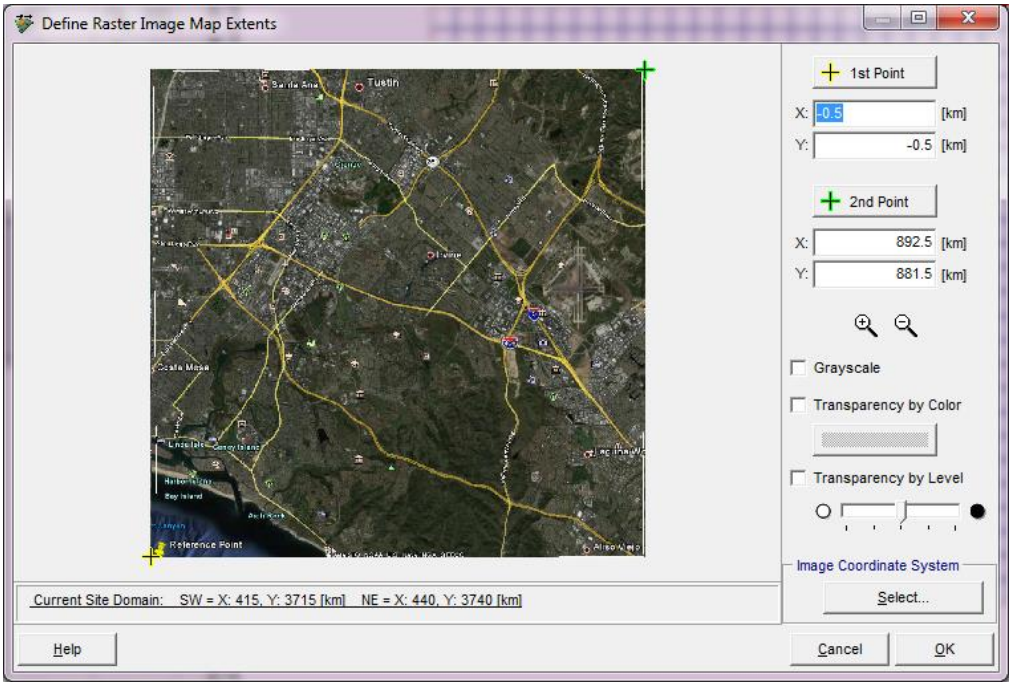


10. Now that you have completed all the information required under the **New Project Wizard** you will see your CALPUFF View project open with the main graphical area displaying the following information:
    - Meteorological Grid (25 x 25 grid cells)
    - Computational Grid (marked by a red box)
    - Green Box?

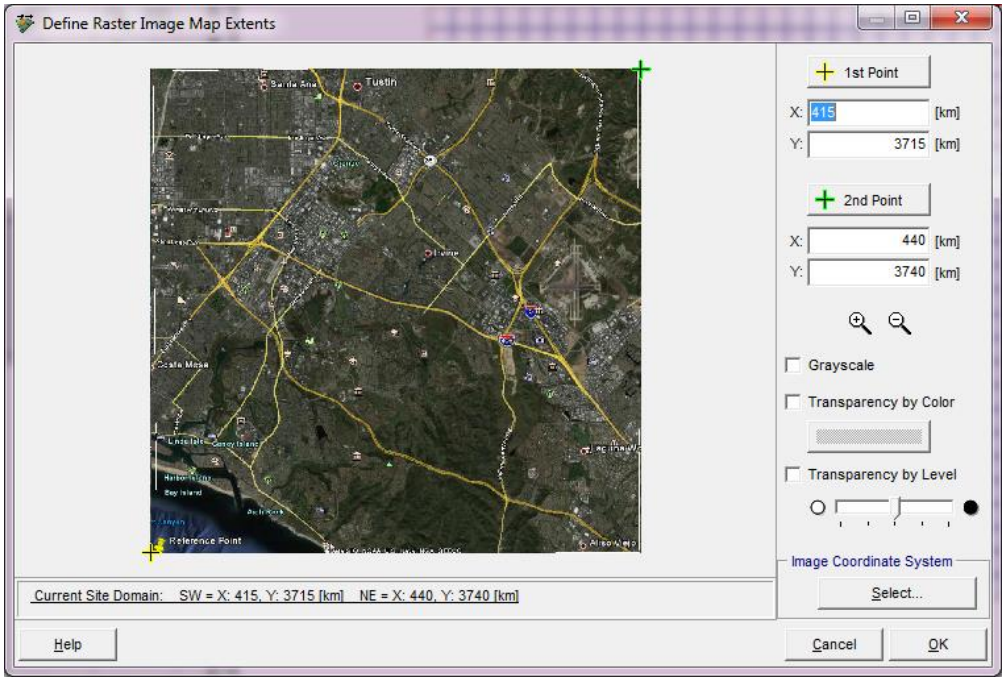


Importing Base Map

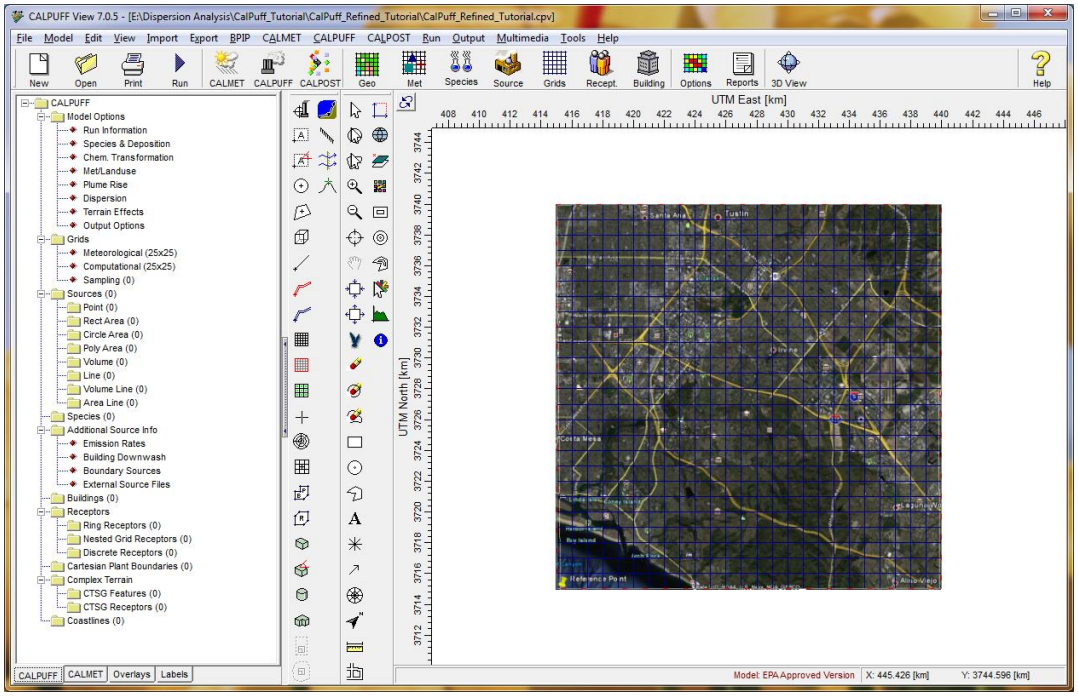
11. Select **Import | Base Maps... | Raster Images...** from the menu and specify the location of the map file to be imported. To specify **1st Point** and **2nd Point** Click on **Current Site Domain** on bottom of window



Then click on **Set To Domain Extents.**

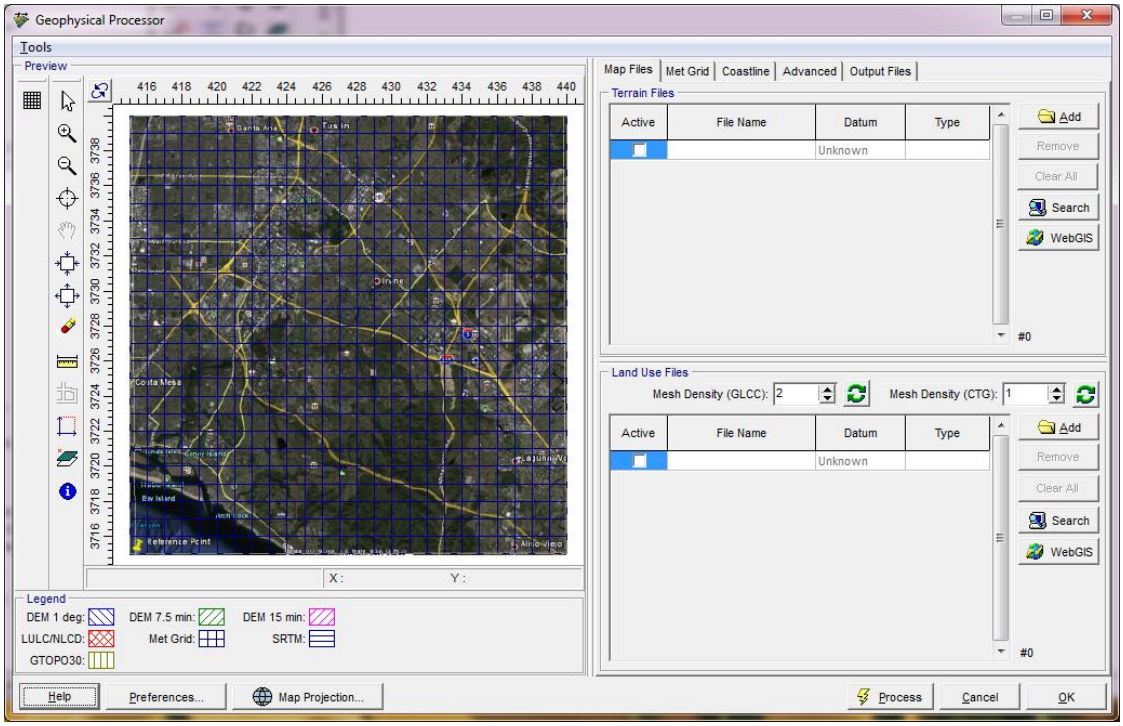


Then click **OK.** Result is below:

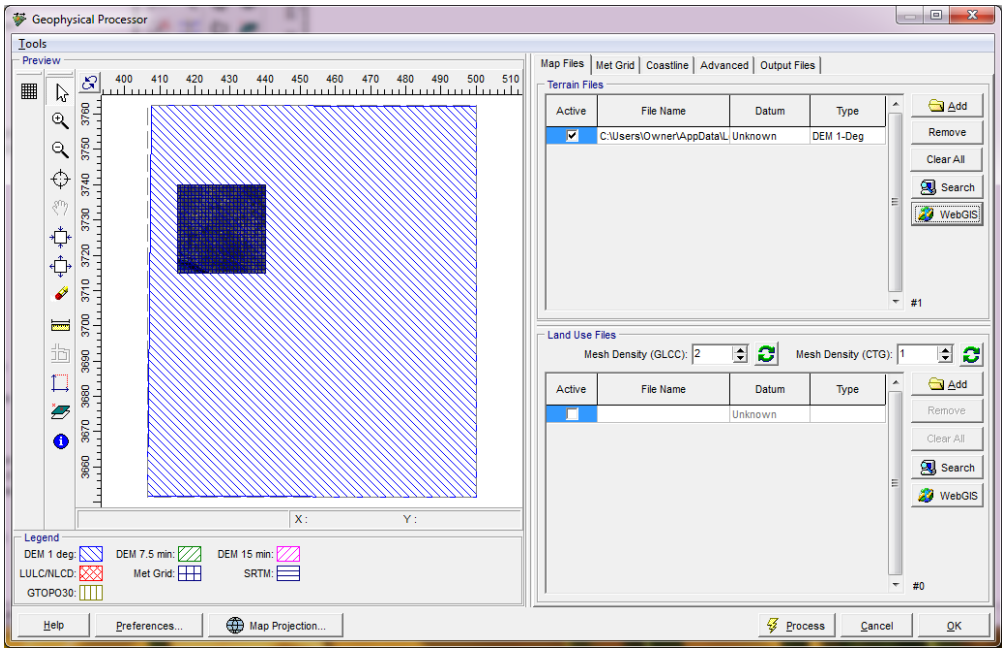


Geophysical Processing

12. Click the **Geo** button located on the menu toolbar to open the **Geophysical Processor** window. The **Geophysical Processor** is where you specify the terrain and land use (LULC) digital data files necessary for the geophysical preprocessing of data required by the CALMET model.



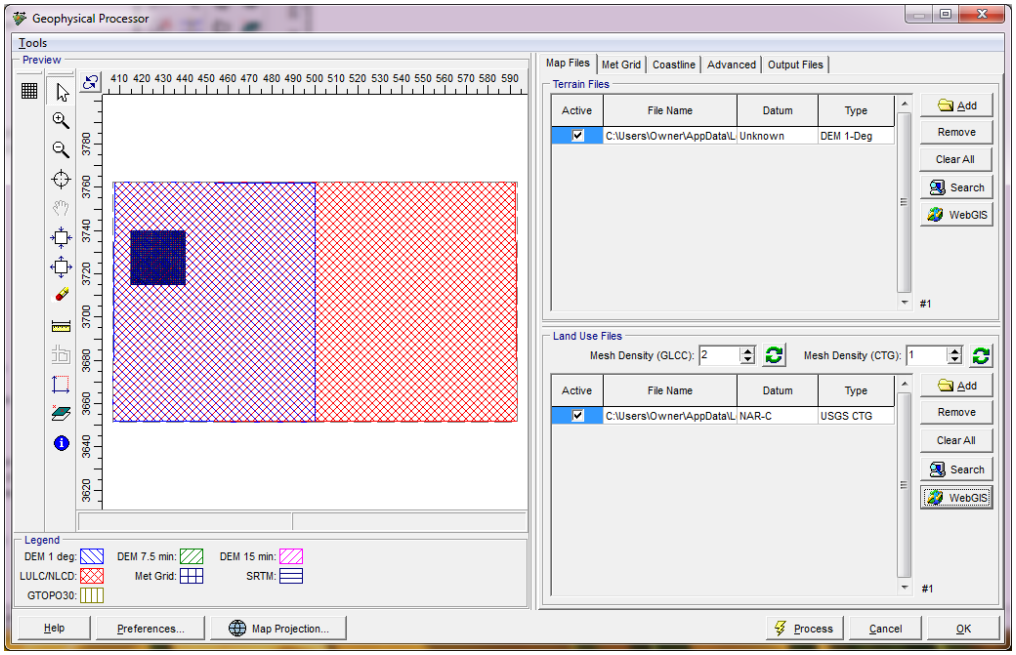
Under the **Terrain Files** section, click on the **WebGIS** button. A pop-up menu is displayed. Select the option **Load DEM 1-Deg.** Terrain data files covering your modeling domain will be automatically downloaded from WebGIS and displayed under the Terrain Files table and Preview area.



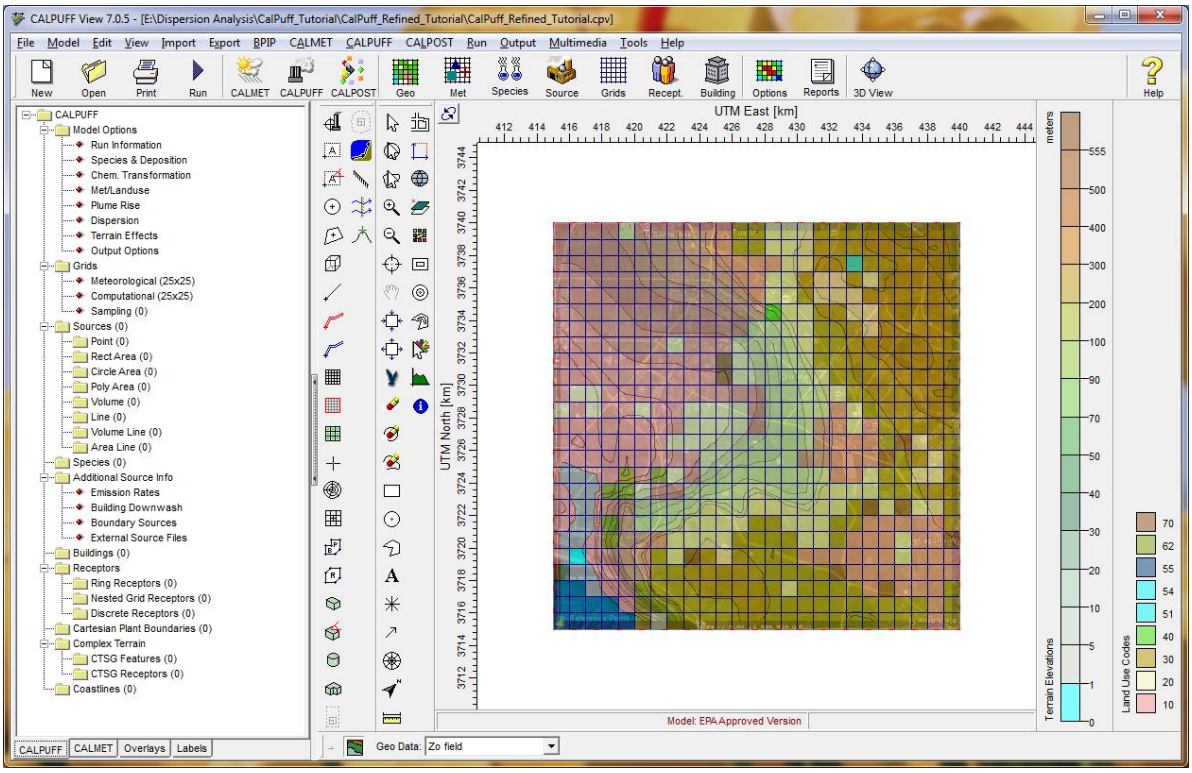
Under the Land Use **Files** section, click on the **WebGIS** button. A pop-up menu is displayed. Select the option **Load USGC CTG (US 200m).** Land use data files covering your modeling domain will be automatically downloaded from WebGIS and displayed under the **Land Use Files** table and **Preview** area.

Still in the **Geophysical Processor**, press the several tabs available in this window to learn more about its contents:

- **Map Files tab:** This tab is where you specify the terrain and land use data files necessary for the geophysical processing of data required by the CALMET model.
- **Met Grid tab:** This tab contains the Meteorological Grid (Met Grid) parameters that you initially set up within the **New Project Wizard**.
- **Coastline tab:** This tab contains coastline processing options.
- **Advanced tab:** Seasonal terrain parameters are available in this tab. You can change the geophysical values (Albedo, Bowen Ratio, Leaf Area Index, Soil Heat Flux, Anthropogenic Heat Flux, and Surface Roughness) associated with each land use category These land use properties are classified using the USGS land classification system.
- **Output Files tab:** In this tab you can see all of the input/output files generated after the geophysical data was processed. After processing, if you are interested in seeing these files, double click on any of them to open the file in a text editor. These files are stored in your project folder in a folder called **ProjectName_geo**.



13. You are now ready to process your geophysical data. Press the **Process** button located at the bottom of the **Geophysical Processor** window. The several preprocessors (e.g., TERREL, CTGPROC, MAKEGEO, etc.) are run in sequence and without the need of any further input from you. Once the process finishes, you will see the processed geophysical data displayed under the CALPUFF View main window (e.g., terrain contours, land use categories, etc.).

