Model - Visualization and Analysis Tool (Model-VAT) User's Manual

Software version 1.7

December 2019

Acknowledgements

In cooperation with the U.S. EPA and the University of North Carolina Institute for the Environment (UNC-IE), the South China University of Technology (SCUT) developed the Model-VAT software. They provided much of the content for this User's Guide through VERDI User's Manual.

Model-VAT User's Manual Contents

Table of Figure

1 Introduction

1.1 Overview of Model-VAT

This manual describes how to use the Model - Visualization and Analysis Tool (Model-VAT). Model-VAT is a flexible and modular visualization software tool that allows users to visualize multivariate gridded environmental datasets created by environmental modeling systems such as the Community Multiscale Air Quality (CMAQ) modeling system and the Weather Research and Forecasting (WRF) modeling system. These modeling systems produce files of gridded concentration, deposition fields, and meteorological parameters that users need to visualize and compare with observational data both spatially and temporally. Model-VAT can facilitate these types of analyses.

This manual is divided into ten chapters that provide the user with instructions on how to use Model-VAT and explain the application of the software showing examples for its use.

1.2 Computer Requirements

The computer hardware requirements for Model-VAT are typically modest, though this will vary depending on the complexity of the analysis. Model-VAT 1.7 is distributed as an EXE file, as appropriate, for each of the following supported platforms:

- >=32-bit Windows 7.
- >=64-bit Windows 7.

Model-VAT works best in a 64-bit Windows environment. With a 32-bit installation there are limits on the memory available to the software application; it can utilize no more than 2

GB of RAM. This will impact performance when processing large spatial datasets. A solid-state drive (SSD) has also shown improved performance over hard disk drives (HDD).

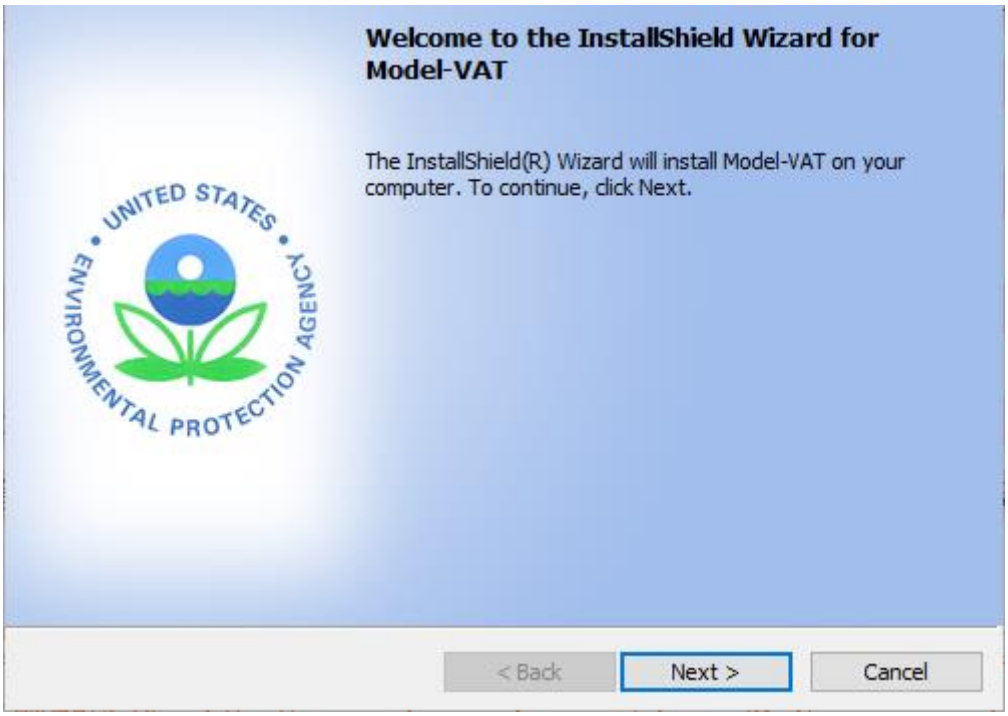
1.3 Model-VAT Installation Instructions

1.3.1 Installing Model-VAT

Download Model-VAT Software Package from the ABaCAS website. This tool and corresponding example data are available for registered users at this website: http://www.abacas-dss.com/abacas/Software.aspx. The installation of Model-VAT is very simple. Double click **Setup.exe** in your installation directory to bring up the setup wizard. Then follow the setup wizard by clicking **Next** or **Finish** to complete the installation (Figure 1.3-1).

To install Model-VAT for Windows, unzip the file to a local directory on your Windows 10 computer. If you are unable to install Model-VAT on your computer, please check to see whether your user account is authorized to install software. You may need to request that a user with a computer administrator account install Model-VAT, or provide you with an account that has permission to install software. For more information about user account types, click Start and select Control Panel and then click on the User Account icon.

NOTE: You do not need to install Model-VAT under a Program Files directory or in the root directory on one of your hard disk drives. Therefore, you should not need Administrator rights to install Model-VAT. If your system is under strict control from your Administrator, you may be able to unzip the Model-VAT distribution under your documents directory; however, you may have problems with a space in your path.



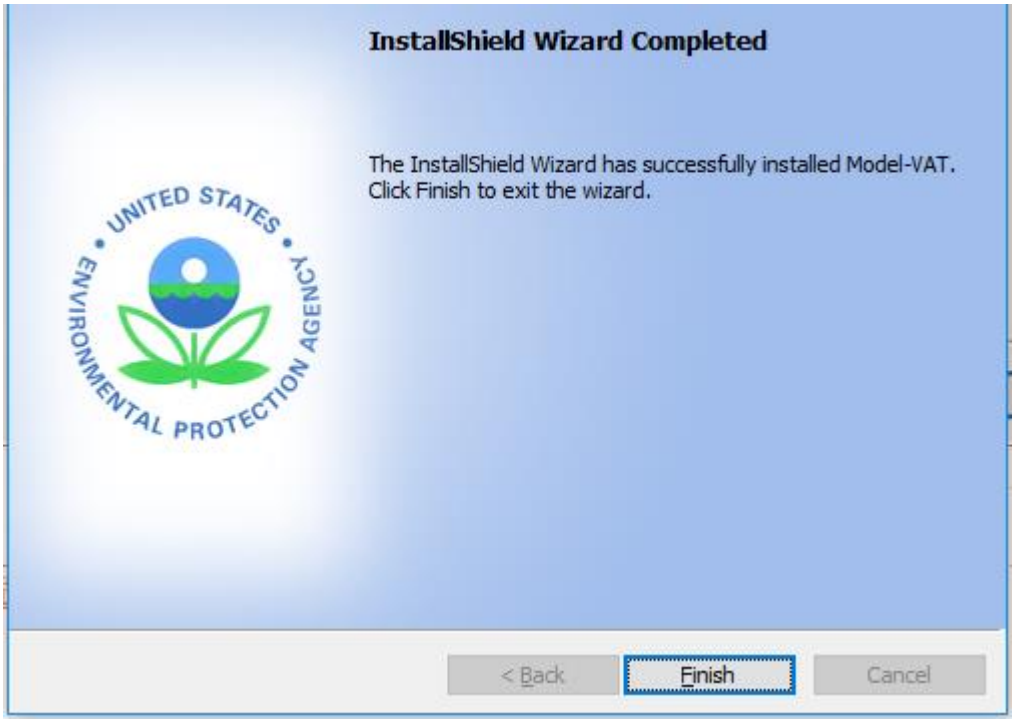


Figure 1.3-1. Install Model-VAT

1.3.2 Uninstalling Model-VAT

To uninstall Model-VAT, go to **Control Panel, Programs and Features,** and remove **Model-VAT.** Note that uninstalling Model-VAT does not also remove any results files that you have created with Model-VAT.

2 Terminology

This chapter explains common terms used in this user s manual and in the model, and references, where possible, other sections in this manual to find more detailed information.

Community Multi-scale Air Quality (CMAQ) Model. An open-source photochemical grid air quality model that the U.S. EPA and others rely upon to predict levels and changes in pollutant concentrations.

Grid Cell. One of the many geographic, or spatial, components within a Grid Definition. These can be regularly or irregularly shaped.

Lat/Long. Latitude and longitude information to specify the geographic coordinates of a spatial location. The CMAQ model data are usually provided for each grid cell identified by the latitude and longitude of the grid cell's center point. Latitude identifies the north-to-south location of a point on the Earth. Longitude identifies the east to west location of a point on the Earth.

Layer. In GIS, a layer represents a logical separation of mapped data usually representing a theme.

Model Data. Pollutant concentration data that are generated by running an air quality model such as CMAQ. This is different from monitor data, which are based upon observed concentrations.

Monitor Data. Pollutant concentration data that are based upon measurements from an air quality monitor. Raw monitor data usually refers to data that are taken directly from measurement networks, with no additional processing of the data. Monitor data are different from model data, which are based upon numerical predictions from an air quality model.

Shapefile. A shapefile is a particular type of GIS file, and has a *.shp* extension. These files are accompanied by companion files with *.shx* and *.dbf* extensions, and can be used to create Shapefile Grid Definitions. See http://www.esri.com/library/whitepapers/pdfs/shapefile.pdf for more information.

3 Starting Model-VAT

3.1 Starting Model-VAT

3.1.1 Starting Model-VAT from standalone Model-VAT Software

If you have successfully installed standalone Model-VAT, you can double-click the 

icon from your computer desktop to open Model-VAT.

3.1.2 Starting Model-VAT from ABaCAS

If you have installed the ABaCAS software, you can also start Model-VAT from ABaCAS. There are two ways to open it from ABaCAS. One is to open it from ABaCAS Start Page (Figure 3.1-1) and the other one is to open it from ABaCAS Home Page (Figure 3.1-2).