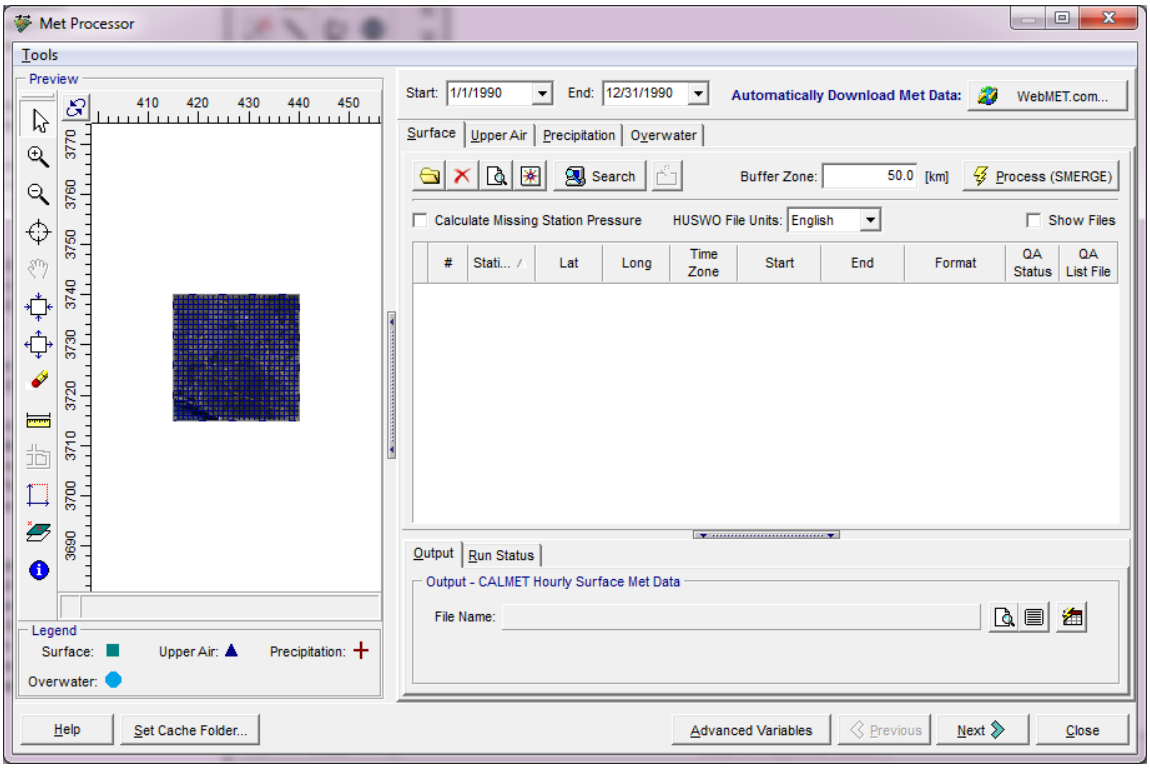


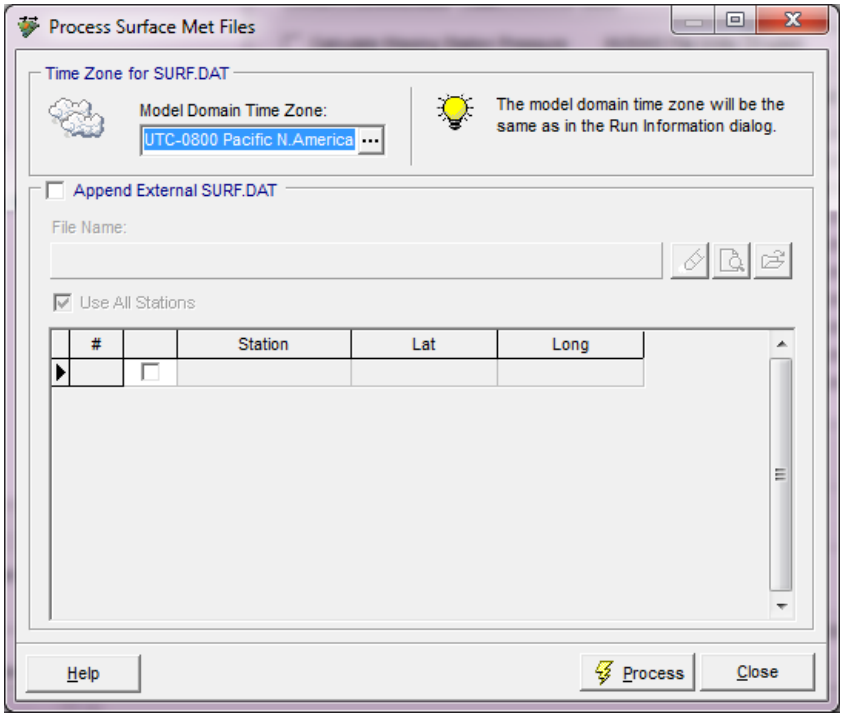
Meteorological Processor

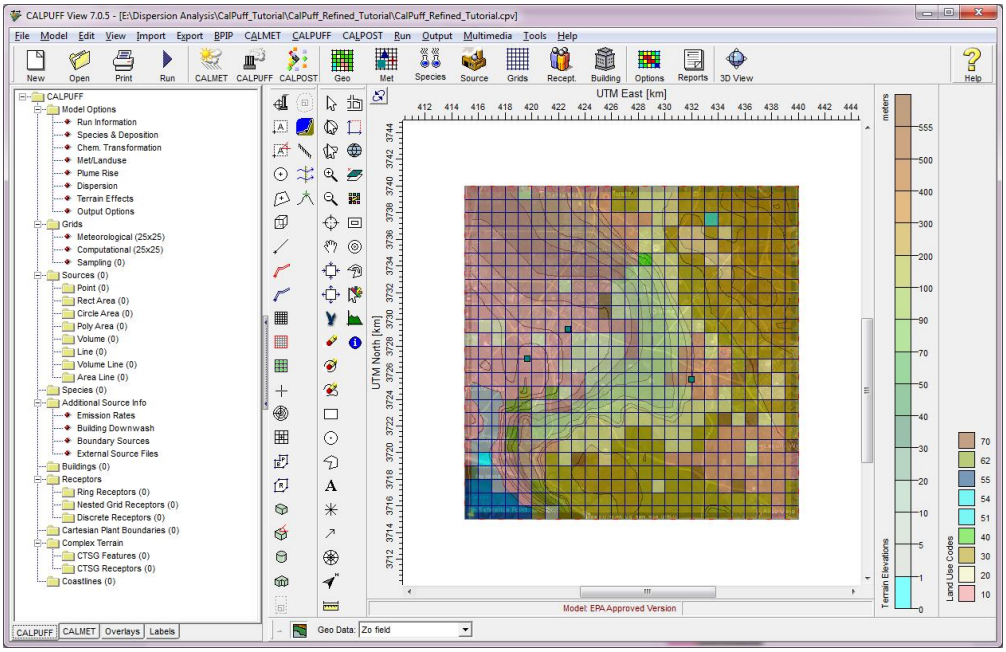
14. Click the **Met** button located on the menu toolbar to open the **Met Processor** window.

Surface Data (Green Square Box)

15. From the **Met** Processor window - **Surface tab**, specify the following parameters: Buffer = 50 km. Download Met data and click on **Process (SMERGE)**

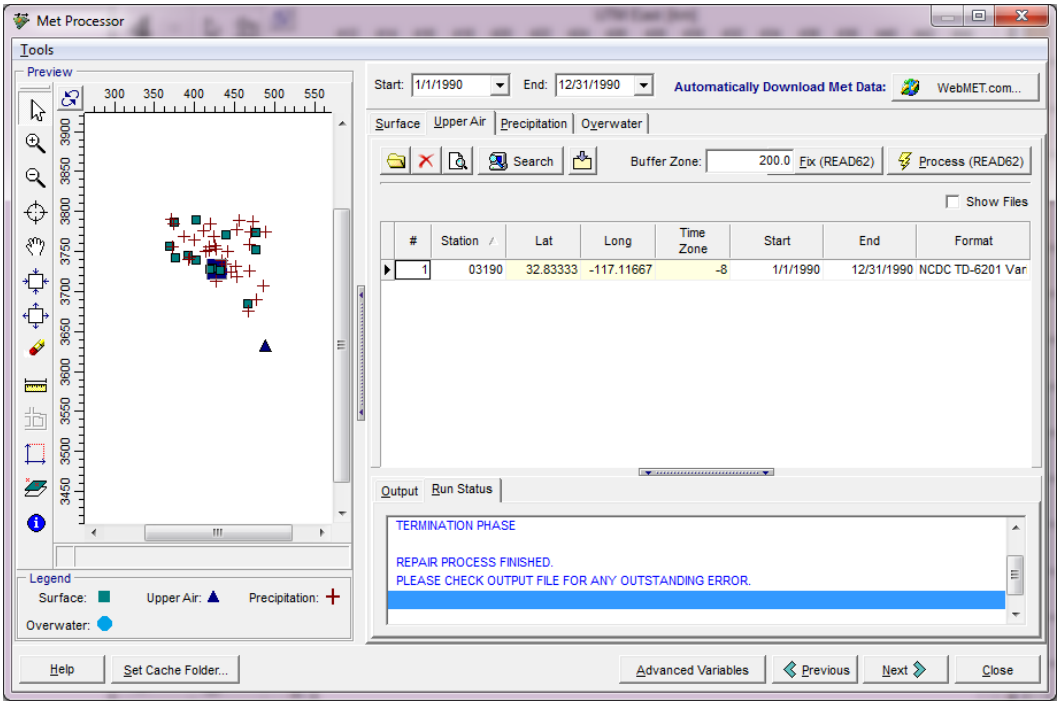

Met Processor
Tools
Preview
410
420
430
440
450
3770
3760
3750
3740
3730
3720
3710
3700
3690
Legend
Surface:
Upper Air:
Precipitation:
Overwater:
Start: 1/1/1990
End: 12/31/1990
Automatically Download Met Data:
WebMET.com...
Surface
Upper Air
Precipitation
Overwater
Search
Buffer Zone: 50.0 [km]
Process (SMERGE)
Calculate Missing Station Pressure
HUSWO File Units: English
Show Files
Stati... Lat Long Time Zone Start End Format QA Status QA List File
Output
Run Status
Output - CALMET Hourly Surface Met Data
File Name:
Help
Set Cache Folder...
Advanced Variables
Previous
Next
Close

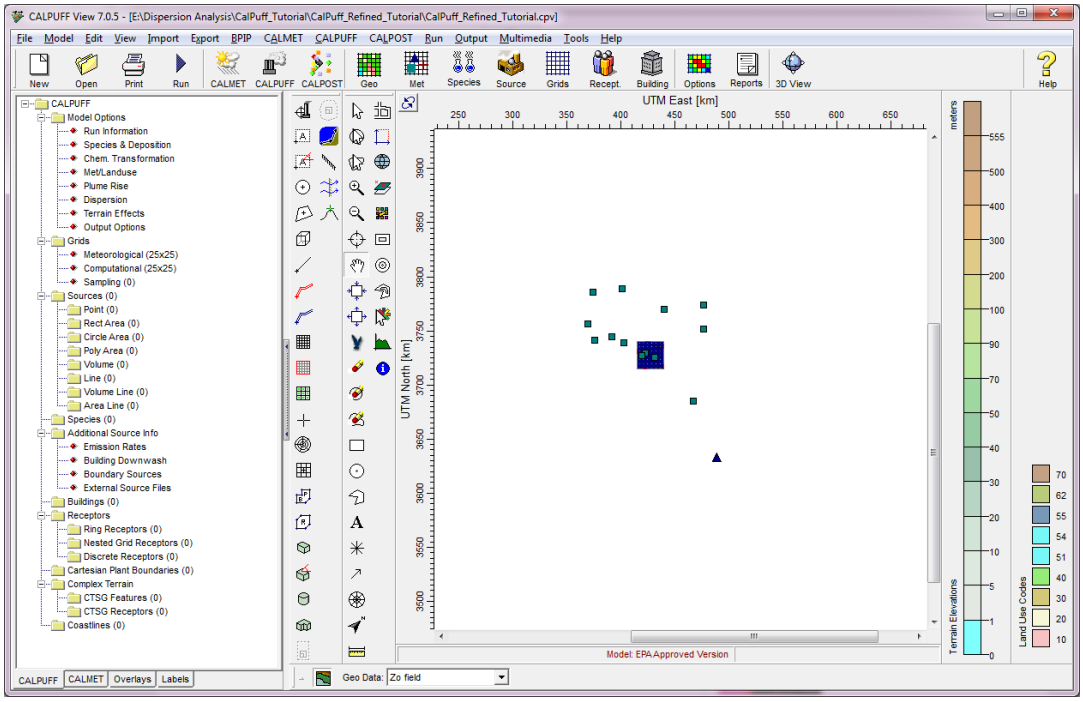

Process Surface Met Files
Time Zone for SURF.DAT
Model Domain Time Zone:
UTC-0800 Pacific N.America
The model domain time zone will be the same as in the Run Information dialog.
Append External SURF.DAT
File Name:
Use All Stations
Station Lat Long
Help
Process
Close



Upper Air Data (Blue Triangle)

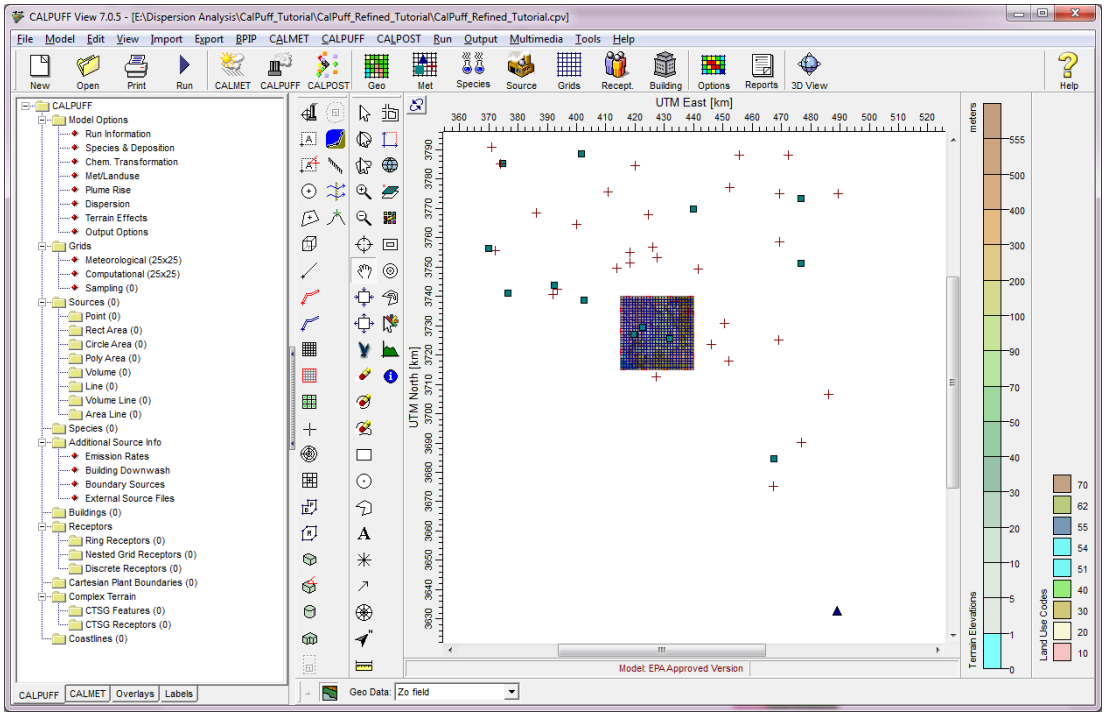
16. From the **Met Processor** window – **Upper Air** tab. Download Met data and click on **Process (Read62)**



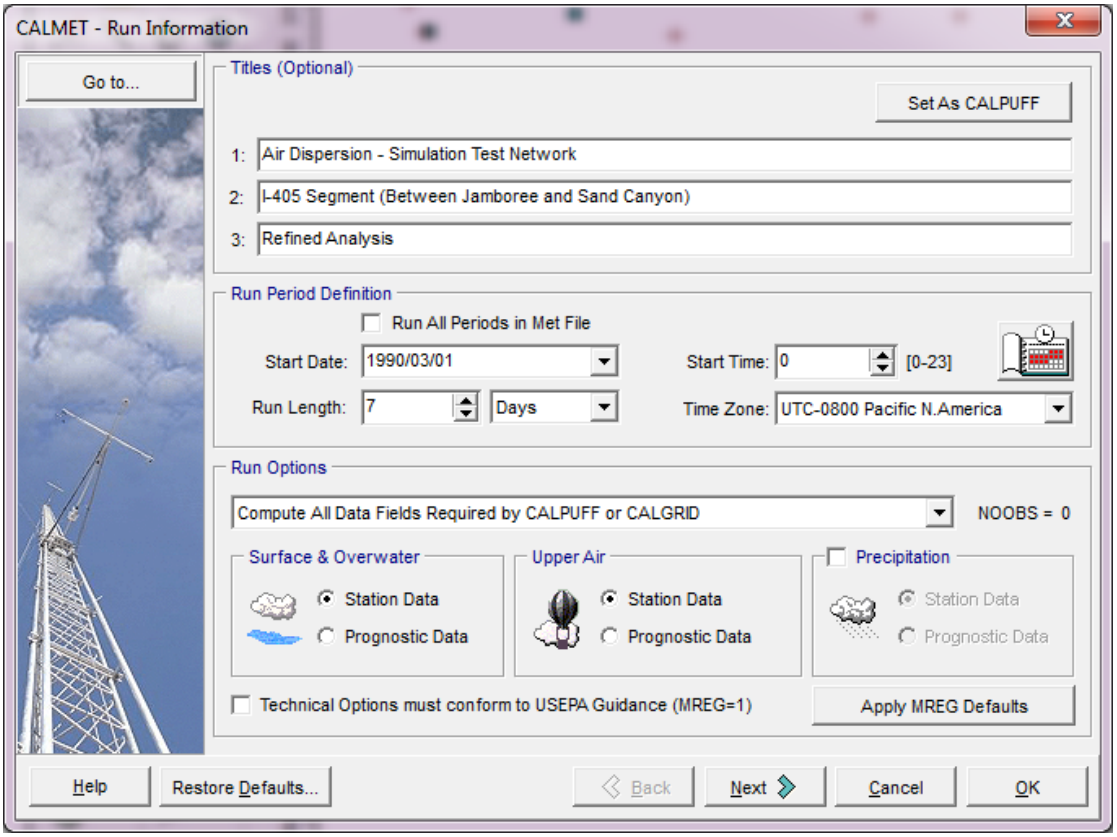


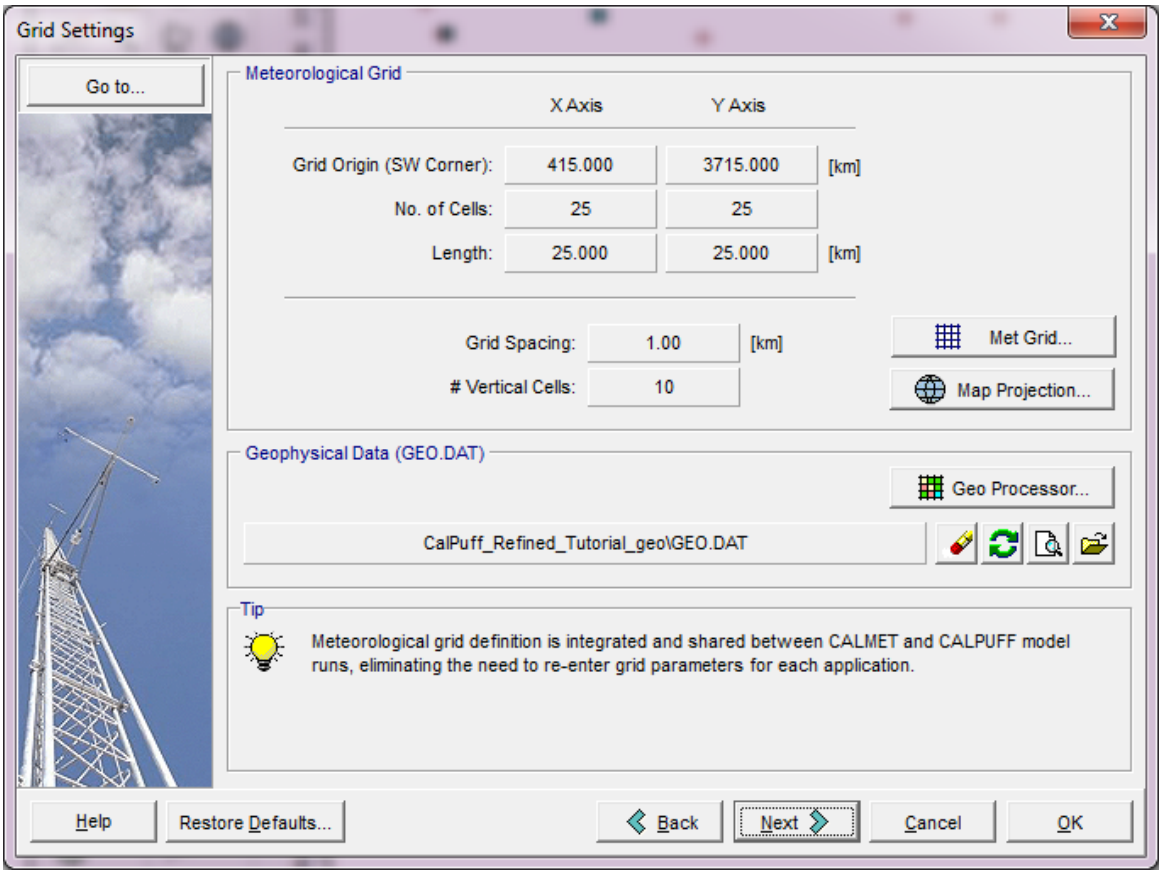
Precipitation Data (Maroon +)

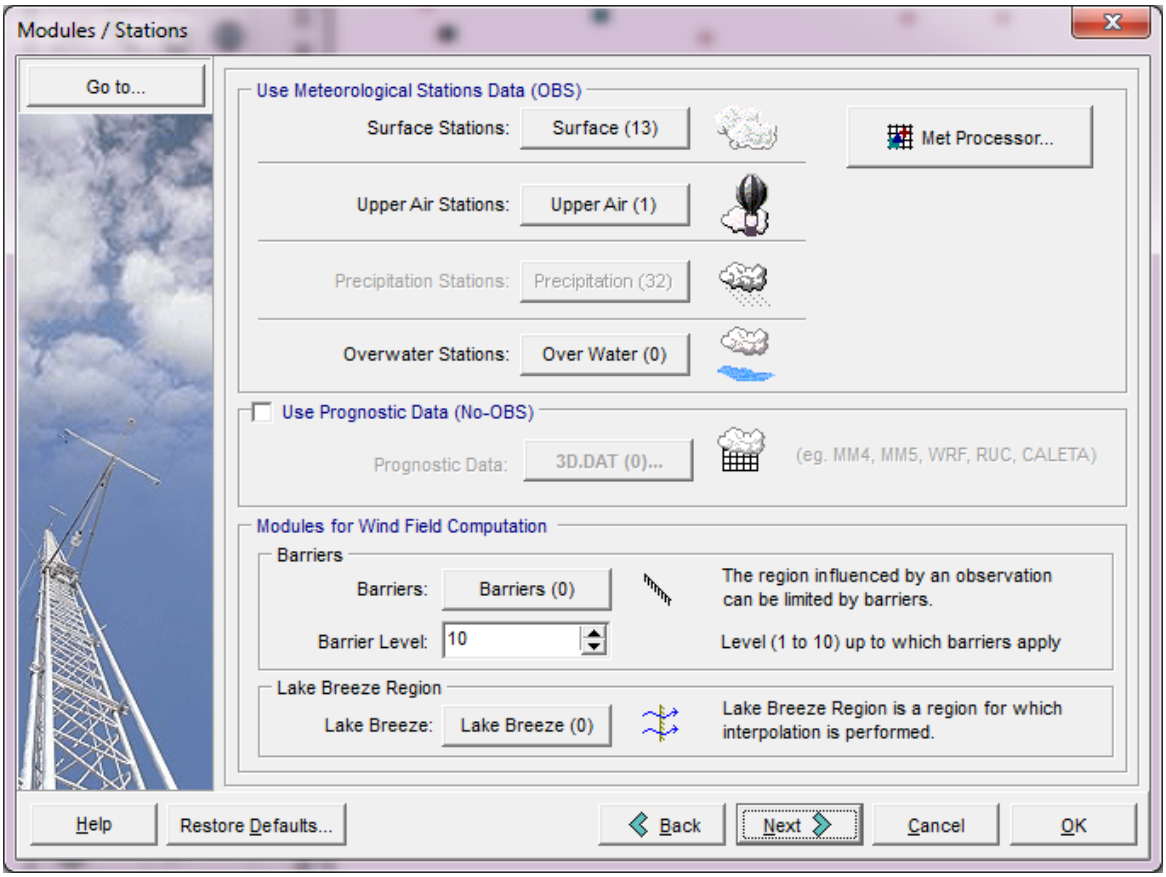
17. From the **Met Processor** window – **Precipitation** tab. Download Met data and click on **Process (PMERGE)**