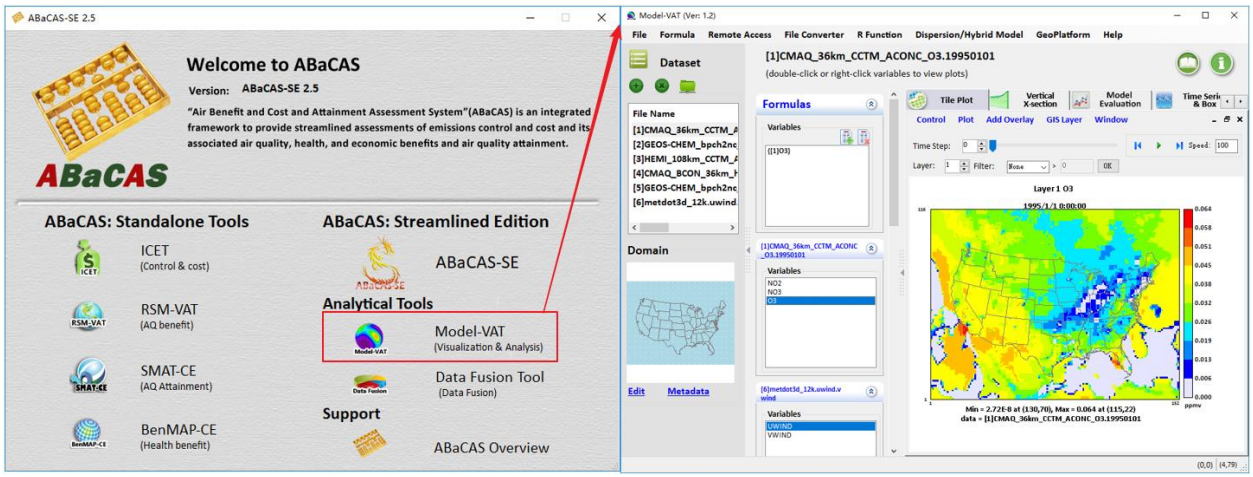


Figure 3.1-1. Open Model-VAT on Start Page

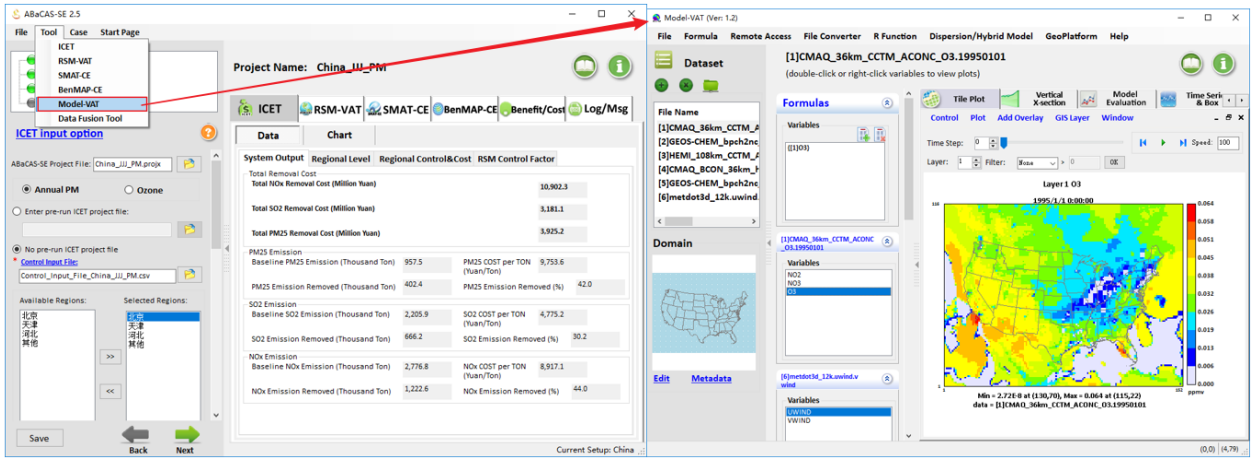


Figure 3.1-2. Open Model-VAT on Home Page

3.2 Main Window

When Model-VAT starts and the main window is displayed (Figure 3.2-1). The top of the main window contains a menu bar with the main window options (**File, Formula, Remote Access,**

File Convertor, R Function, Dispersion/Hybrid Model, GeoPlatform, and Help)(Details shown in Chapter 4).

Below the menu bar, the Model-VAT window is divided into two main areas: a parameters area consisting of **Dataset** and **Domain** on the left side and a plots area consisting of **Formula (**see Chapter 6**)** & **Variables** and **Plot** on the right side. You can resize the entire window with your mouse. You can also fold the parameters area by clicking the arrow on the dividing line between left and right part.

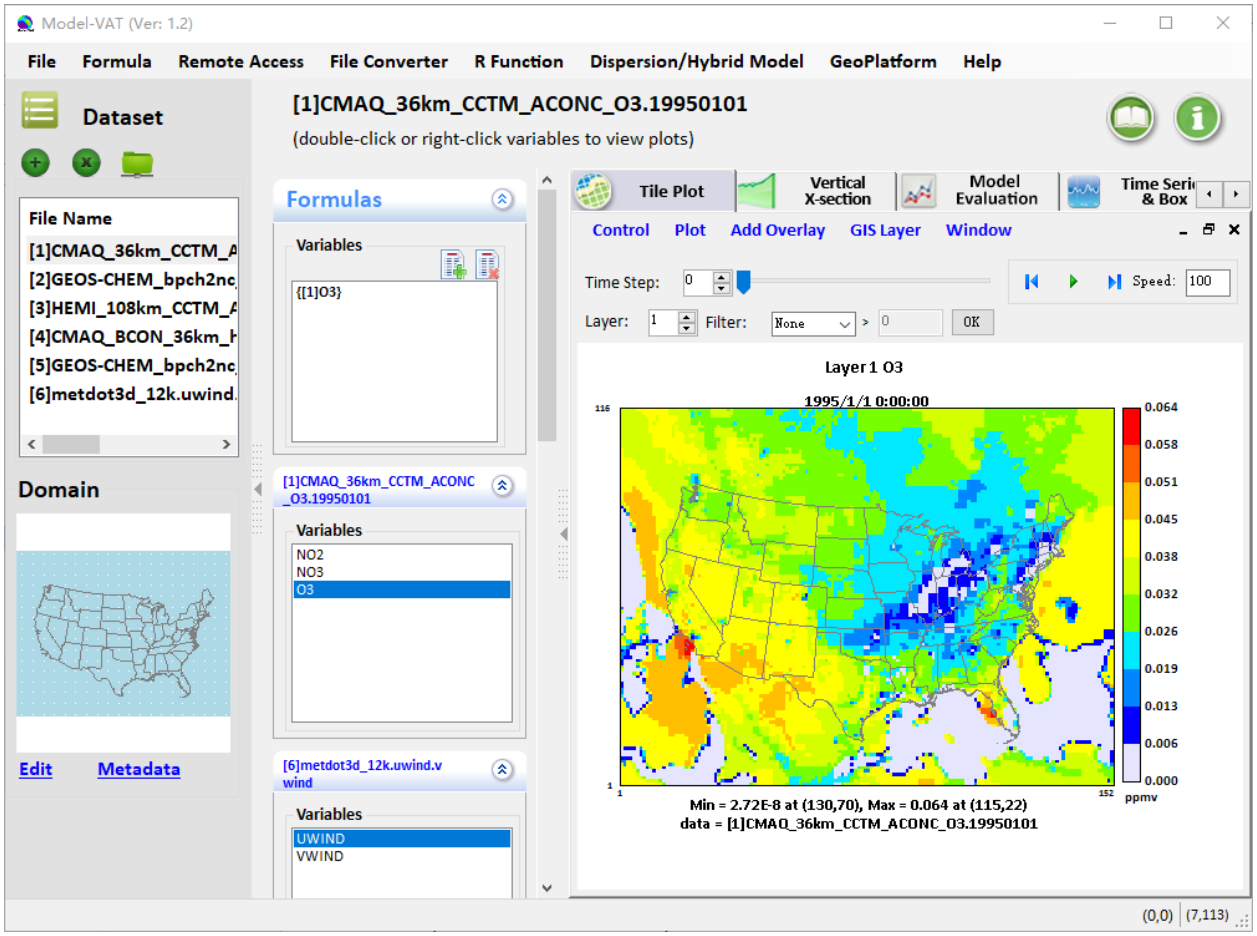


Figure 3.2-1. Model-VAT Main Window

The parameters area contains two parts:

- The **Dataset** is used to load in the dataset files that you want to work with in this session (see Chapter 5).

- Use the **Domain** (see Chapter 5) to set the region that the users want to see in **Tile Plot**.

The plot area contains two parts:

- **Formula & Variables**:

  The **Formula** is used to create a formula that refers to the variable and the dataset that you are interested in plotting (see Chapter 6). All plots in Model-VAT are generated from

formulas. A formula can be as simple as a single variable from one dataset or it can be an equation that uses variables from one or more datasets.

Once the datasets are loaded, Model-VAT automatically displays the lists of variables that are in the datasets. Variables of each loaded dataset are displayed on **Variables** area. Double-click on the name of a variable listed on the variables to add it as a tile plot on the Tile Plot pane.

- The **Plot** area is consisted of several plot panes, which are **Tile Plot**, **Vertical X-section**, **Time Series & Box**, **Scatter Plot**, **3-D Plot**, and **Data Output**. Users can see different kinds of plots according to their need (see Chapter 8).