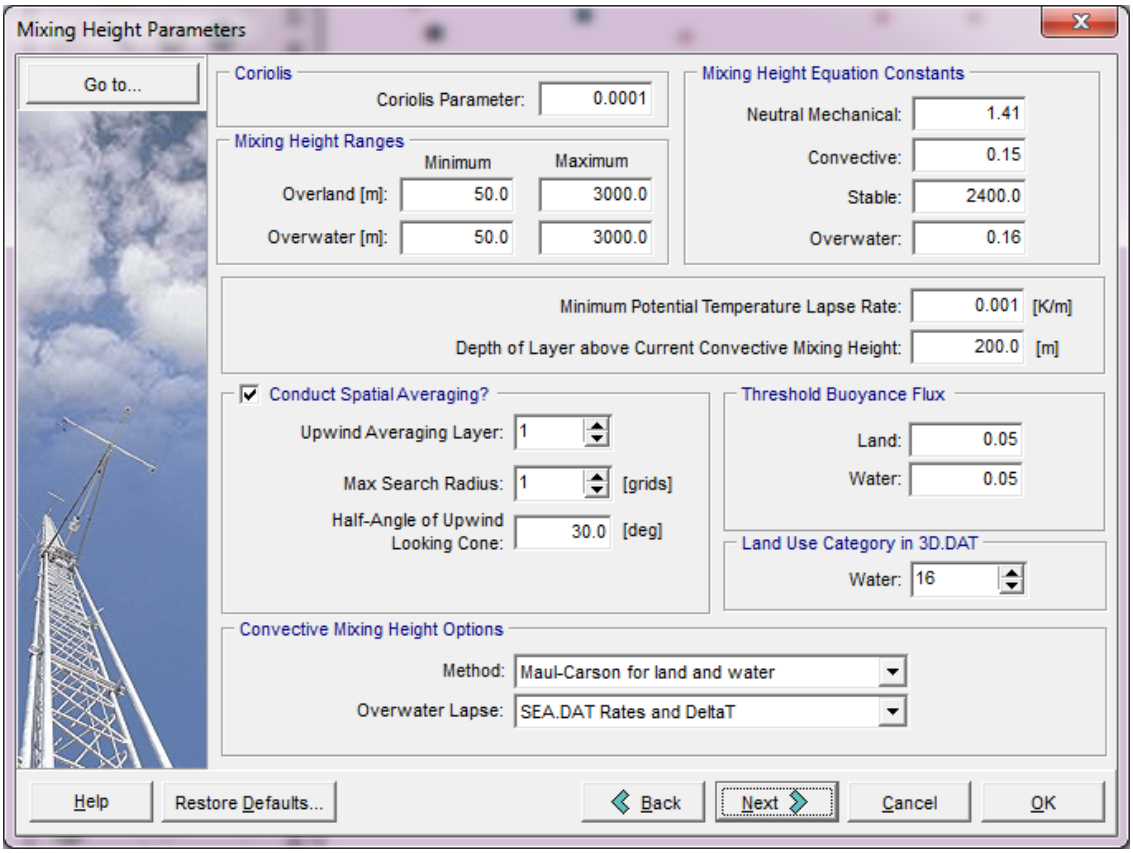
18. Click on the **CALMET** menu toolbar button to open the **CALMET Wizard**. And enter data as shown below.

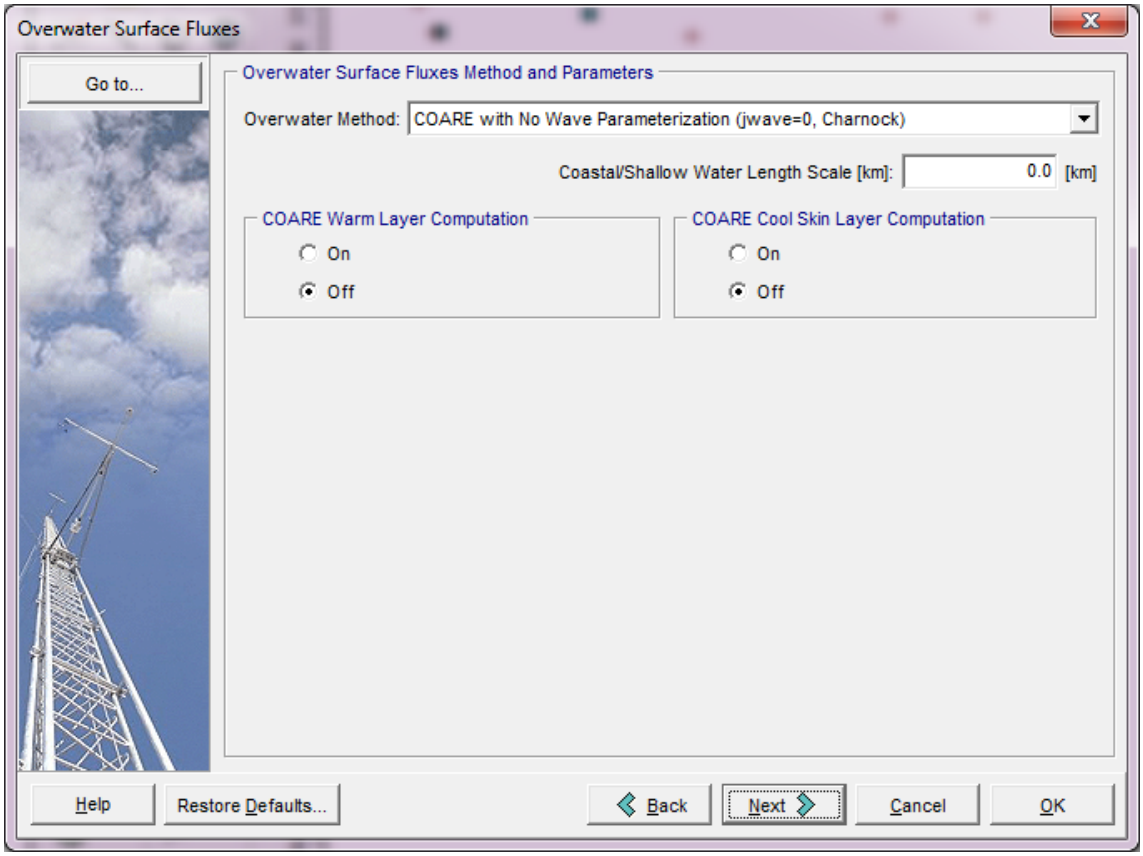


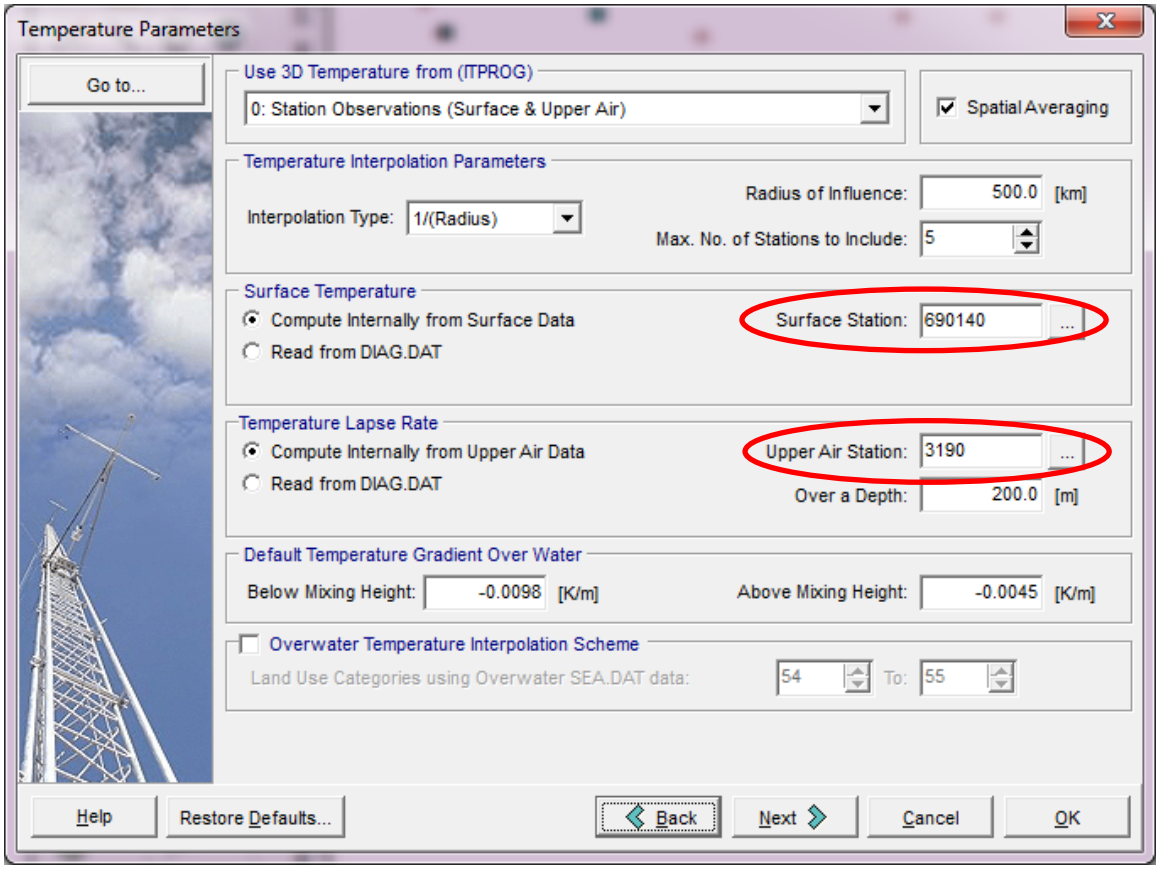




The **two screens** that follow contain default CALMET parameters that should not be changed. A typical CALMET wind field will use the default parameters specified here. Be sure that you fully understand the CALMET calculations before you change anyone of the default values. Click Next.




Overwater Surface Fluxes
Go to...
Overwater Surface Fluxes Method and Parameters
Overwater Method: COARE with No Wave Parameterization (jwave=0, Charnock)
Coastal/Shallow Water Length Scale [km]: 0.0 [km]
COARE Warm Layer Computation
On
Off
COARE Cool Skin Layer Computation
On
Off
Help
Restore Defaults...
Back
Next
Cancel
OK

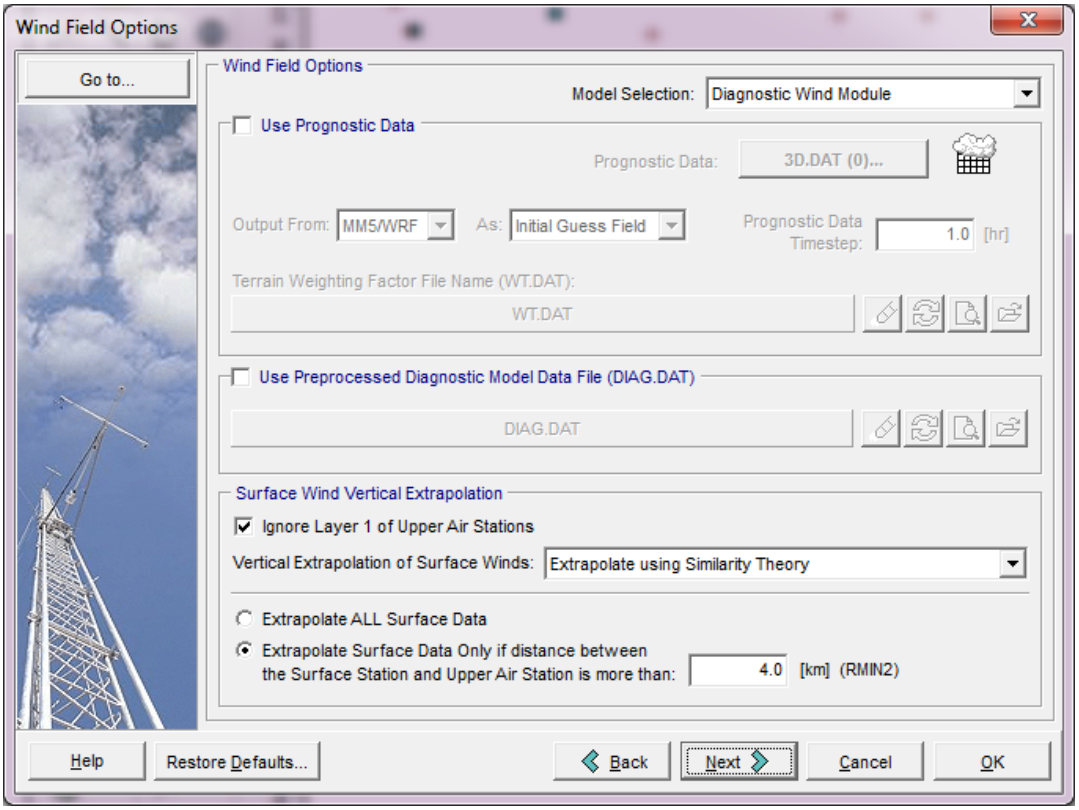

Temperature Parameters
Go to...
Use 3D Temperature from (ITPROG)
0: Station Observations (Surface & Upper Air)
Spatial Averaging
Temperature Interpolation Parameters
Interpolation Type: 1/(Radius)
Radius of Influence: 500.0 [km]
Max. No. of Stations to Include: 5
Surface Temperature
Compute Internally from Surface Data
Read from DIAG.DAT
Surface Station: 690140
Temperature Lapse Rate
Compute Internally from Upper Air Data
Read from DIAG.DAT
Upper Air Station: 3190
Over a Depth: 200.0 [m]
Default Temperature Gradient Over Water
Below Mixing Height: -0.0098 [K/m]
Above Mixing Height: -0.0045 [K/m]
Overwater Temperature Interpolation Scheme
Land Use Categories using Overwater SEA.DAT data: 54 To: 55
Help
Restore Defaults...
Back
Next
Cancel
OK

The Wind Field Options screen is where you specify your gridded wind fields of prognostic model output (e.g., MM4, MM5, WRF, or RUC). This is the first of four dialogs associated with the two-step wind field generation. The first option, listed at the top of this dialog, is to choose between a diagnostic or objective analysis of the observed data to produce the initial guess wind field.

1) Diagnostic Wind Model: This option uses a two-step approach to the computation of the wind fields. In Step 1 the initial guess wind field is adjusted for the kinematic effects of terrain, slope flows, blocking effects, and three-dimensional divergence minimization. Step 2 combines the available wind observations with the Step 1 gridded wind field to develop the final wind field.
2) Objective Analysis Only: This option provides an objective analysis of all available surface and upper-air observations. This option will generate a spatially varying wind field. When using this option the user is advised to allow the surface data to be extrapolated vertically, unless the surface data is strongly influenced by local terrain effects. As there are large differences in terrain in this example the diagnostic wind module will be used.

There is also an option to initialize the wind field using a gridded field from a prognostic model. This option is typically preferred if the data is available because it takes into account geographic features to develop its Mesoscale flow patterns. The Mesoscale Meteorology data provide a good starting point in the form of an initial guess wind field which can be further refined within CALMET based on local topography and weather observations.

Typically the first layer of upper air stations is ignored. Since upper air observations are only taken every 12 hours, the time interpolated surface wind values from the upper air observations are usually of no use. For a similar reason, the finer resolution hourly surface observations are typically extrapolated into the layers aloft, providing more information on observed local effects to the upper layers.

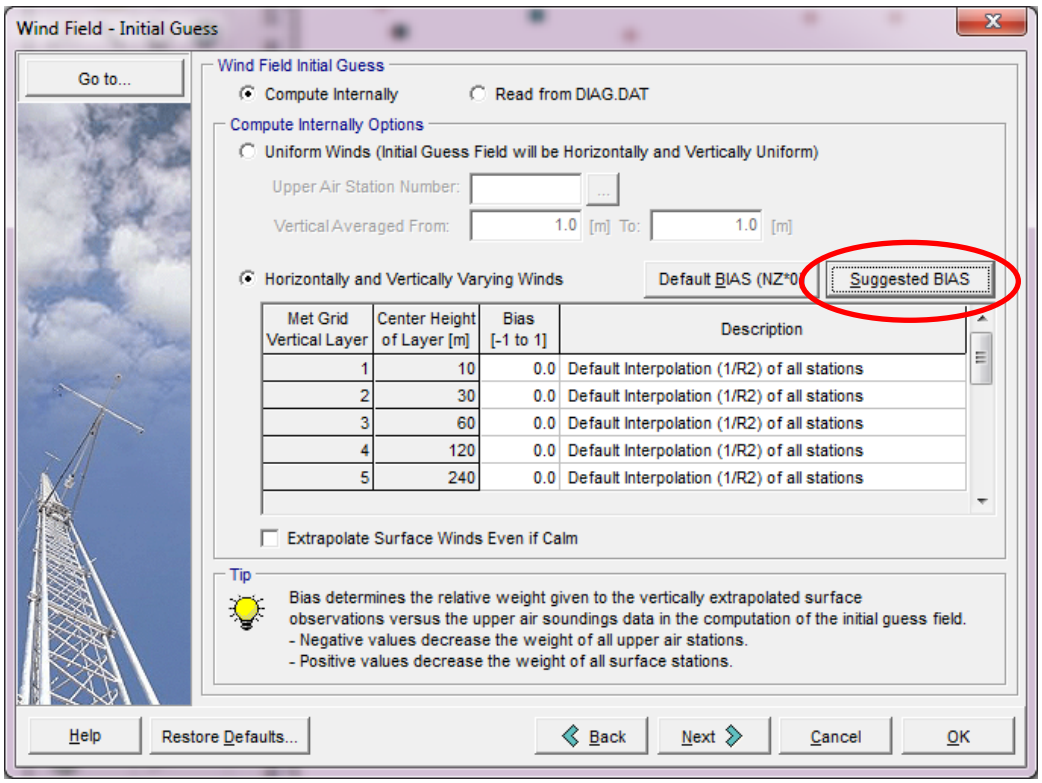


The **Wind Field – Initial Guess** screen is where more specific information is input for generating the initial guess wind field. It is generally discouraged to use the user specified DIAG.DAT to initialize the wind field, unless you are a very advanced modeler. Under this screen, you have the opportunity to dictate how the initial Step 1 winds will be interpolated to each grid cell in each vertical layer; this is controlled by the Bias values.

Bias = 0: Normally, Bias values are set to 0 so that the upper air and surface observations are given equal weight in the 1/r2 interpolations.
Bias = -1: This value indicates that the upper air observations are eliminated for that layer.
Bias = +1: this value indicates that the surface observations are eliminated for that layer. These are important to consider in regions of significant terrain, as the surface and upper air observations may not be directly related.

Press **Suggested BIAS** > **Yes**

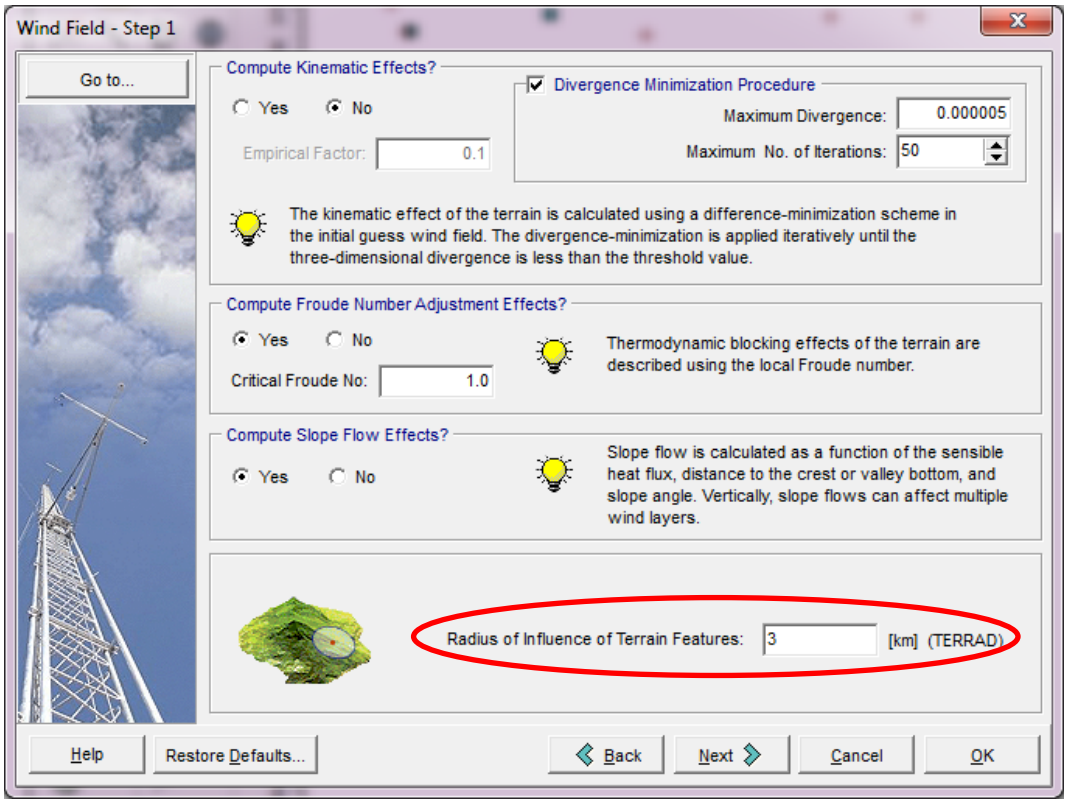
Please note that the Bias for the first layer must be -1.0 because **Ignore Layer 1 of Upper Air Stations** was selected in the **Wind Field Options** screen. Click **Next**.



Wind Field – Step 1 – when developing the Step 1 wind field, CALMET adjusts the initial guess field to reflect kinematic effects of the terrain, slope flows, and blocking effects. Slope flows are a function of the local slope and altitude of the nearest crest. The crest is defined as the highest peak within a radius (**TERRAD**) around each grid point.

If you have selected appropriate met grid spacing to properly reflect the modeled terrain, the **TERRAD** value will not be more than a few grid cells. Specifying the appropriate **TERRAD** value may take multiple CALMET runs.

Parameter	Value
Radius of Influence of Terrain Features	3 km (Grid spacing = 1 km, thus 3 Grid Cells)

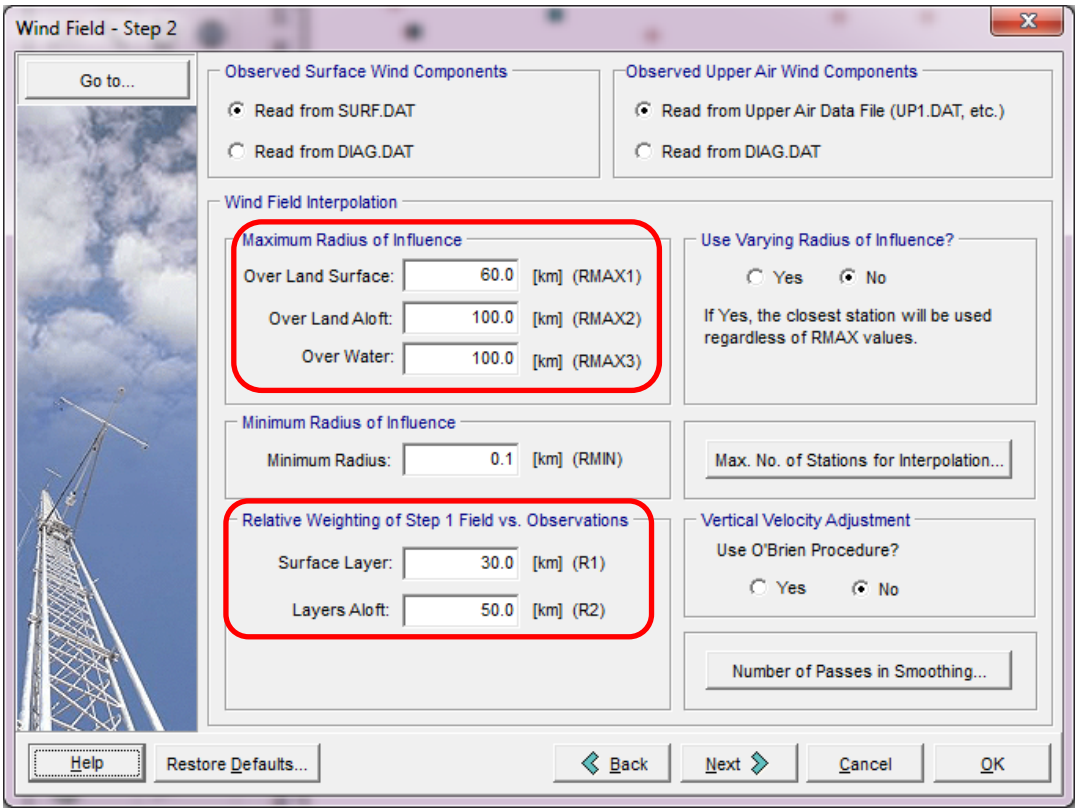


Wind Field – Step 2 Screen - In Step 2, observations are incorporated into the Step 1 wind field to produce a final wind field. Each observation site influences the final wind field within a radius of influence (parameters **RMAX1** at the surface and **RMAX2** aloft). Observations and Step 1 field are weighted by means of parameters **R1** at the surface and **R2** aloft: at a distance **R1** from an observation site, the Step 1 wind field and the surface observations are weighted equally.

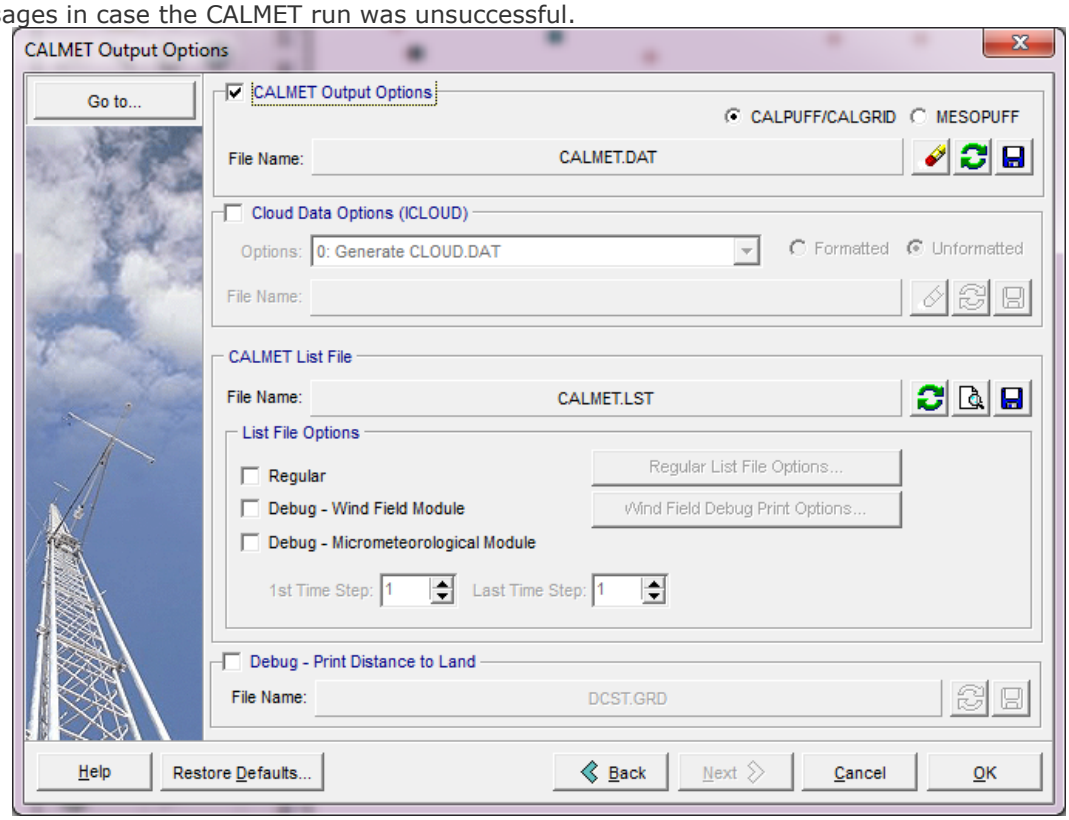
In complex terrain, channeling (blocking effects) and slope flows contribute significantly to the wind field. Therefore relatively small values of **R1** and **R2** would be selected (e.g. 1 km and 2 km), producing a large weight of the Step 1 field. In case **RMAX1** and **RMAX2** are set to large values (100 km), it would become irrelevant as the small **R1** and **R2** values limit the influence of the surface observations.

Maximum Radius of Influence	Value
Over Land Surface (RMAX1)	60 km
Over Land Aloft (RMAX2)	100 km
Over Water (RMAX3)	100 km
Relative Weighting	**Value**
Surface Layer (R1)	30 km
Layers Aloft (R2)	50 km

Check help file for below screen shot for selecting values:



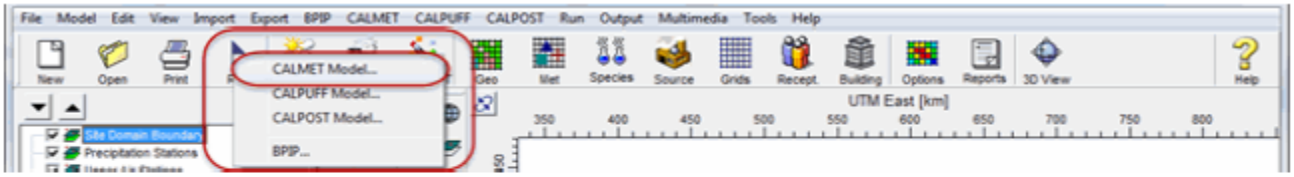
The final screen in the **CALMET Wizard** is the **CALMET Output Options**. In this screen, CALMET output files and options are displayed. The main output file for CALMET is the binary file **CALMET.DAT**. The **CALMET.LST** file is also called the List File. This file is in ASCII format and contains a summary of the CALMET input data, summary output data, and any error messages in case the CALMET run was unsuccessful.



Click **OK** to close the **CALMET** wizard.

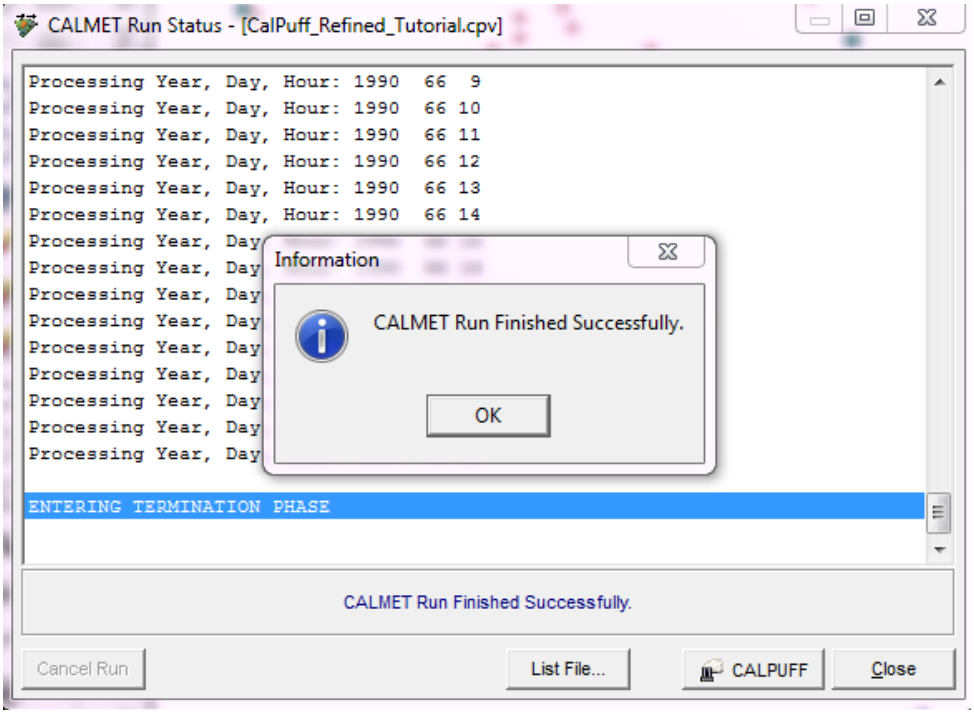
CALMET Run & Visualization

Press the **Run** menu toolbar button and then select **CALMET Model** from the pop up menu. This will start the CALMET run. If any information is missing, a message will be displayed.



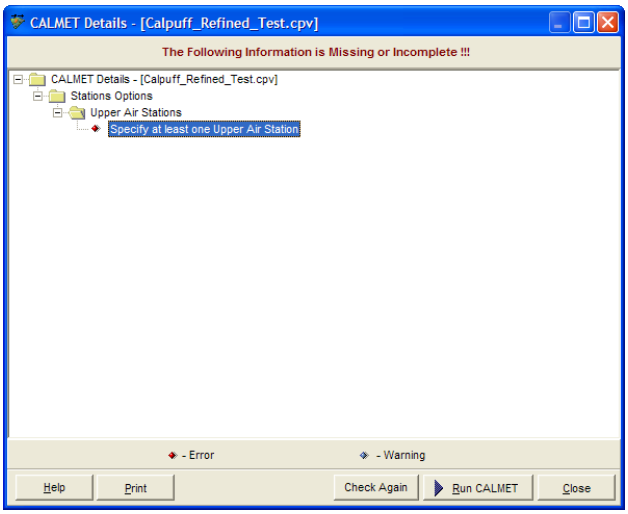
The most common reasons for a CALMET run to finish unsuccessfully are related to the parameters of the executable used. TRC Corp. provides three standard sizes of executable that differ primarily in the number of met grid cells you can have. You can specify which CALMET executable you would like to use under **File | Preferences |**

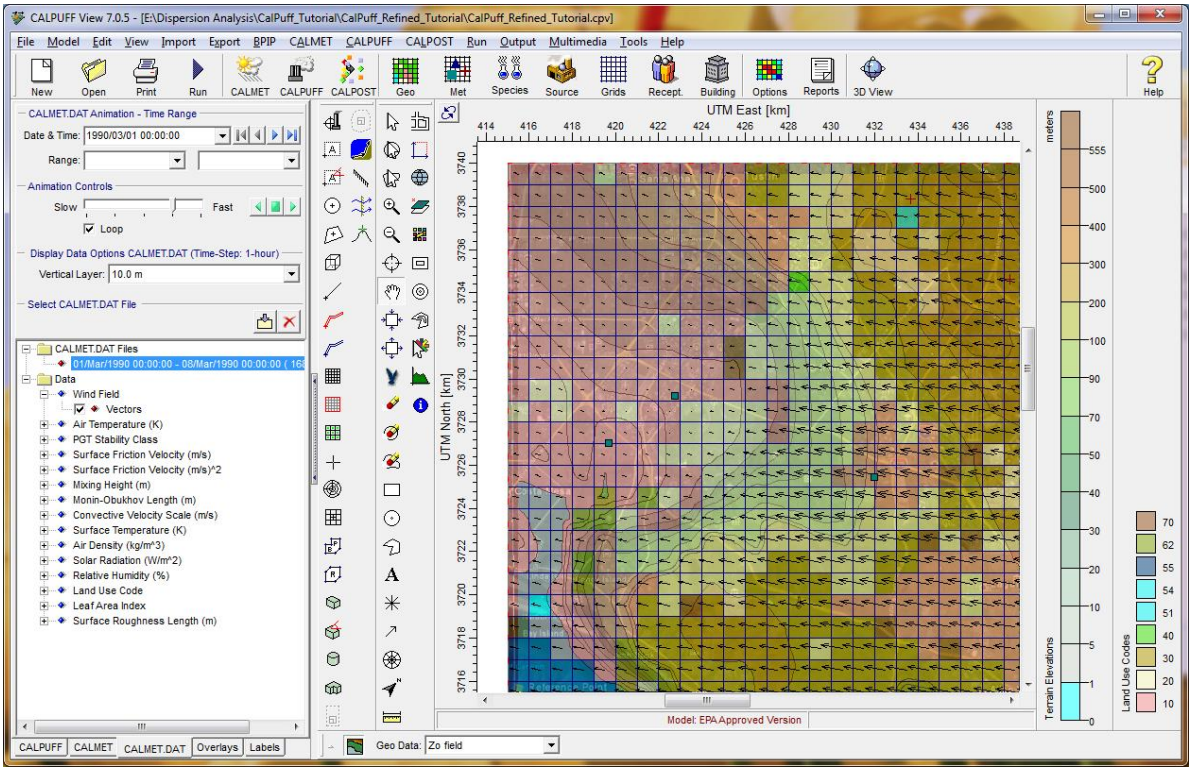
When the CALMET run finishes, a message will be displayed stating that the CALMET run finished successfully. Press the **OK** button and then the **Close** button. The CALMET wind field will be automatically displayed in the CALPUFF View graphical area.



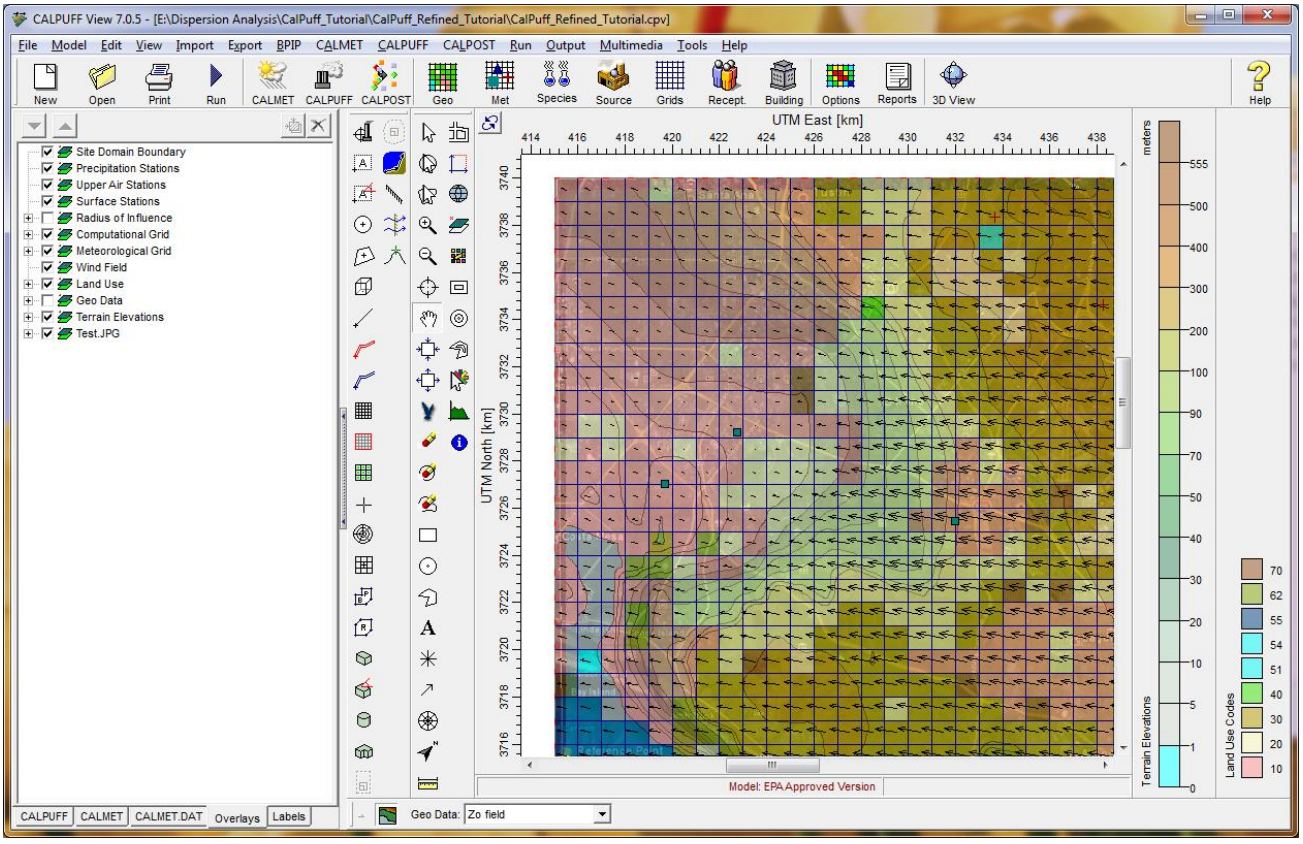
Select **Run | Error Checking | CALMET Details…** to see if any necessary parameters has been missed. If CALMET project has been completed successfully you will see “All Data Completed!”

Upper Air Station is always missing! Solution: Use Start Date: 1/1/1990 and End Date: 31/12/1990. This results access of 1 station 03160. Use second option to fix.