4 Navigating Model-VAT s Main Menu Options

Table 4-1 shows the main menu options that are available on the top menu bar in Model-VAT's main window (Figure 3.2-1). These options are discussed in detail below.

Table 4-1. Model-VAT Main Menu Options

File	Formula	Remote Access	File Converter	R Function	Dispersion/Hybrid Model	GeoPlatform	Help
Open Project Save Project New Project Load demo project View Script Editor Options Exit			To CMAQ IO/API To BenMAP-CE input format To SMAT-CE input format				Model-VAT Help About

4.1 File

Click **File** to open the **File** list (Figure 4.1-1). Users can select different functions to achieve in five options,

Open Project, **Save Project**, **New Project**, **Load demo project**, **View Script Editor**

, **Options**, and **Exit**.

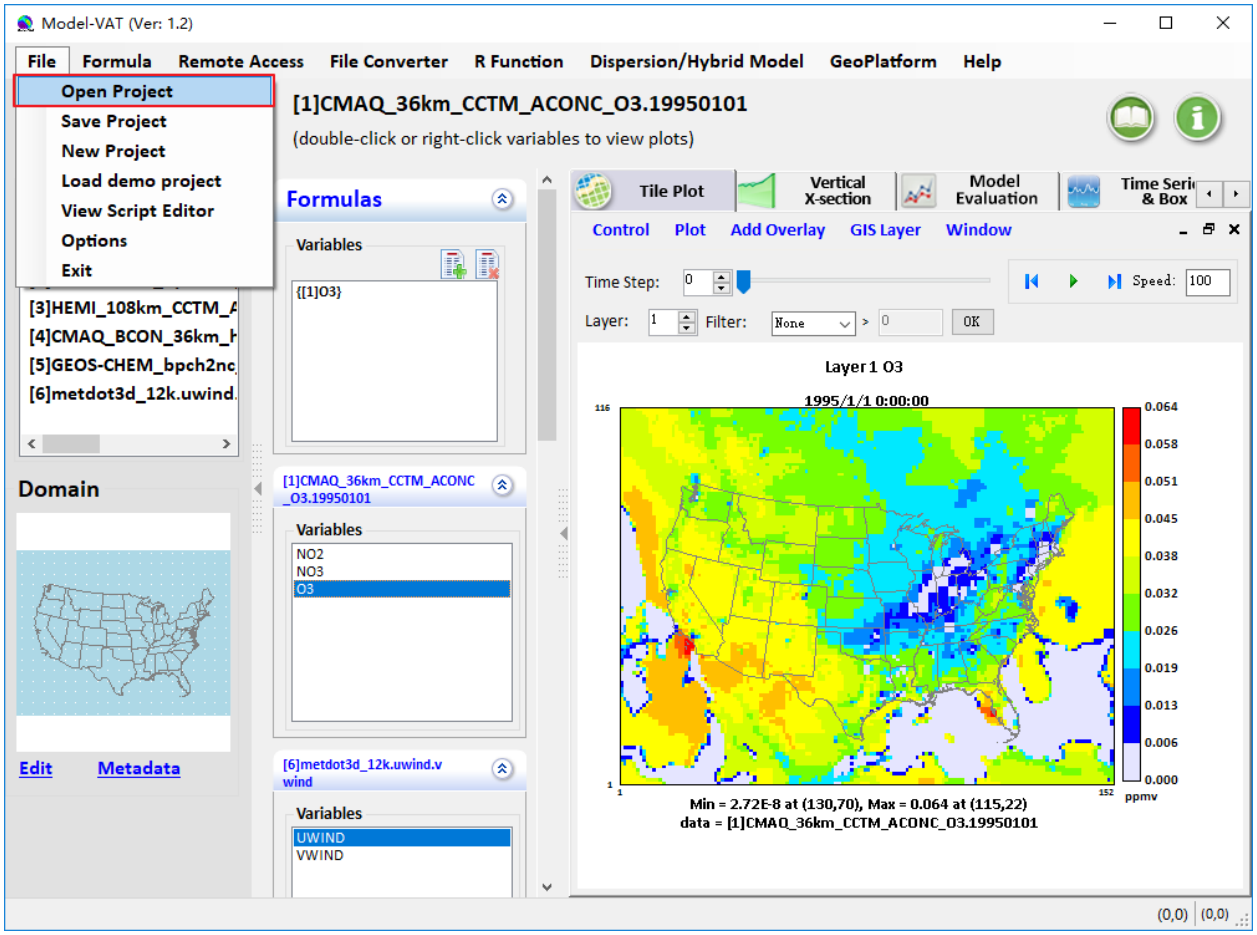


Figure 4.1-1. File List

- **Open Project**: Open an existing project in user's computer.
- **Save Project**: Overwrite the current project users are opening.
- **New Project**: Create a new project.
- **Load demo project**: Load the demo project we provided for users to be familiar with Model-VAT.
- **View Script Editor**: Batch generate graphics in the specified format.
- **Options**: Choose to show **Language**, including Chinese and English.
- **Exit**: Exit Model-VAT.

4.2 Formula

Click **Formula** to open the **Formula** window. Users can also click  to open it **(**see Chapter 6**)**.

4.3 Remote Access

For modeling users, they often need to write scripts to visualize/analysis (V/A) their model output data or download them to local PC from a remote system (e.g., Linux system) for further study. As we known, modeling output data often contain many variables and are usually very large in size. However, what users are interested in often is just one or two variables.

Therefore, no matter writing scripts or downloading the whole model output file especially the large file can be quite cumbersome and inefficient for data analysis. To address these problems, we developed this module for users to easily extract and visualize/analysis their desired subset of data directly from their remote system.

Click **Remote Access** to open the **Remote Access** window (Figure 4.3-1). This part provides three functions: recent sites, site management, and local/download files (Figure 4.3-1).