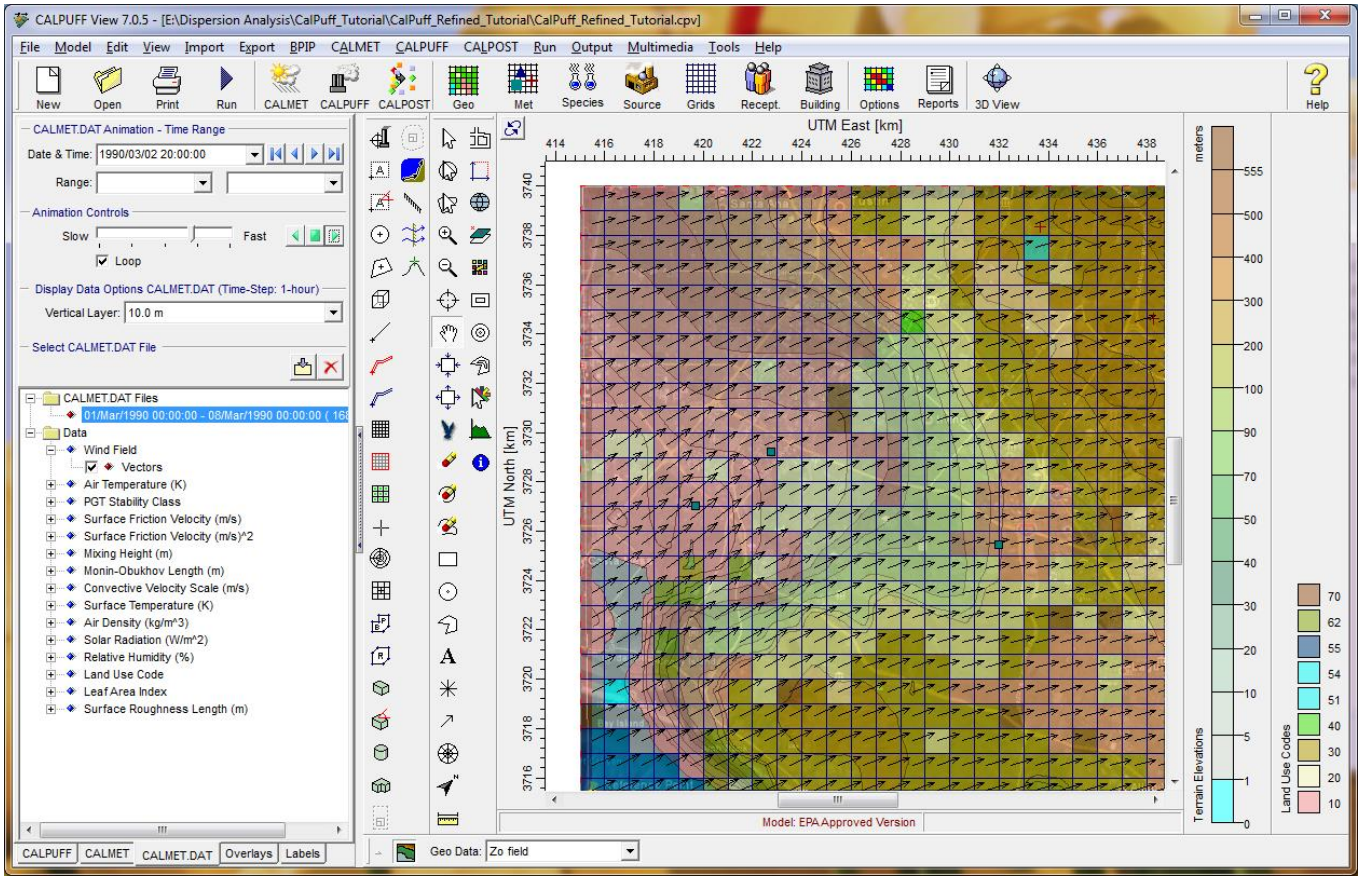




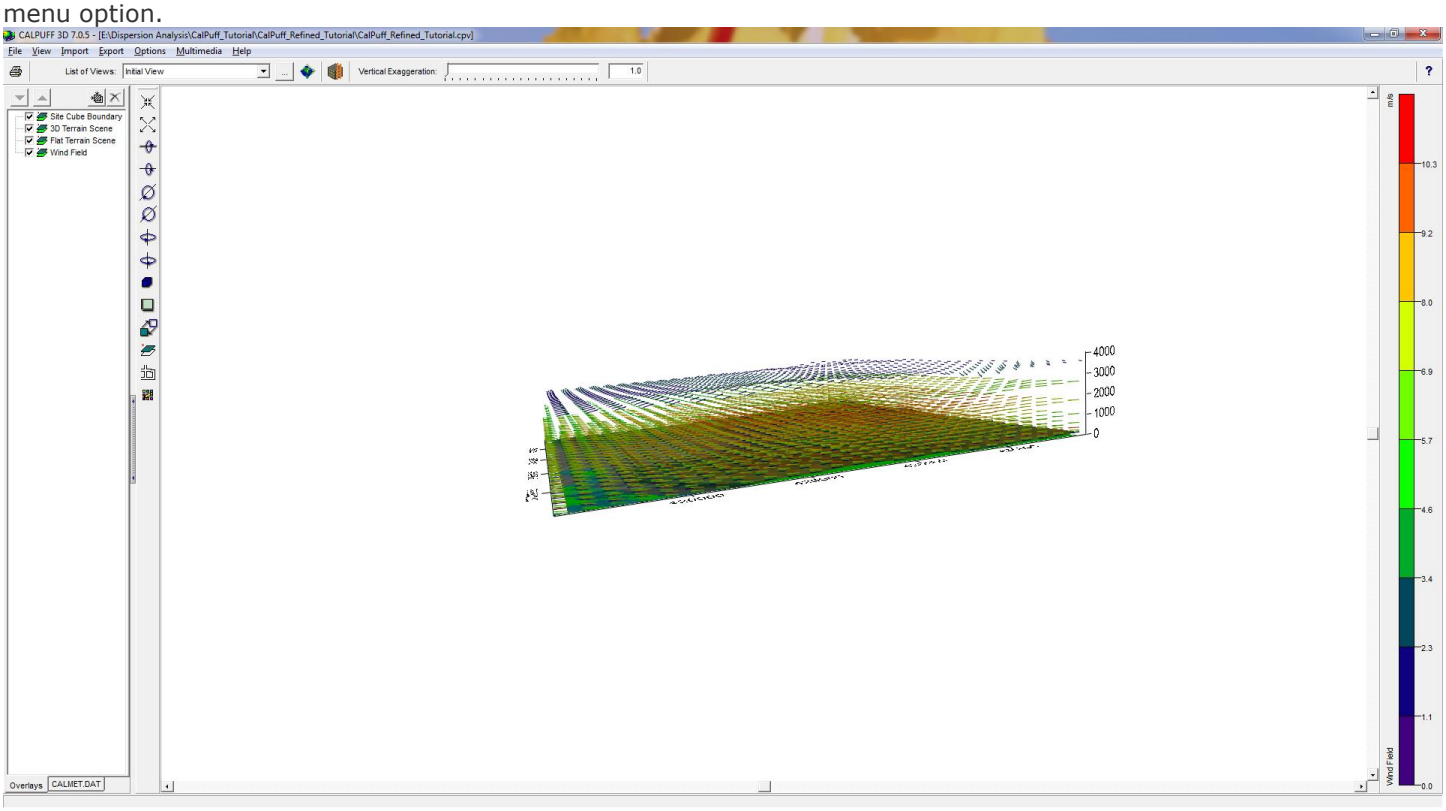
Under **Overlays** tab, located at the bottom section of the tree view panel, uncheck the boxes of several layers to make them invisible. Make sure only the **Wind Field** layer and the **Terrain Elevations** layer are checked. This will make the visualization of the wind field much clearer.



Using the selection tool in the **CALMET.DAT** tab, located at the top section of the tree view panel, select the vertical layer (e.g., 10.0 m) and preview the wind field for any hour by pressing the next, previous buttons available on the right side of the **Date & Time** field. You can also play an animation, scrolling through each hour automatically backwards or forwards, using **Animation Controls**.



You can also visualize the CALMET wind field in **3D** by pressing the button and selecting **3D View...** from the pop-up menu option.

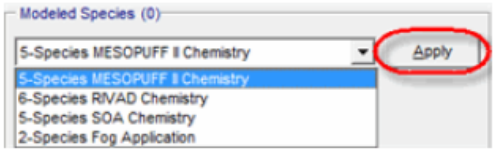
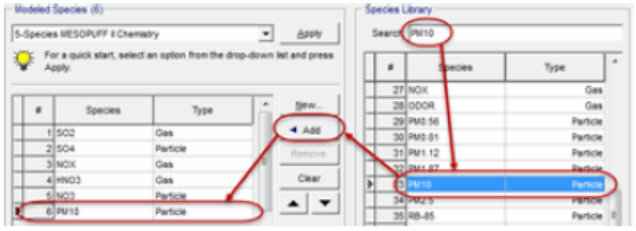


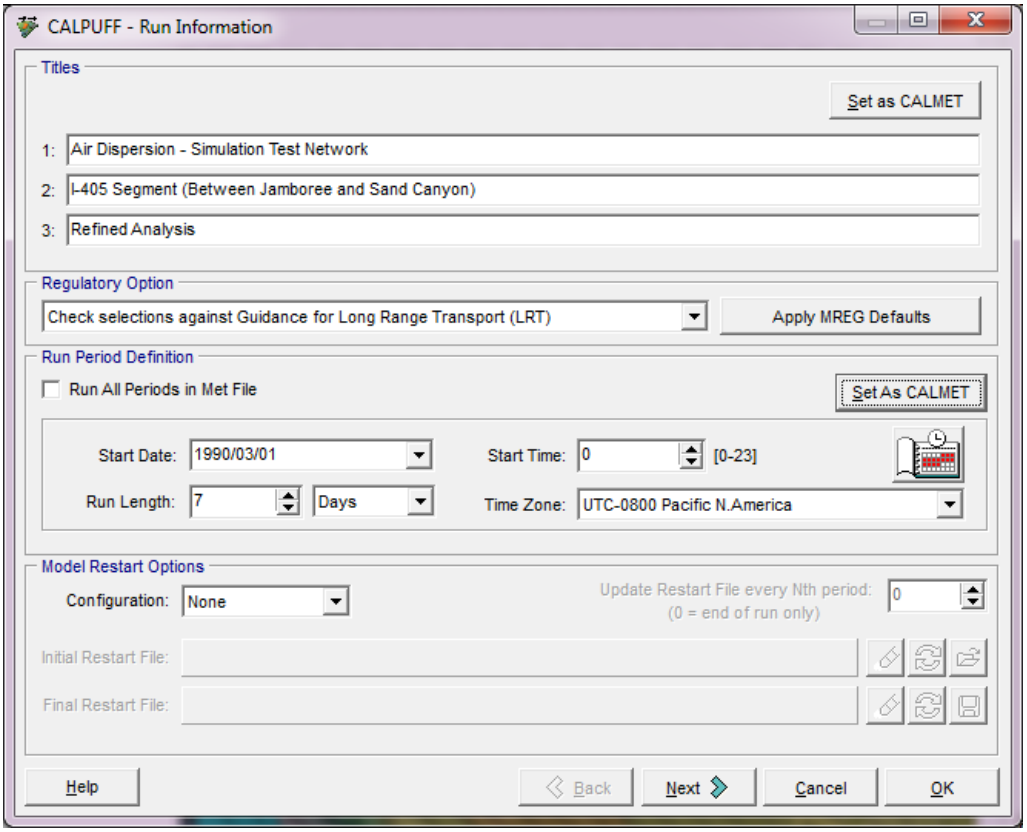
CALPUFF Options
CALPUFF Parameters

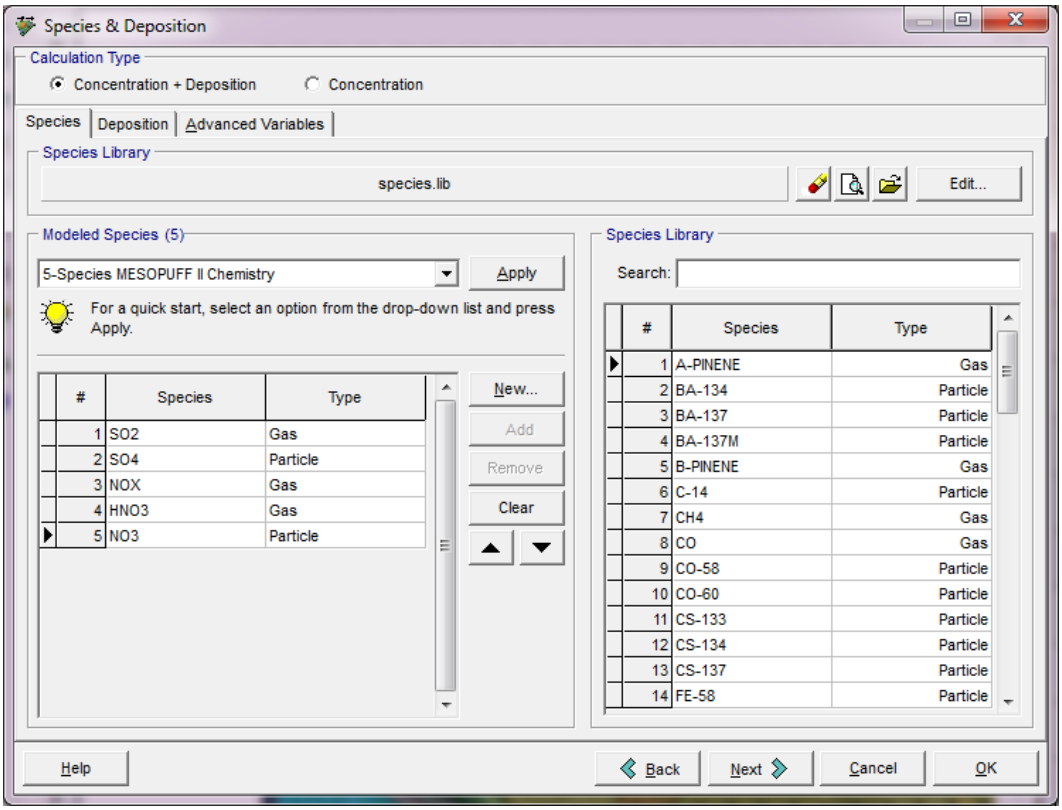
When you finish running the CALMET model and visualizing your wind field, press the **CALPUFF** button.

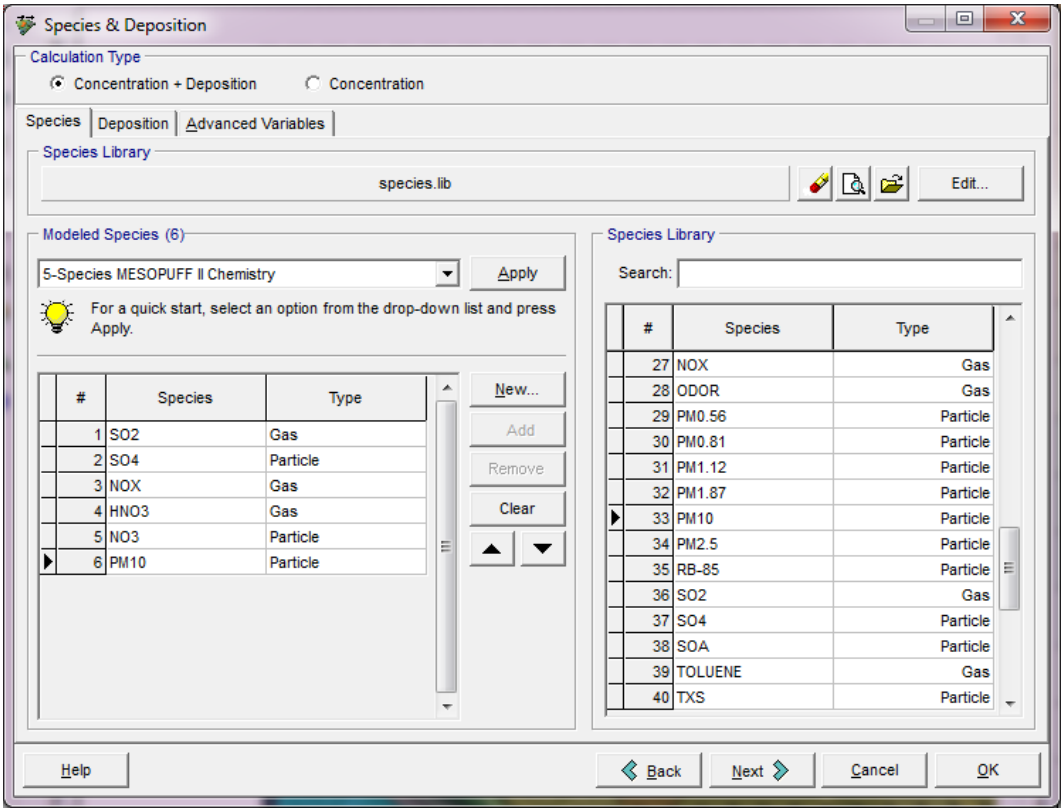
The **CALPUFF Wizard** is displayed**.** Enter the information according to table below in the several screens available in this wizard. Press the **Next** button to go through all the screens of the **CALPUFF Wizard**. If a page is not addressed in the table below, this means that no changes are required and you may skip it by clicking **Next** without changing anything.

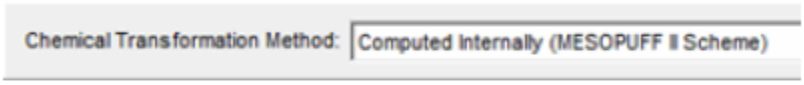
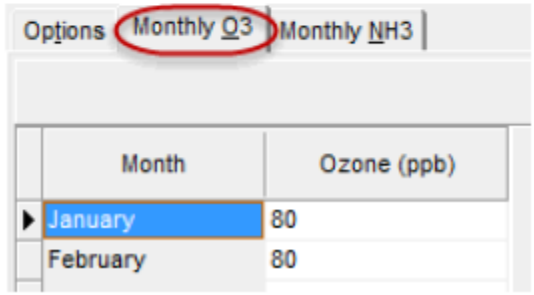
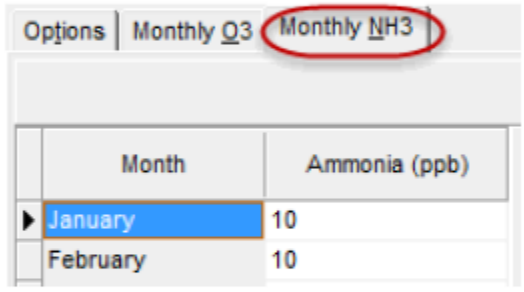
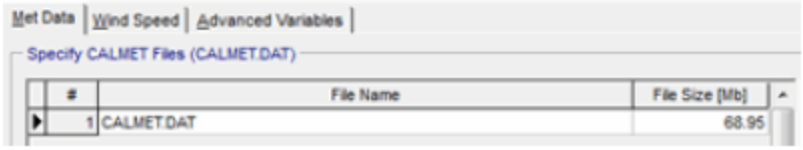
Wizard Screen	Option
Run Information	**Titles:** Specify any run information titles. **Tip:** Press the Set as CALMET button to copy the titles you specified in the **CALMET Wizard**.
Run Information	**Regulatory Option:** Check selections against Guidance for Long Range Transport (LRT) Regulatory Option: Check selections against Guidance for Long Range Transport (LRT) **Tip:** This option is used by US modelers performing Class I Area modeling projects.
Run Information	**Run Period:** Run the same period as you ran for CALMET. **Tip:** Press the Set as CALMET button to copy the Run Period you specified in the **CALMET Wizard**.

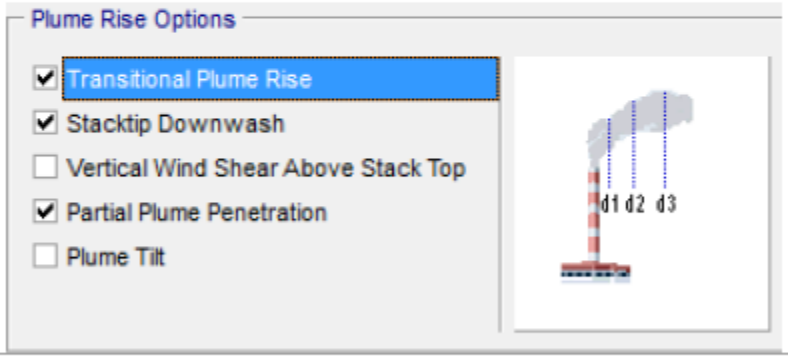
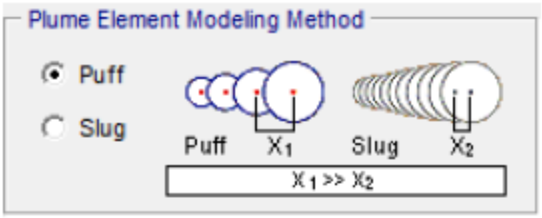
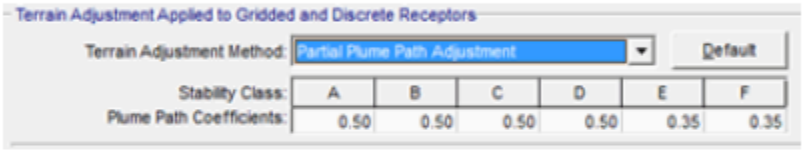
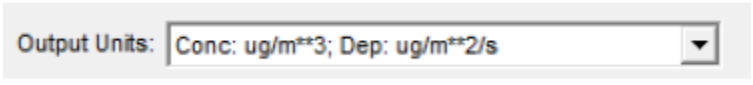
Species & Deposition	**Modeled Species:** SO2, SO4, NOX, HNO3, NO3, PM10 **Tip:** From the drop-down list, select the **5-Species MESOPUFF II Chemistry** option and press the **Apply** button. 
Species & Deposition	Using the **Species Library** on the right side of the dialog, select **PM10** and press the **Add** button to add PM10 to the bottom of the **Species** list. 