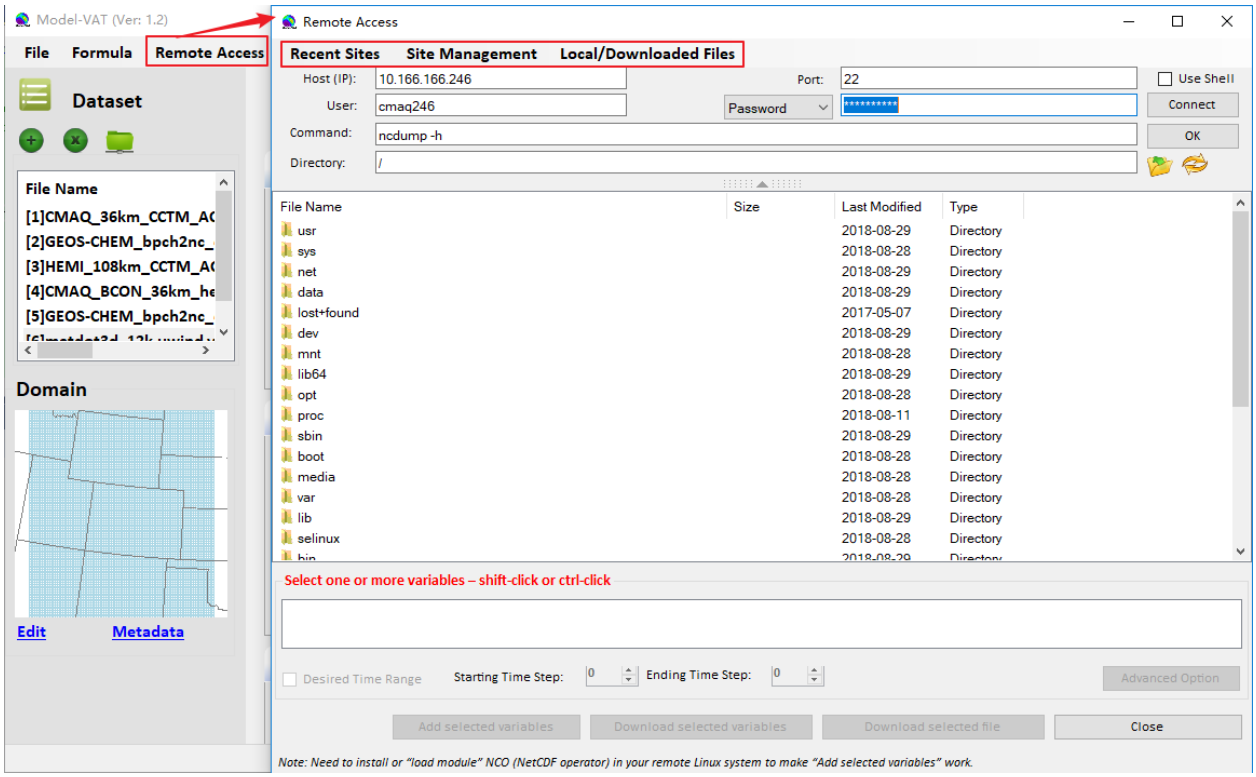


Figure 4.3-1. Remote Access Window

- **Recent Sites:** Record sites that have been connected and saved.
- **Site Management** will open when users click **Site Management** (Figure 4.3-3)**.**
- **Local/Downloaded Files** will open when users click **Local/Downloaded Files** (Figure 4.3-4).
- **Desired Time Range**: Users can select time range when users check this option (Figure 4.3-2).
- **Add selected variables**: Add selected variables to main window (Figure 4.3-2).
- **Download selected variables**: Download the variables users are selecting.
- **Download selected file**: Download and add the file users are selecting to main window. And users can download multiple files at the same time.
- **Close**: Close this window. After closing, it will remember the previous setup and resume this setup next time when **Remote Access** is launched again.