Species & Deposition
Calculation Type
Concentration + Deposition
Concentration
Species
Deposition
Advanced Variables
Species Library
species.lib
Edit...
Modeled Species (5)
5-Species MESOPUFF II Chemistry
Apply
For a quick start, select an option from the drop-down list and press Apply.
Species Type
1 SO2 Gas
2 SO4 Particle
3 NOX Gas
4 HNO3 Gas
5 NO3 Particle
New...
Add
Remove
Clear
Species Library
Search:
Species Type
1 A-PINENE Gas
2 BA-134 Particle
3 BA-137 Particle
4 BA-137M Particle
5 B-PINENE Gas
6 C-14 Particle
7 CH4 Gas
8 CO Gas
9 CO-58 Particle
10 CO-60 Particle
11 CS-133 Particle
12 CS-134 Particle
13 CS-137 Particle
14 FE-58 Particle
Help
Back
Next
Cancel
OK


Species & Deposition
Calculation Type
Concentration + Deposition
Concentration
Species
Deposition
Advanced Variables
Species Library
species.lib
Edit...
Modeled Species (6)
5-Species MESOPUFF II Chemistry
Apply
For a quick start, select an option from the drop-down list and press Apply.
Species Type
1 SO2 Gas
2 SO4 Particle
3 NOX Gas
4 HNO3 Gas
5 NO3 Particle
6 PM10 Particle
New...
Add
Remove
Clear
Species Library
Search:
Species Type
27 NOX Gas
28 ODOR Gas
29 PM0.56 Particle
30 PM0.81 Particle
31 PM1.12 Particle
32 PM1.87 Particle
33 PM10 Particle
34 PM2.5 Particle
35 RB-85 Particle
36 SO2 Gas
37 SO4 Particle
38 SOA Particle
39 TOLUENE Gas
40 TXS Particle
Help
Back
Next
Cancel
OK

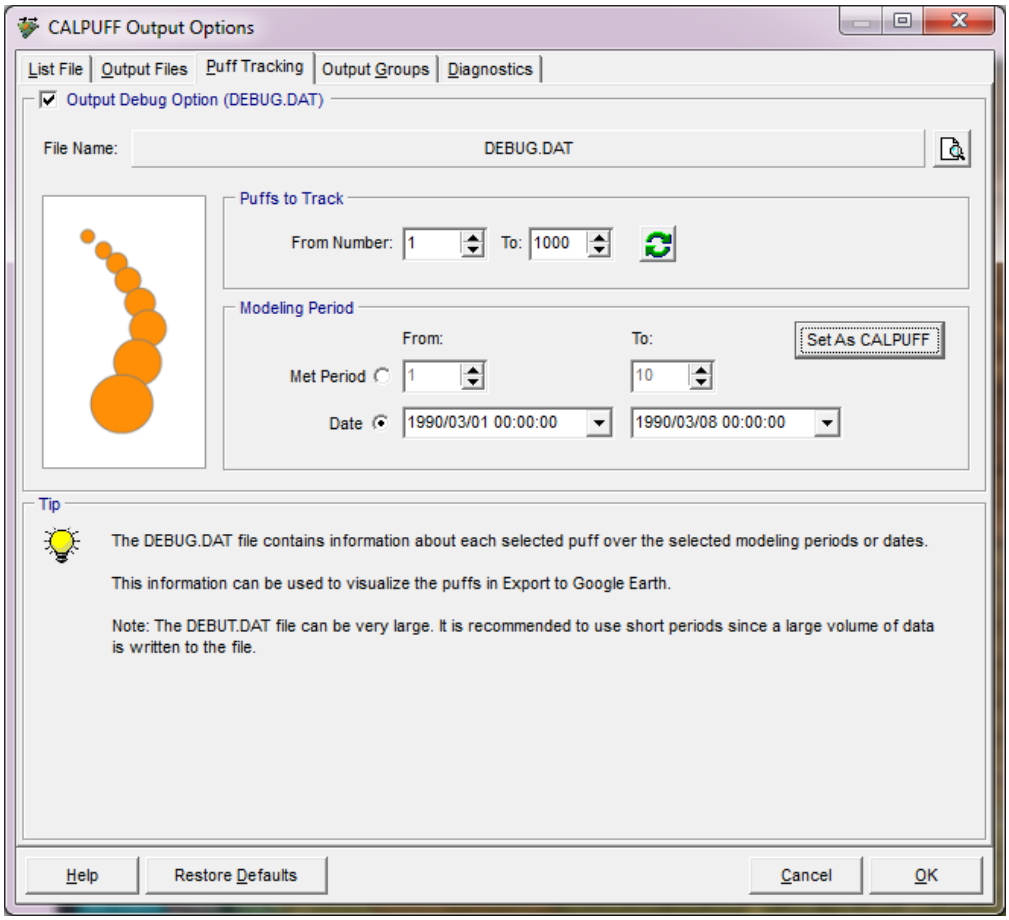
Wizard Screen	Option
Chemical Transformation	**Chemical Transformation Method:** Computed Internally (MESOPUFF II Scheme). 
Chemical Transformation	**Monthly O3:** For this tutorial use the default values of 80 ppb for all months. **Tip:** The monthly constant background concentrations of ozone (O3) should be specific to your modeling region. 
Chemical Transformation	**Monthly NH3:** For this tutorial use the default values of 10 ppb for all months. **Tip:** The monthly constant background concentrations of ammonia (NH3) should be specific to your modeling region. 
Meteorological/Land Use	**Met Data:** If your CALMET run was successful, then you should see the **CALMET.DAT** file (output from CALMET run) automatically specified under the table. 

Wizard Screen	Option
Plume Rise	**Plume Rise Options:** Make sure the options below are checked. • Transitional Plume Rise • Stacktip Downwash • Partial Plume Penetration 
Dispersion	Select **Puff.** 
Complex Terrain Effects	Select **Partial Plume Path Adjustment.** 
CALPUFF Output Options - List File	**Output Units:** Conc: ug/m**3, Dep: ug/m**2/s 
CALPUFF Output Options - List File	Leave default selections.
CALPUFF Output Options - Output Files	Make sure the option for **Relative Humidity - Visibility Output File** is checked (VISB.DAT).

Under the **CALPUFF Wizard - CALPUFF Output Options** screen, press the **Puff Tracking** tab. See the next step, Puff Tracking, for instructions on how to use this option.

Puff Tracking (Doesn't work for Area Sources- Uncheck Output Debug Option)

The **Puff Tracking** option allows you to visualize individual puffs for each emission source for the duration of the specified modeling period. The visualization of these puffs is done in Google Earth, after the CALPUFF model is run.
Please note that the **Puff Tracking** option produces a very large CALPUFF.LST file that can reach many gigabytes in size. Therefore, you should set the modeling period for this option to a smaller number of days in case the CALPUFF model is being run for a longer period of time.



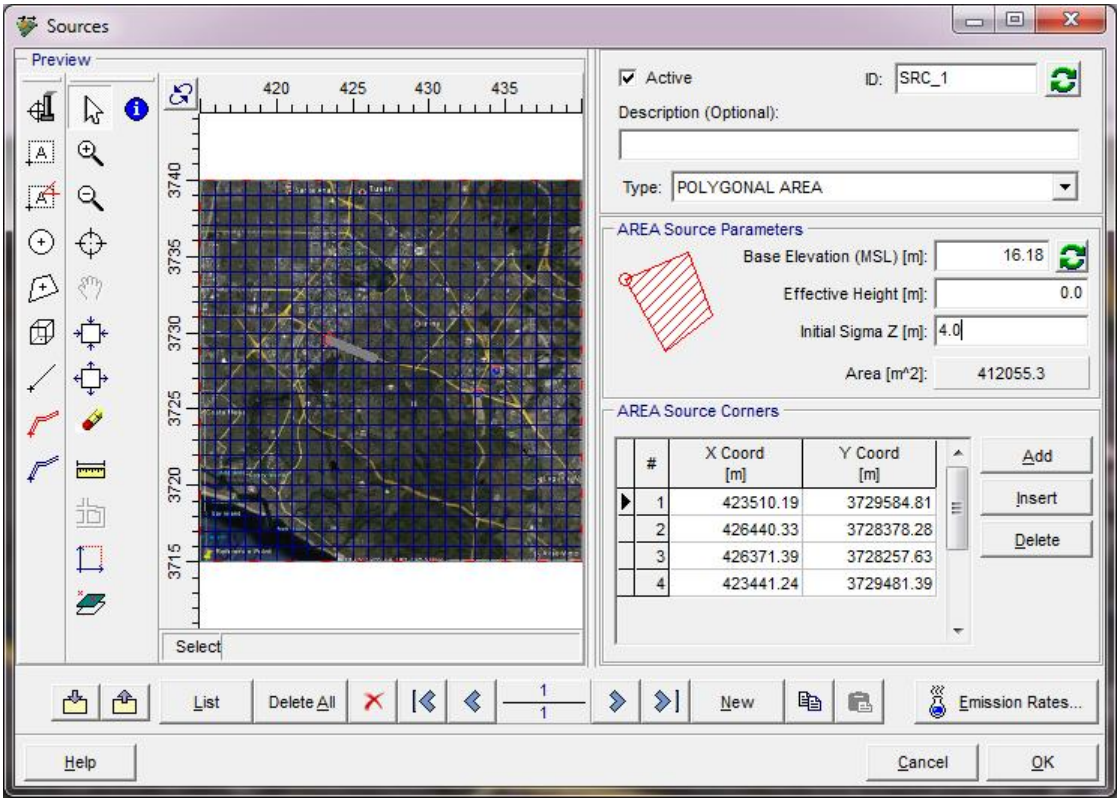
When you finish specifying the puff tracking option, then press the **OK** button to close the **CALPUFF Wizard**.

Specifying Source and Emissions

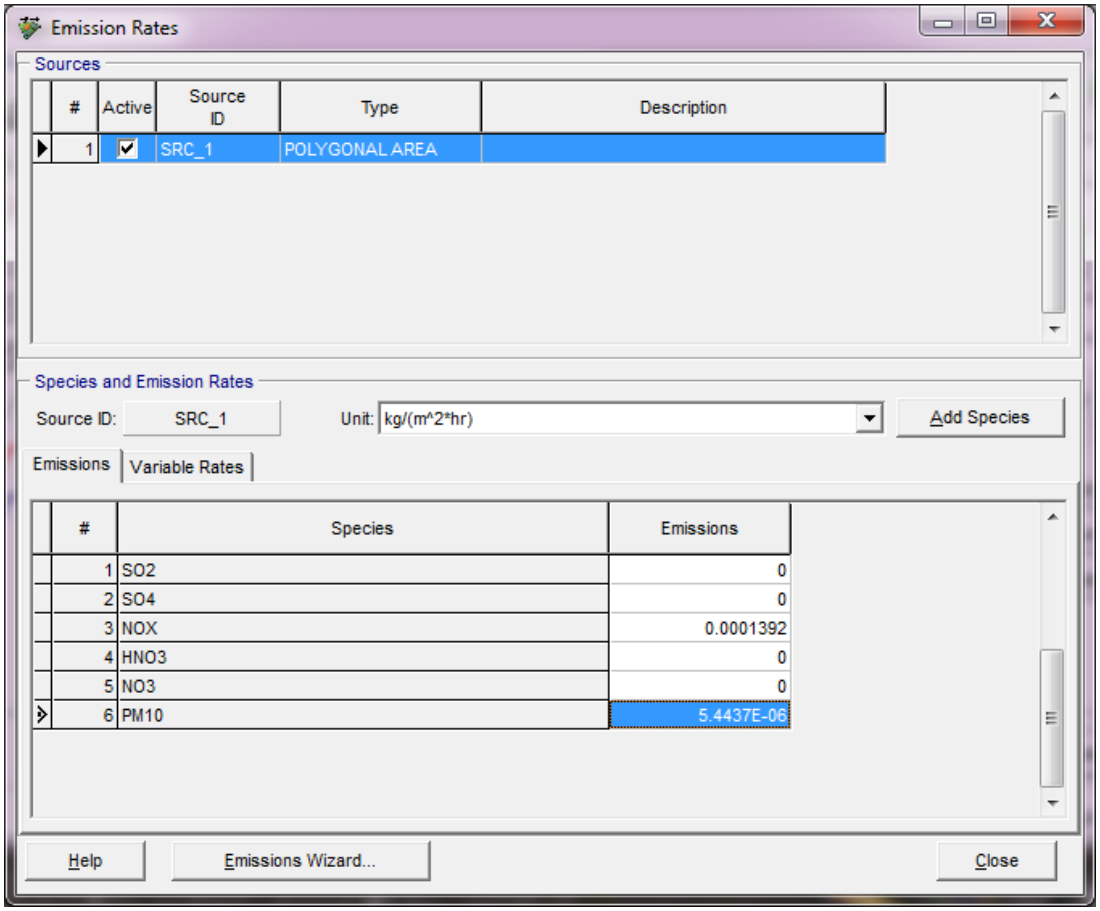
Follow the steps below on how to specify source parameters for the **Refined Tutorial**:

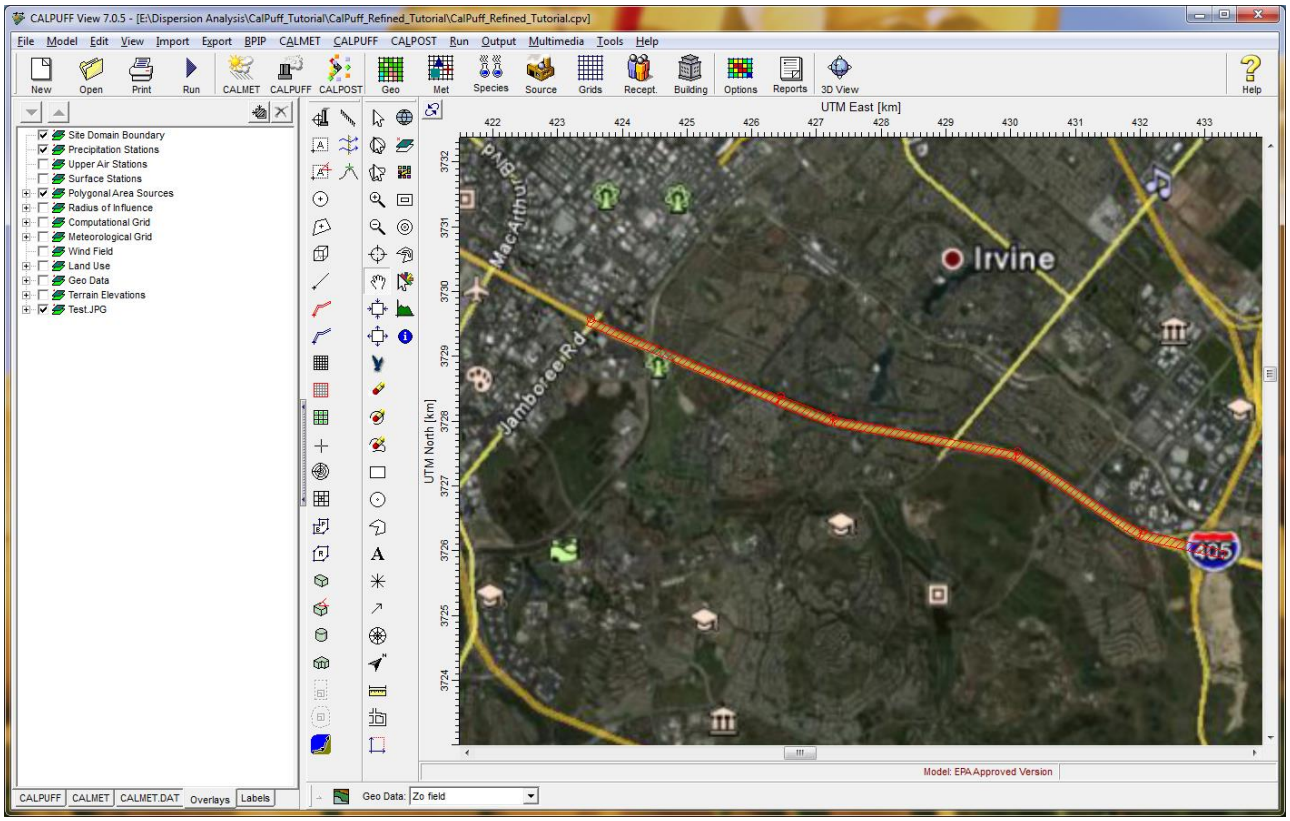
Using the **Polygon Area Source** tool, click at the border of study area freeway segment. The **Sources** dialog is displayed.

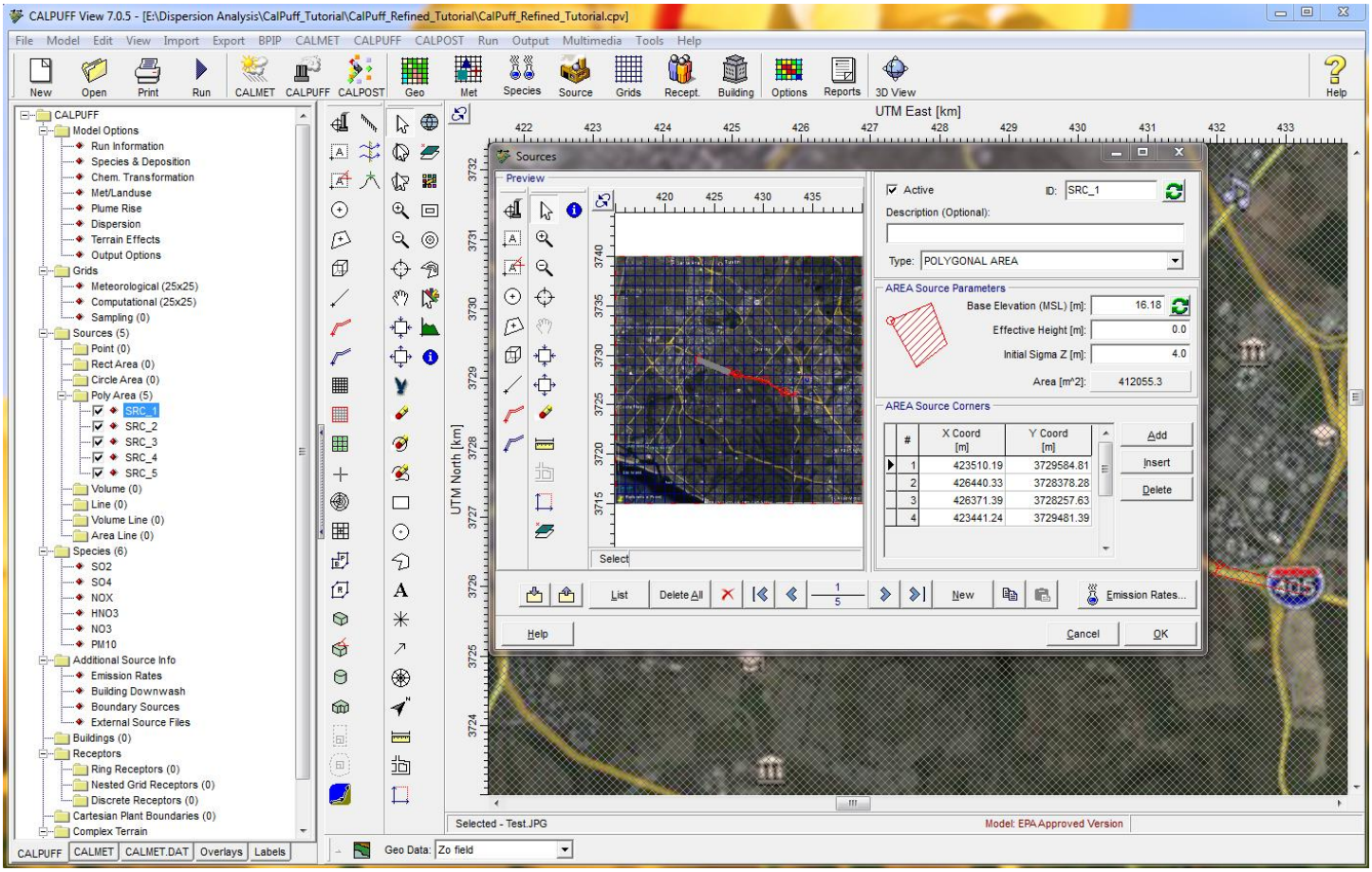
Effective Height [m] = 0
Initial Sigma Z [m] = 4.0



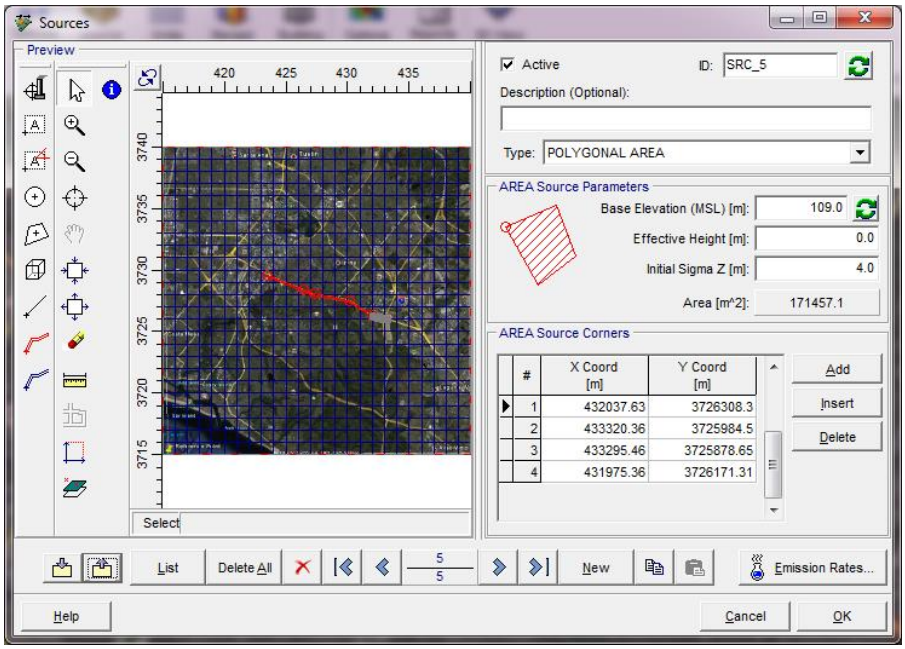
After specifying the source parameters, press the **Emission Rates** button. **Unit: Kg/(m^2*Hr).** Then enter emission rates:
These emission rates (24 hour average) are from Microscopic Simulation > Emission Estimation
NO_x = 0.0001392 Kg/(m^2*Hr)
PM_{10} = 5.4437 E-06 Kg/(m^2*Hr)

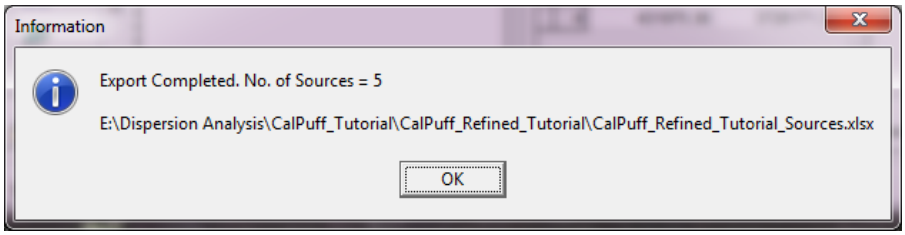


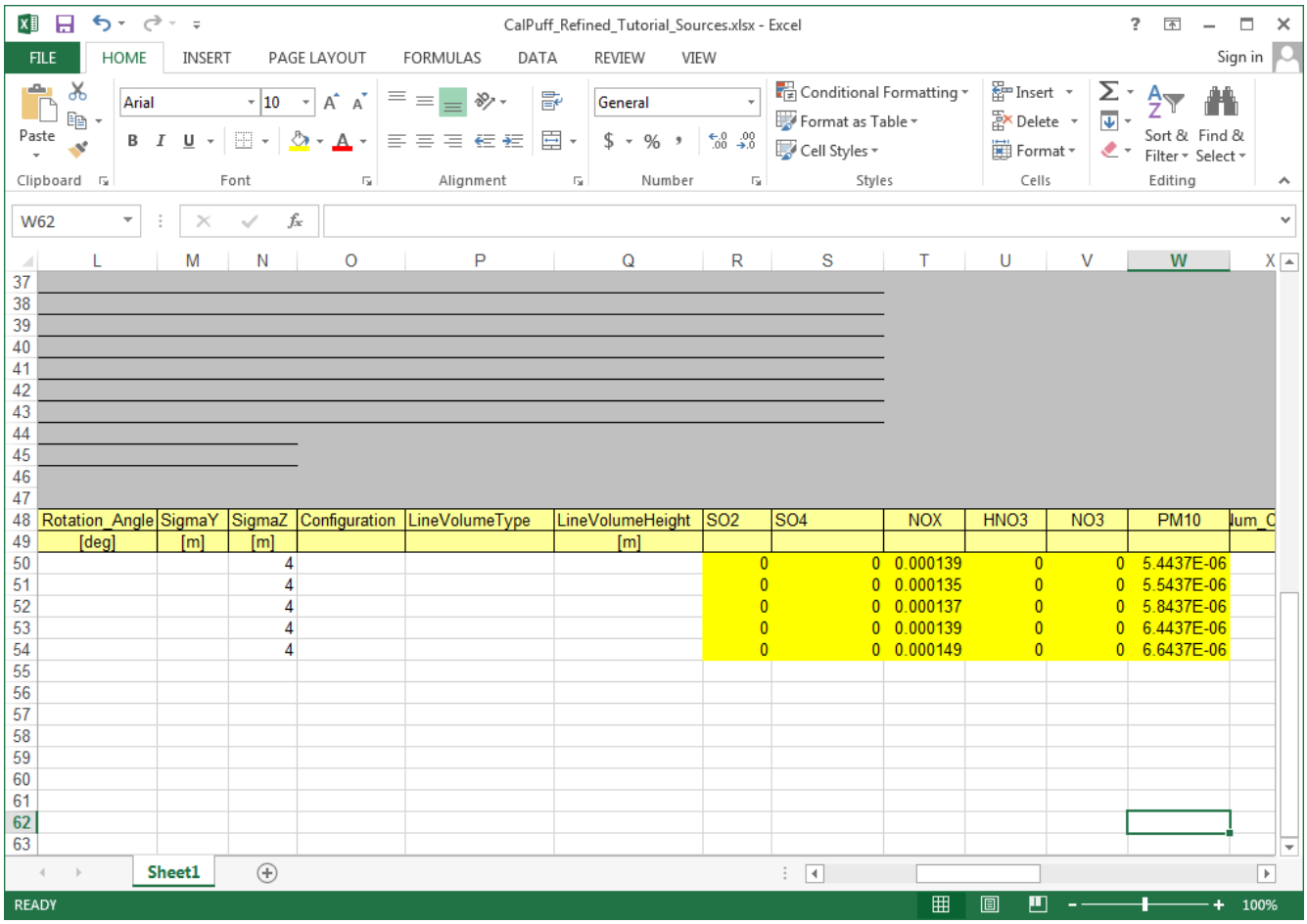

CALPUFF View 7.0.5 - [E:\Dispersion Analysis\CalPuff_Tutorial\CalPuff_Refined_Tutorial\CalPuff_Refined_Tutorial.cpv]
File Model Edit View Import Export BPIP CALMET CALPUFF CALPOST Run Output Multimedia Tools Help
New Open Print Run CALMET CALPUFF CALPOST Geo Met Species Source Grids Recept. Building Options Reports 3D View Help
Site Domain Boundary
Precipitation Stations
Upper Air Stations
Surface Stations
Polygonal Area Sources
Radius of Influence
Computational Grid
Meteorological Grid
Wind Field
Land Use
Geo Data
Terrain Elevations
Test.JPG
UTM East [km]
UTM North [km]
Irvine
MacArthur
JamboreeRd
405
Model: EPA Approved Version
CALPUFF CALMET CALMET.DAT Overlays Labels
Geo Data: Zo field


CALPUFF View 7.0.5 - [E:\Dispersion Analysis\CalPuff_Tutorial\CalPuff_Refined_Tutorial\CalPuff_Refined_Tutorial.cpv]
Sources
Preview
Active
ID: SRC_1
Description (Optional):
Type: POLYGONAL AREA
AREA Source Parameters
Base Elevation (MSL) [m]: 16.18
Effective Height [m]: 0.0
Initial Sigma Z [m]: 4.0
Area [m^2]: 412055.3
AREA Source Corners
X Coord [m] Y Coord [m]
1 423510.19 3729584.81
2 426440.33 3728378.28
3 426371.39 3728257.63
4 423441.24 3729481.39
Add
Insert
Delete
List
Delete All
New
Emission Rates...
Help
Cancel
OK
Selected - Test.JPG
Model: EPA Approved Version

Export "Sources.xlsx" for entering Emission Rates for different sources.







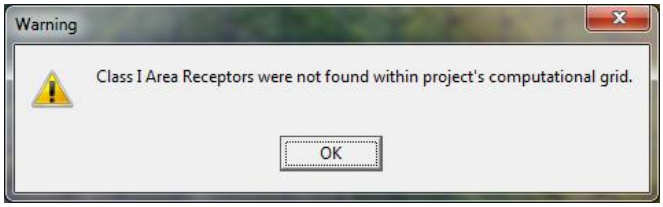
Load "Source.xlsx" data: Top Ribbon > Import > Sources

Note: You need to delete all sources before importing!!!

Specifying Discrete Receptors

Follow the steps below on how to specify the Class I Area discrete receptors for the **Refined Tutorial**:

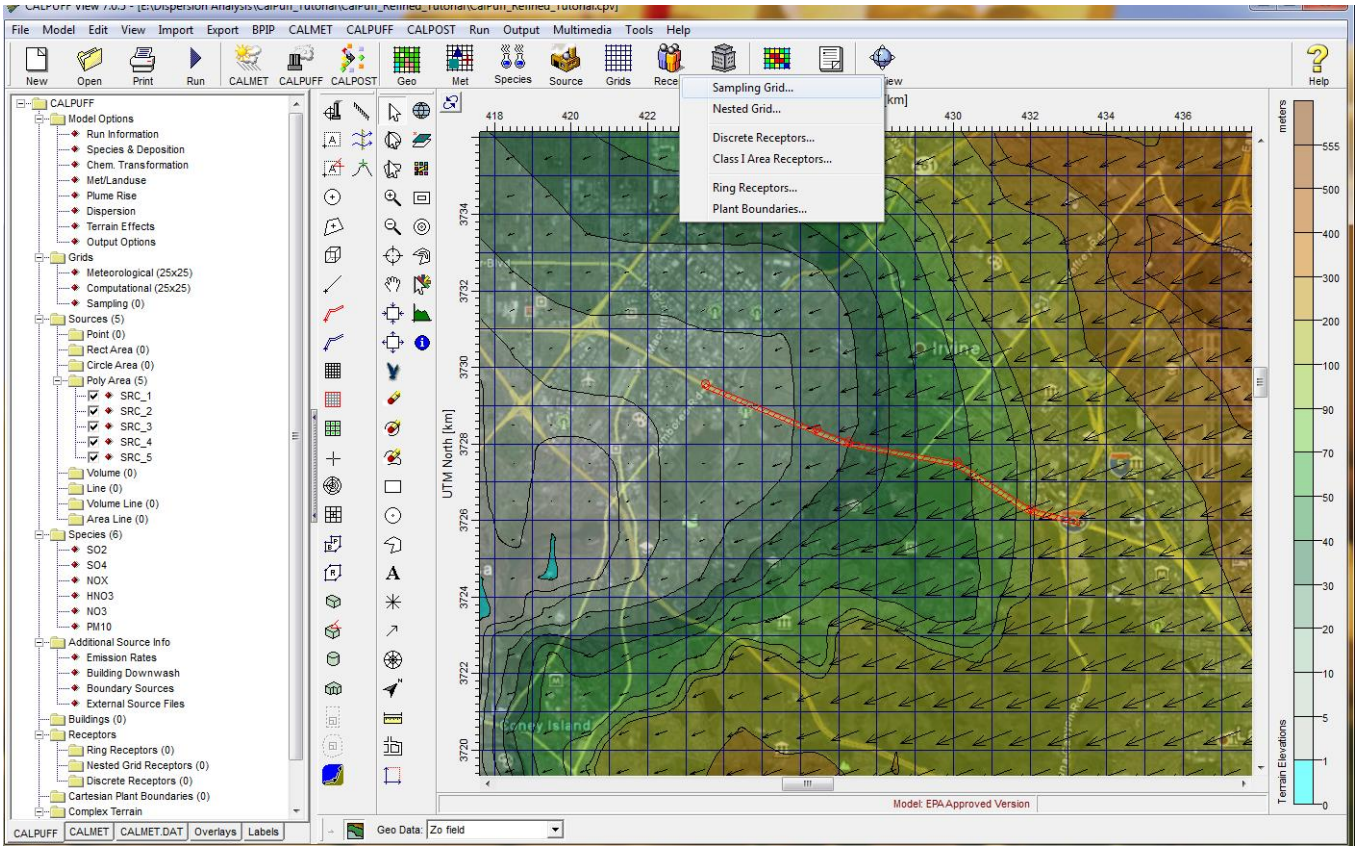
Click on the **Recept** menu toolbar button and then select from the pop-up menu the **Class I Area Receptors** option.



Specifying Sampling Grid

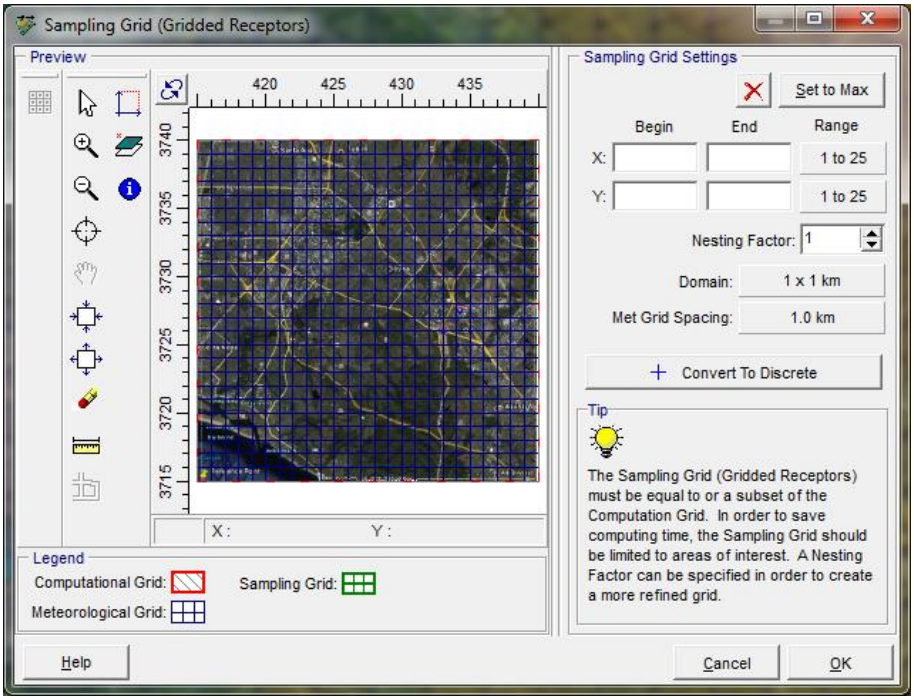
For the **Refined Tutorial**, in addition to the **Class I Area Receptors**, we will also specify the **Sampling Grid**. Although not necessary for the Class I Area analysis, the sampling grid will automatically generate one receptor location at the center of each met grid cell allowing you to visualize the results not only at the national park area (covered by the discrete receptors imported in the previous section) but also throughout the entire met grid domain.

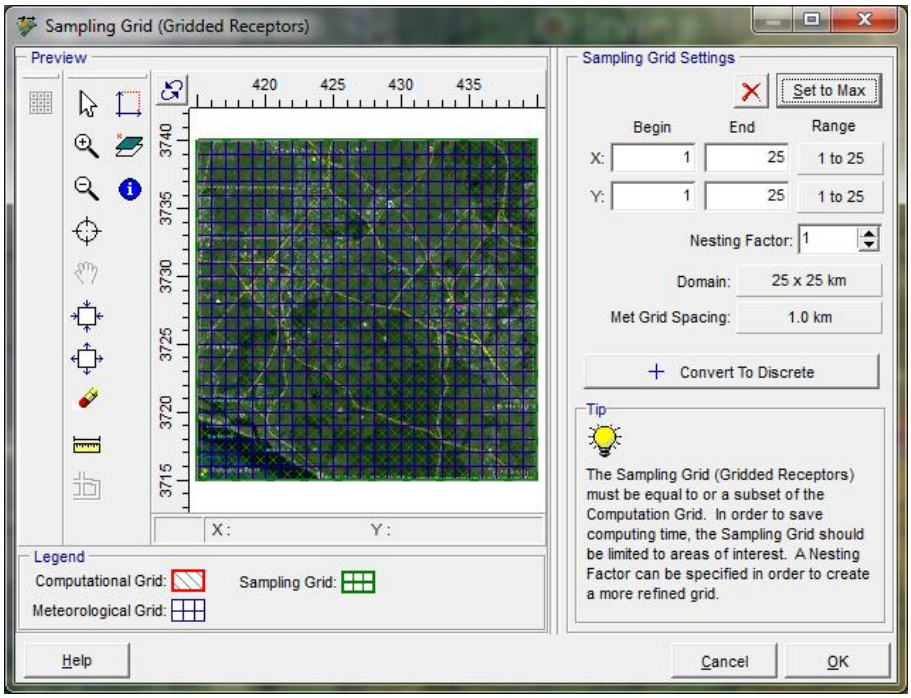
Click on the **Recept** menu toolbar button and then select from the pop-up menu the **Class I Area Receptors** option.



Parameter	Value
Grid Settings	Press Set to Max to assign receptors over the entire met grid domain.
Nesting Factor	1 **Tip:** A nesting factor of 1 will place 1 receptor at the center of each met grid cell. For this case study, since the met cell size is 4km, receptors will be placed 4 km from each other.

Met cell size is 1 km, receptors will be placed 1 km from each other.



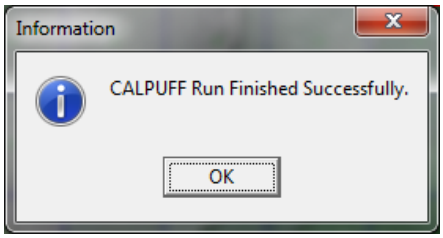


Click the **OK** button to close the **Sampling Grid** dialog.

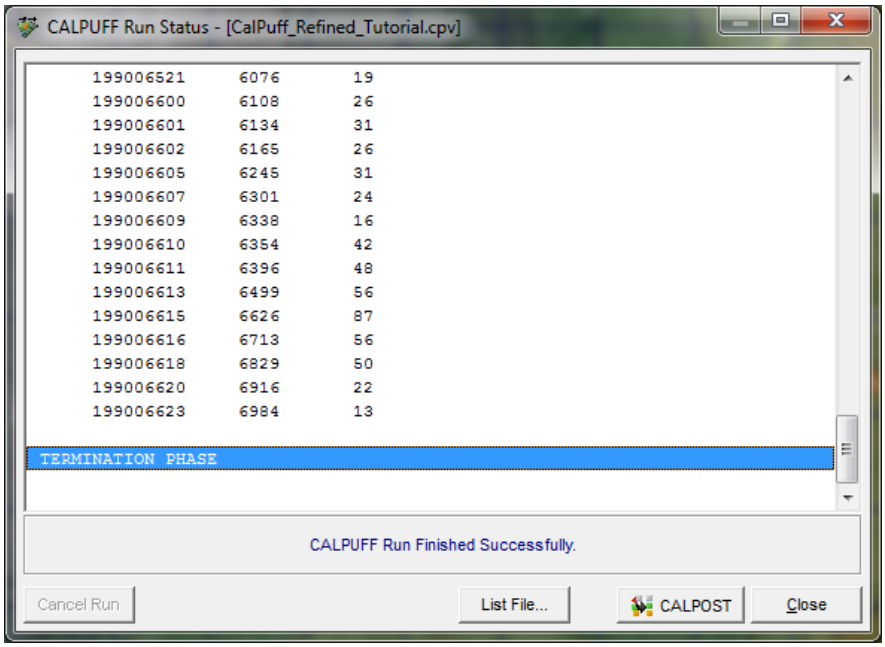
CALPUFF Model Run

Press the **Run** button and the select **CALPUFF Model** from the pop up menu.

When the CALPUFF model finishes running, you will see the message "CALPUFF Run Finished Successfully". Press the **OK** button.



Under the **CALPUFF Run Status** dialog, press the **CALPOST** button. The **CALPOST Options** dialog is displayed.

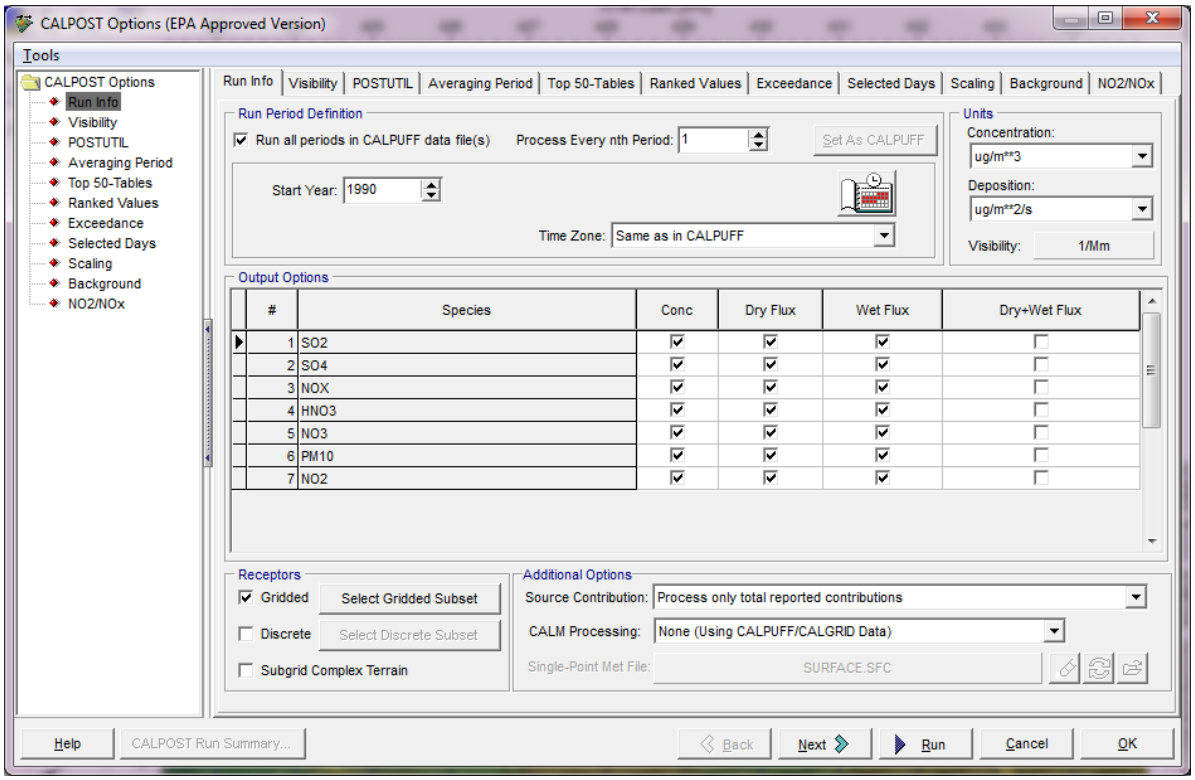


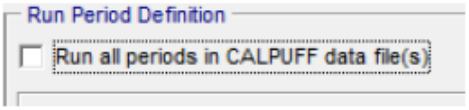
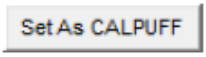
The CALPUFF model run generates output files in binary format which contains results for all the hours of the run period. Concentration results will be output to the CONC.DAT binary file, dry deposition results are output to the DFLX.DAT, etc. These files will need to be post-processed by the CALPOST model in order to get output results that can be viewed and plotted.

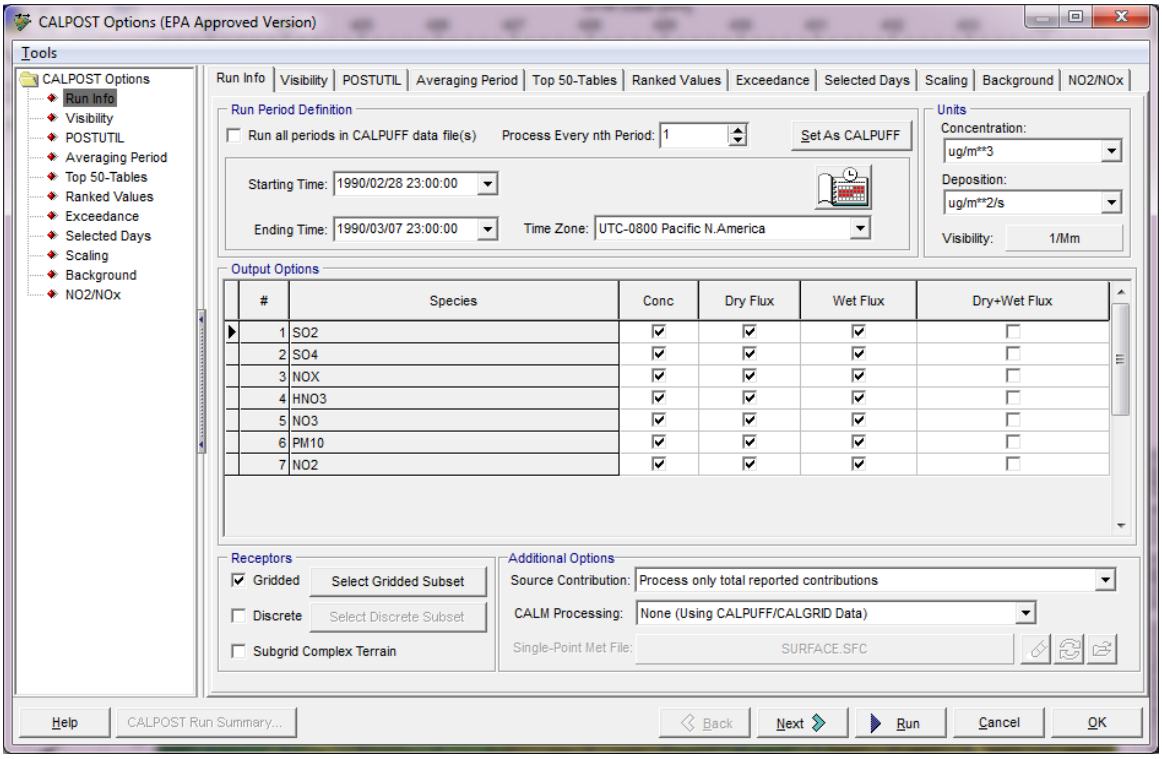
CALPOST Options

Make sure the **CALPOST Options** dialog is open. To open the **CALPOST Options**, press the **CALPOST** menu toolbar button.