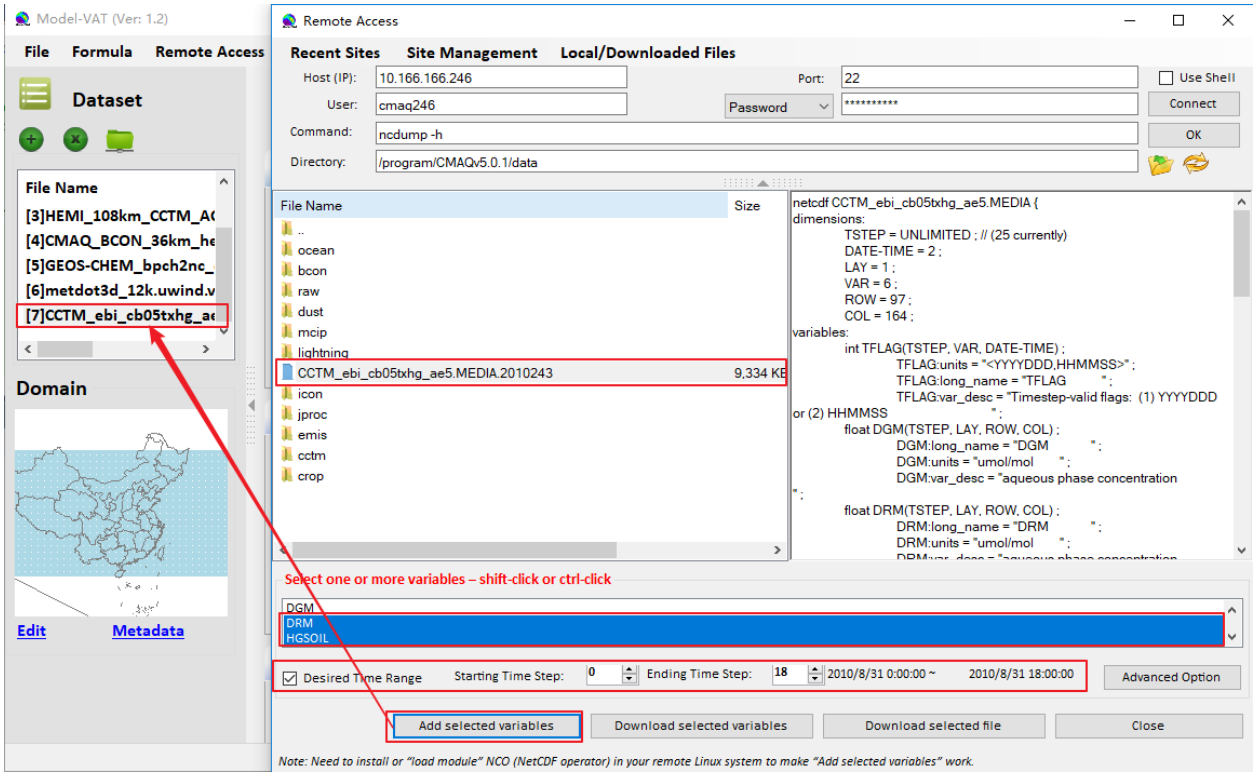


Figure 4.3-2. Add the File to Main Window

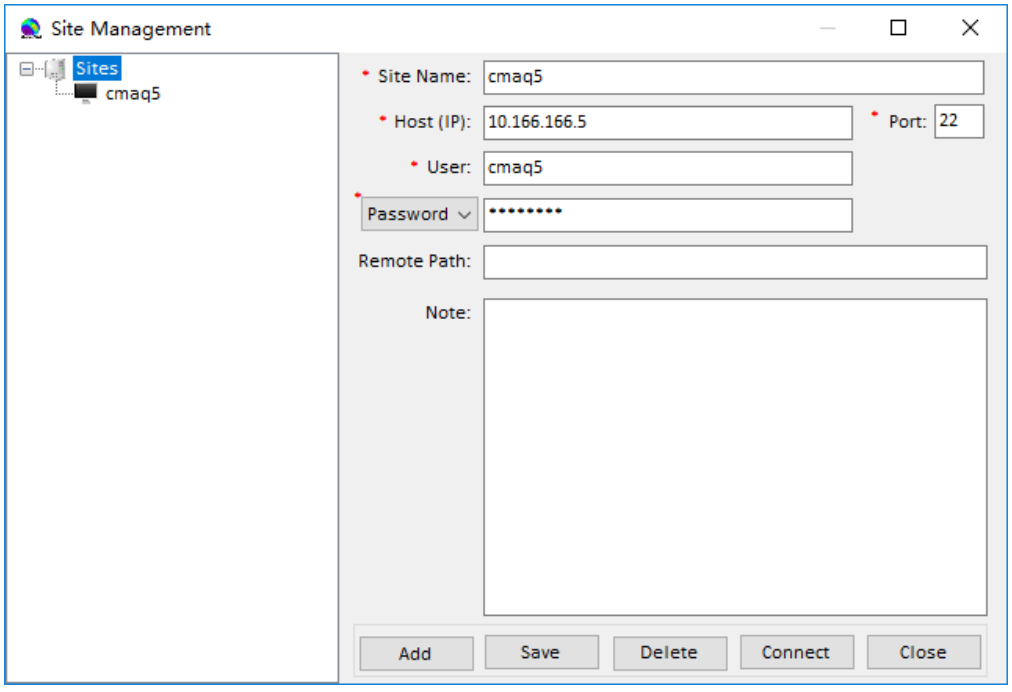


Figure 4.3-3. Site Management Window

- **Add**: Add a new site.
- **Save**: Overwrite the old site.
- **Delete**: Delete the old site.
- **Connect**: Connect the account.
- **Close**: Close the **Site Management** window.

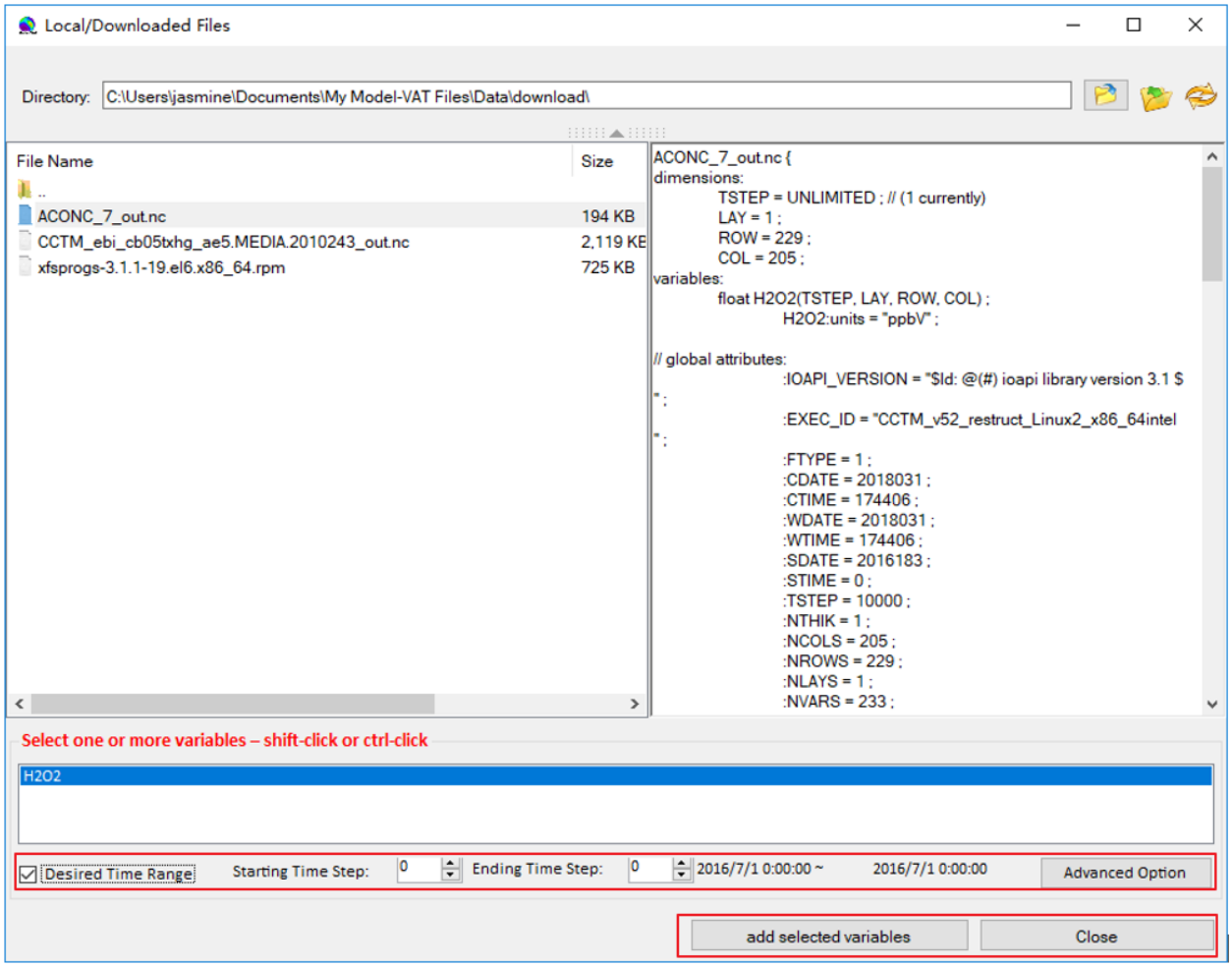


Figure 4.3-4. Local/Downloaded Files Window

- **Desired Time Range**: Users can select time range when users check this option (Figure 4.3-4).
- **Add selected variables**: Add the variables users are selecting to main window (Figure 4.3-4).
- **Close**: Close this window.

4.4 File Converter

Click **File Converter** to open the **File Converter** window. Users can convert different files to different file output format as needed.

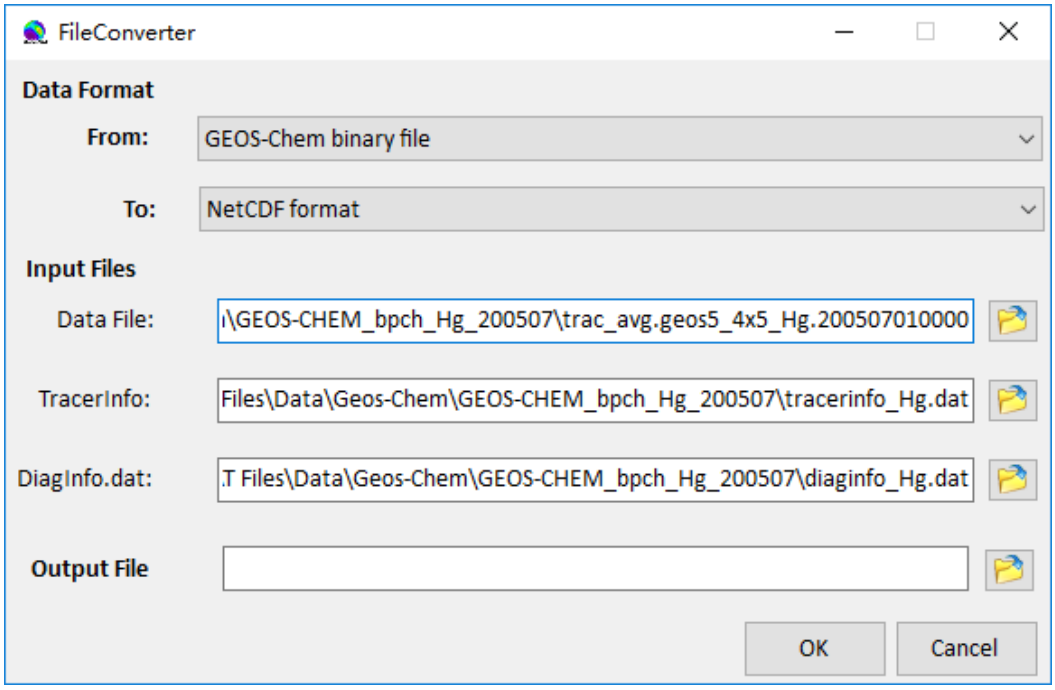


Figure 4.4-1. File Converter Window

Data Format:

Select original format and new format. (This version only supports convert GEOS-Chem binary file to NetCDF format) (Figure 4.4-1).

Input Files:

Data File: Select the file users want to convert;

TracerInfo: TracerInfo.dat file contains name weight and index information about GEOS-CHEM tracers;

DiagInfo.dat: DiagInfo.dat file contains category names and the offsets which they are stored in file tracerinfo.dat.

Output File:

Select converted file path.

4.5 R Function

Click **R Function** to open the **R Function** window (Figure 4.5-1). After loading an R file or writing an R script, Model-VAT can realize those and execute. And the R plot drawn by R script function can be saved.

Figure 4.5-1. R Function Window

- **Load Demo Project**: Load the built-in demo project.
- **Open Project**: Open an existing project in the user s computer.
- **Load R Script**: Load a built-in example script we provided.
- **Save R Script**: Overwrite the old R script.
- **Exit**: Exit the **R Function**.
- **Run**: Run the R script.

There are some notes and examples to help users to edit the R script:

(1) The single variable must begin with '{' and end with '}'. The formula must begin with '@' and end with '@'. For instance, {[2]IJ-AVG-S__BCPI} is identified as a single variable, @{[2]IJ-AVG-S__BCPI}+{[2]IJ-AVG-S__OCPI}@ is identified as a formula. But if users want to draw a single variable, users should also put it enclosed with @, like this, @ {[2] IJ-AVG-S__BCPI} @.

(2) For drawing bar chart, the strings #Bar Chart#, as a tag, must be written on the first line of the script.

(3) In this version, it just can plot the data of chosen variables in layer 1 and time-step 1. But for drawing bar chart, it will draw the Time Series Plot of layer 1. In the future, we will make it flexible.

Figure 4.5-2, Figure 4.5-3, Figure 4.5-4, and Figure 4.5-5 show some examples of R-function graphics produced by Model-VAT.