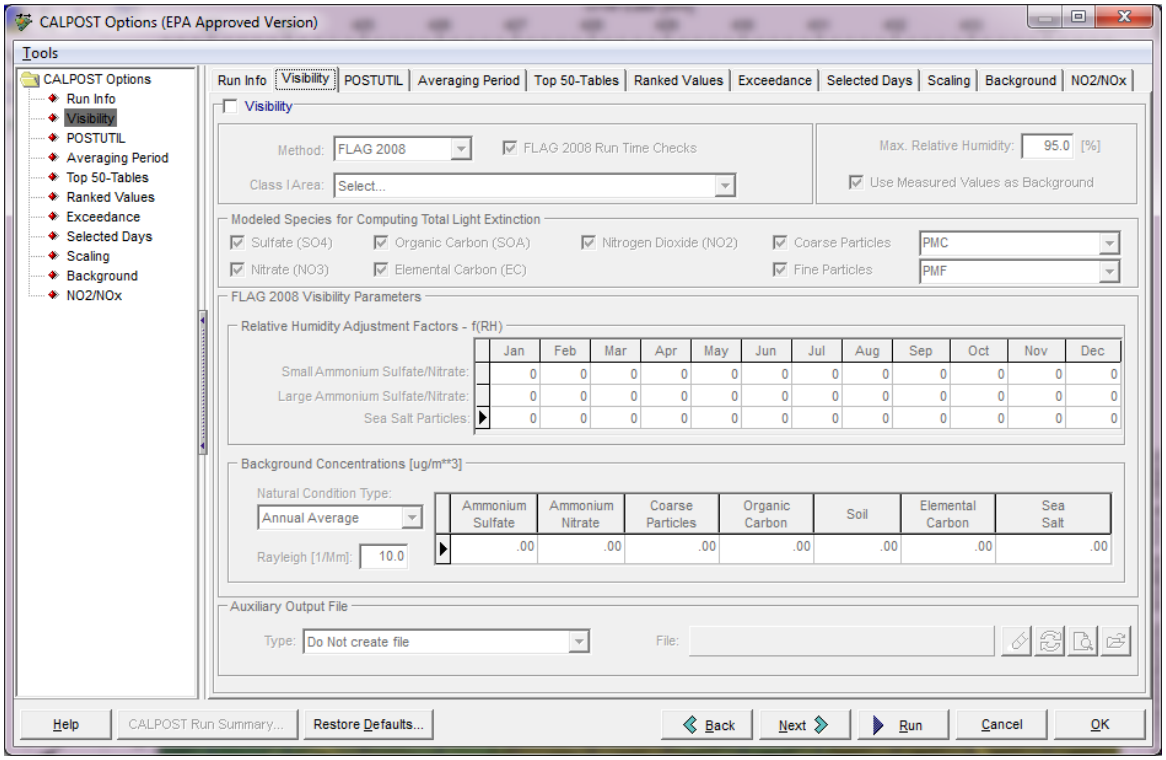
Run Info



Parameter	Value
Run Period Definition	Uncheck the box for the **Run all period in CALPUFF data file(s)** option.  **Tip:** Press the  button to automatically fill in all the run period definition information as it was setup in CALPUFF.
Process Every nth Period	1
Starting Time	1990/08/30 00:00:00
Ending Time	1990/09/02 00:00:00



Make sure the **Conc** (concentration) **Dry Flux** (dry deposition), and **Wet Flux** (wet deposition) options are selected for all species.

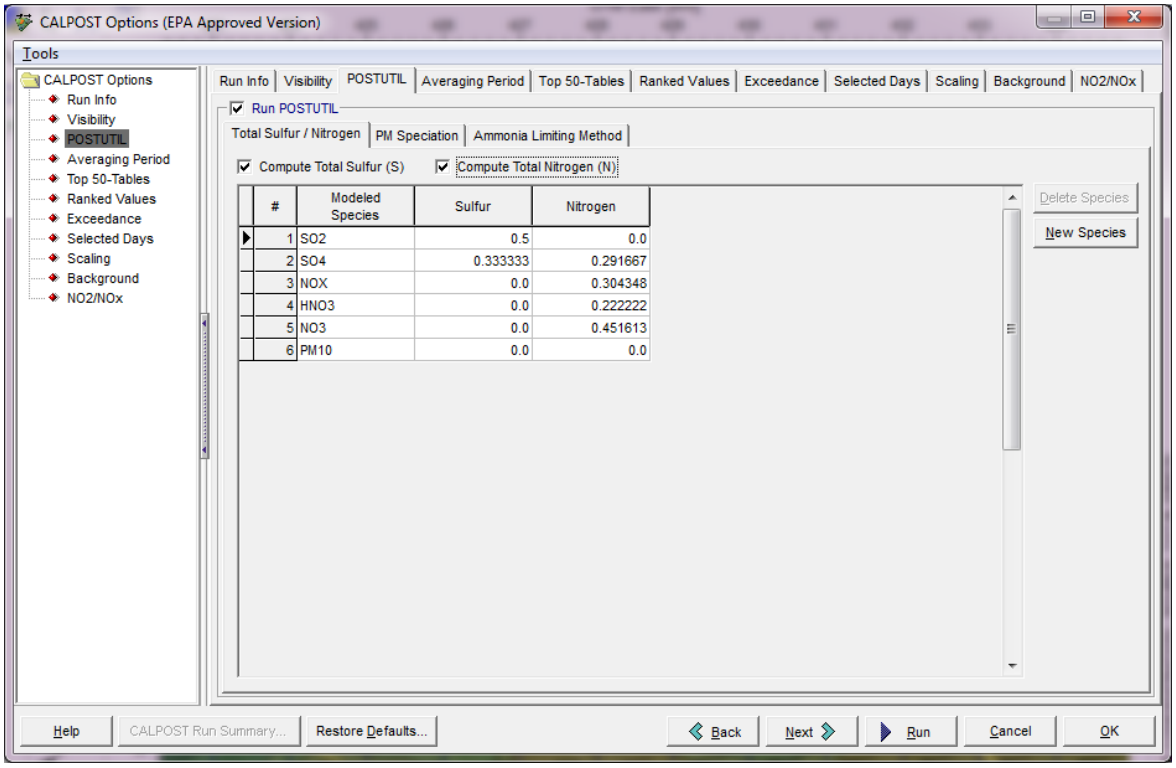


Parameter	Value
Run POSTUTIL	☑
Compute Total Sulfur (S) *	☑
Compute Total Nitrogen (N) *	☑

PM Speciation tab - Run PM Speciation	☐ No
Ammonia Limiting Method tab - Recompute HNO3/NO3 partition by ALM	☐ No

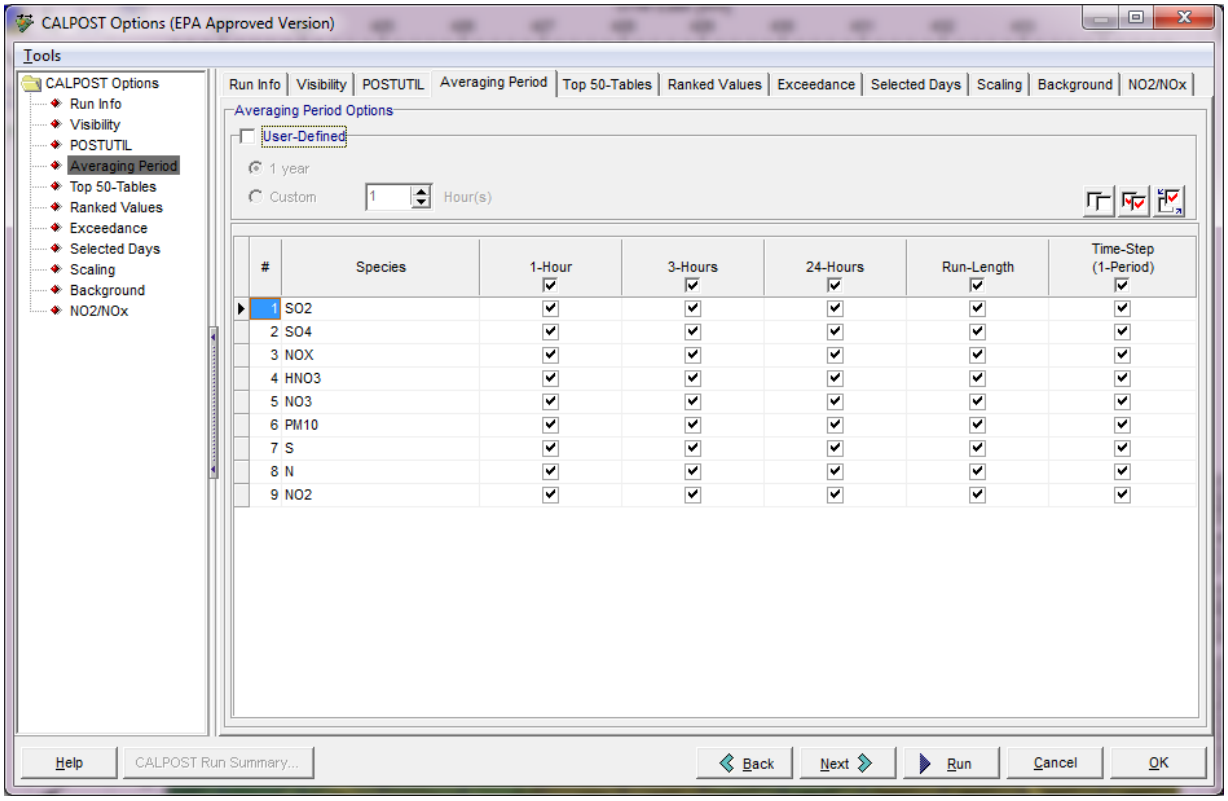
Tip*: This will add two new species to your species list (S and N) under the **Run Info** tab, **Output Options** table.

When producing either a summed wet and dry deposition flux or calculating a new species as a weighted combination of existing species, the starting date, time and run length must be stated explicitly.

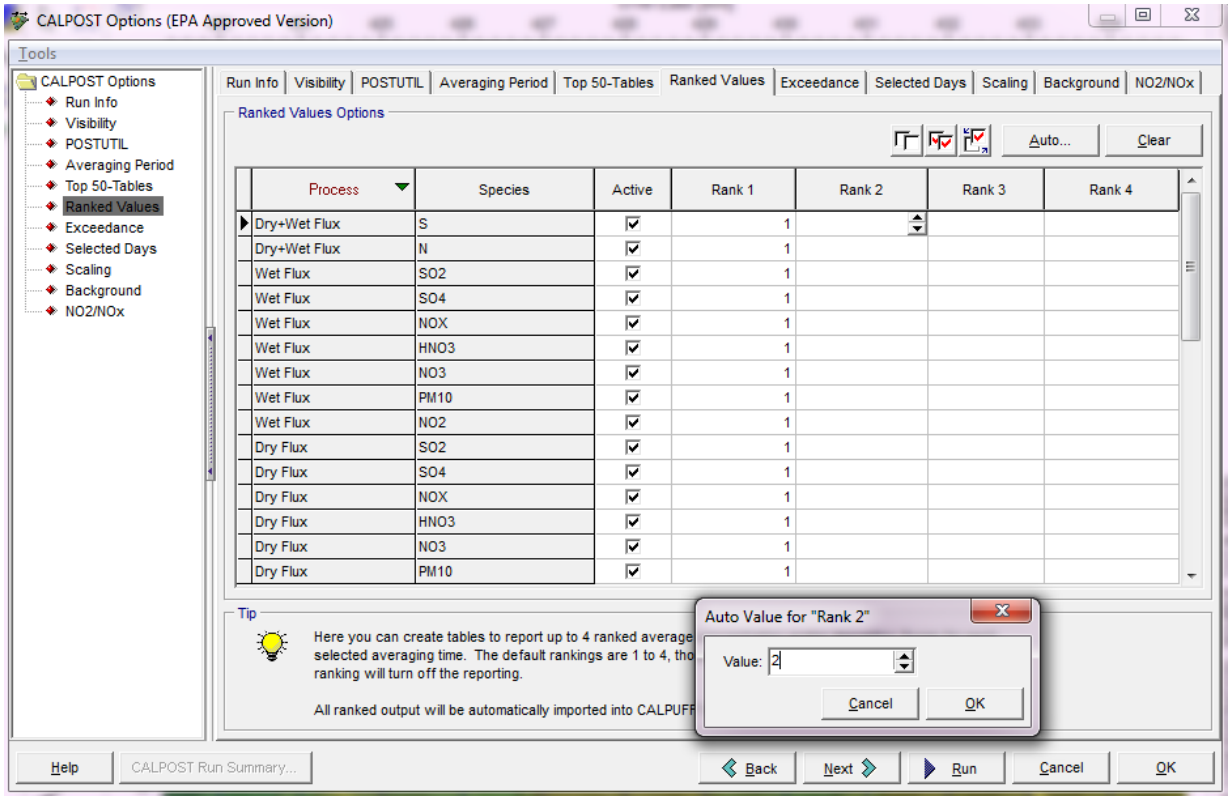


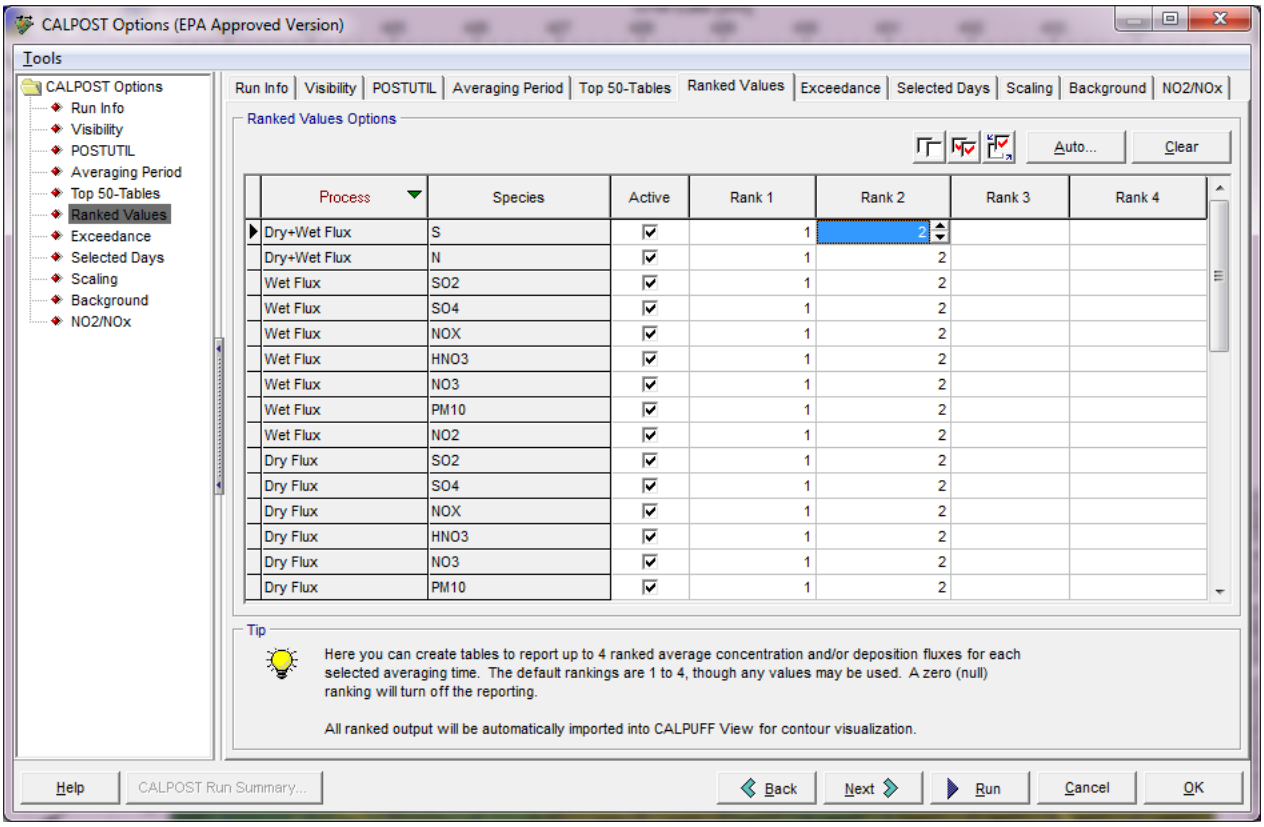
POSTUTIL is a utility that is used to consolidate the deposition fluxes for each species (dry + wet = total). POSTUTIL also can convert sulfate and nitrate fluxes to the total sulfur and total nitrogen fluxes.

Under the **Averaging Period** tab, leave the 1-Hour, 3-Hour, 24-Hour, Run-Length, and 1-Period averaging periods selected for all species.

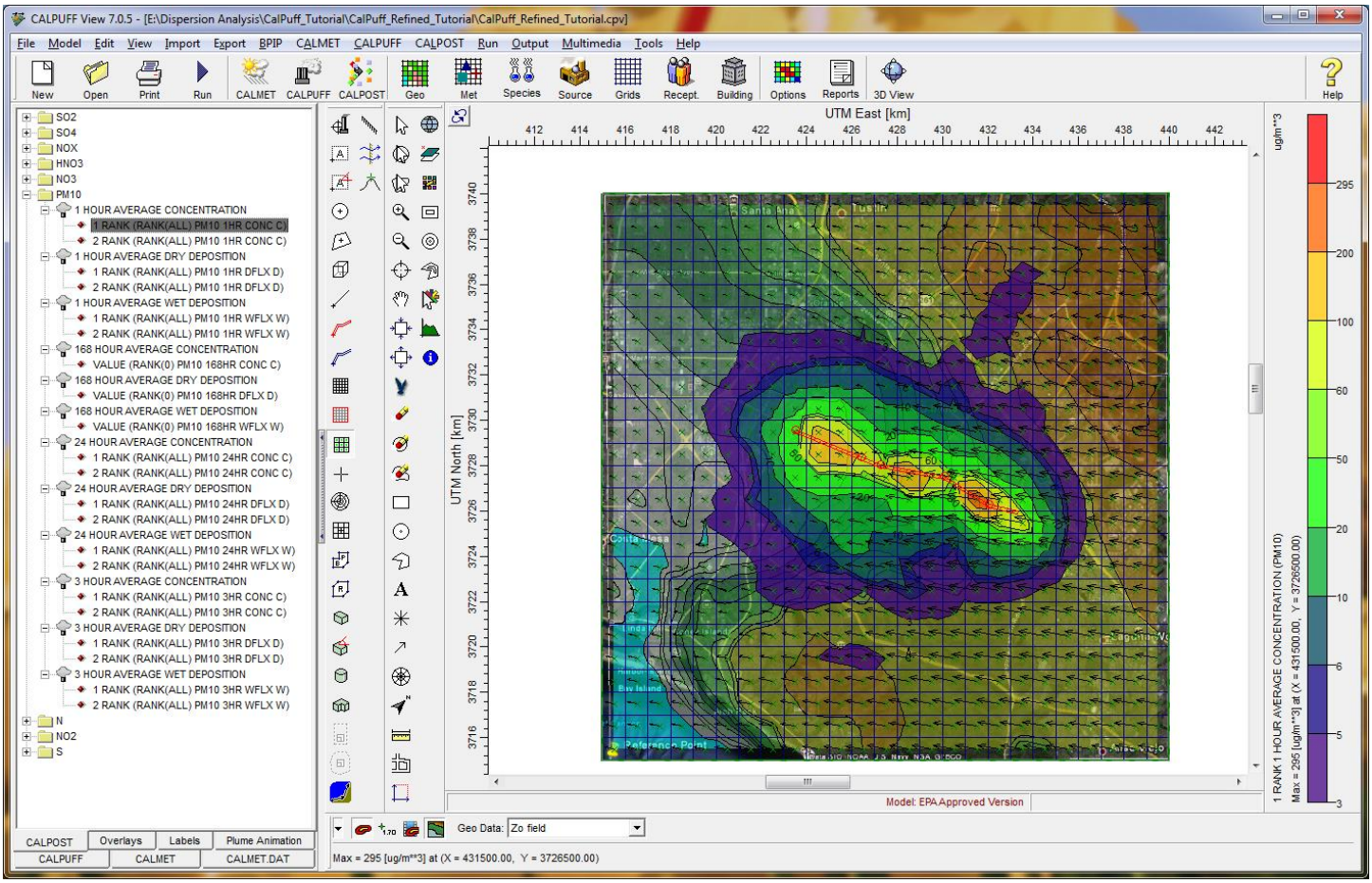


Under the **Ranked Values** tab, note that **Rank 1** is automatically selected as default for all specifies. Press the **Rank 2** column header and select **Auto** from the pop-up menu. Specify the value **2** under the dialog that is displayed and press the **OK** button. This will automatically assigned **Rank 2** for all species.





After you specify all the parameters under the CALPOST Options dialog, press the **Run** button.



After running CALPOST, check your results by looking into the contour plots displayed in the drawing area. Contours are displayed by averaging period for each pollutant. You can see the list of available contour plots by clicking on the CALPOST tab located at the bottom of the Tree View.

You can also visualize model results in a table format by pressing the **Reports** menu toolbar button. Click **Results Summary** in the **Reports** tree and then click **Finish** in **Advanced Filter** dialog to display the results. You can view and print the generated report. Close the **Reports** window when you are finished.