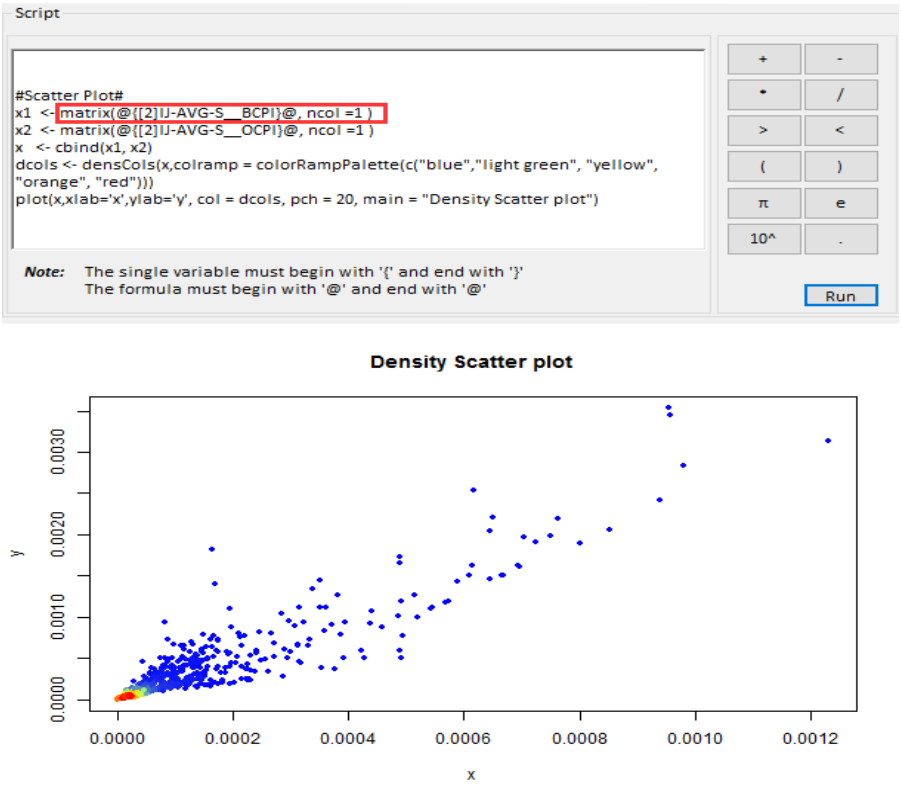


Figure 4.5-2. Scatter Plot

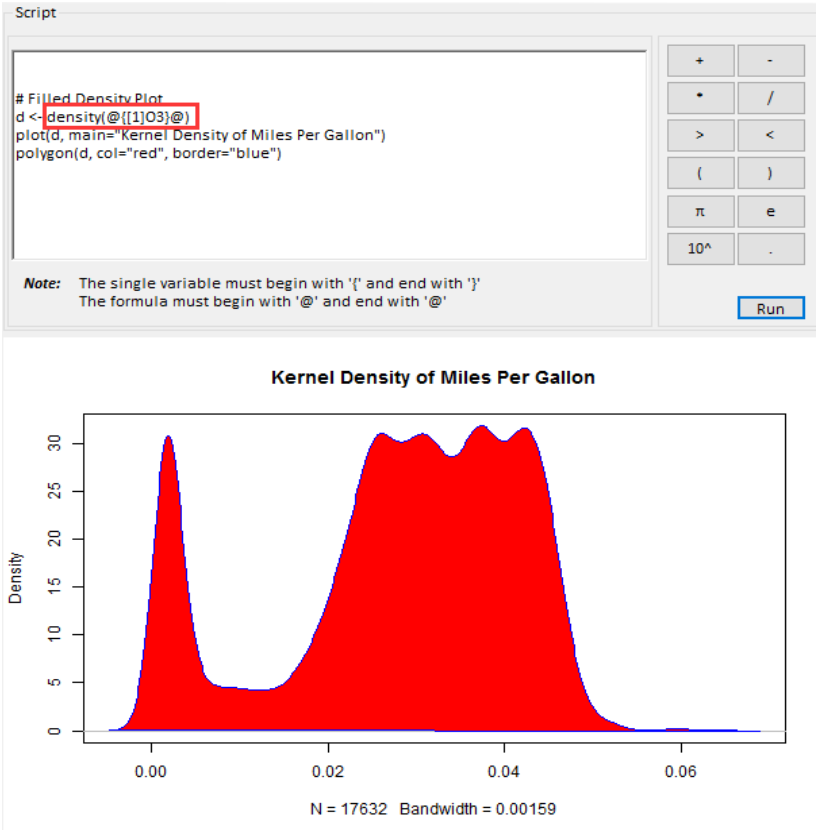


Figure 4.5-3. Density Plot

Figure 4.5-4. Bar Chart 1

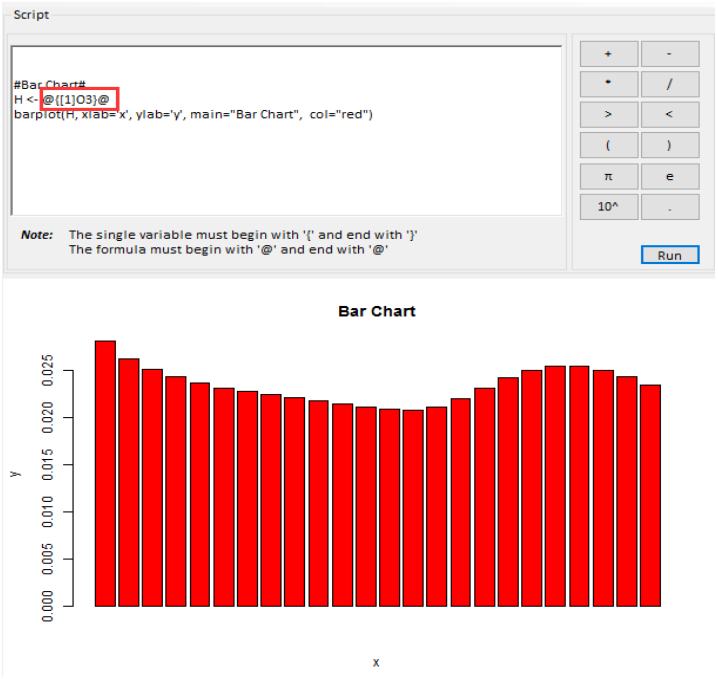


Figure 4.5-5. Bar Chart 2

4.6 Dispersion/Hybrid Model

Click **Dispersion/Hybrid Model** to open **Dispersion/Hybrid Model** window. There are three models for users to check: **AERMOD**, **SCICHEM**, and **AERMOD and CMAQ** (Figure 4.6-1). Users can check one of these models, then click **Run** to draw the responding plot. The **Save** option is under development.

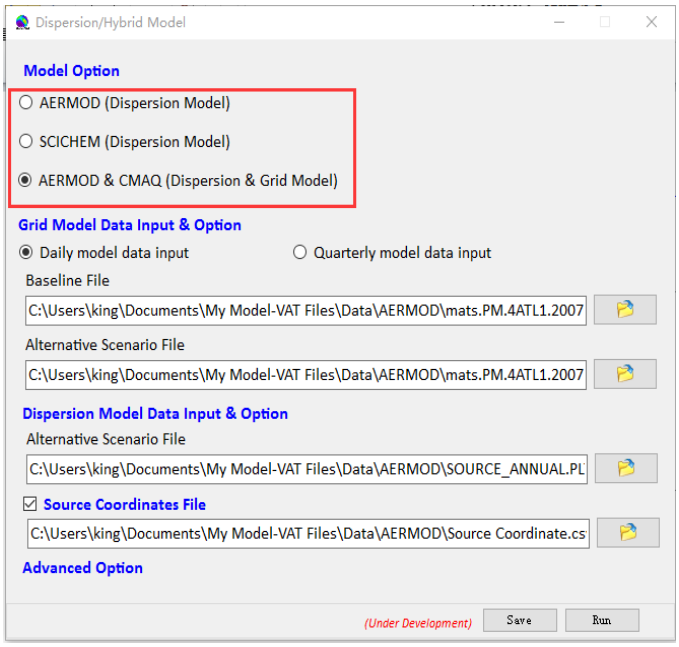


Figure 4.6-1. Dispersion/Hybrid Model Window

AERMOD

When users check the **AERMOD**, users will see the window as show as Figure 4.6-2, and users can check or uncheck **Source Coordinates File**.

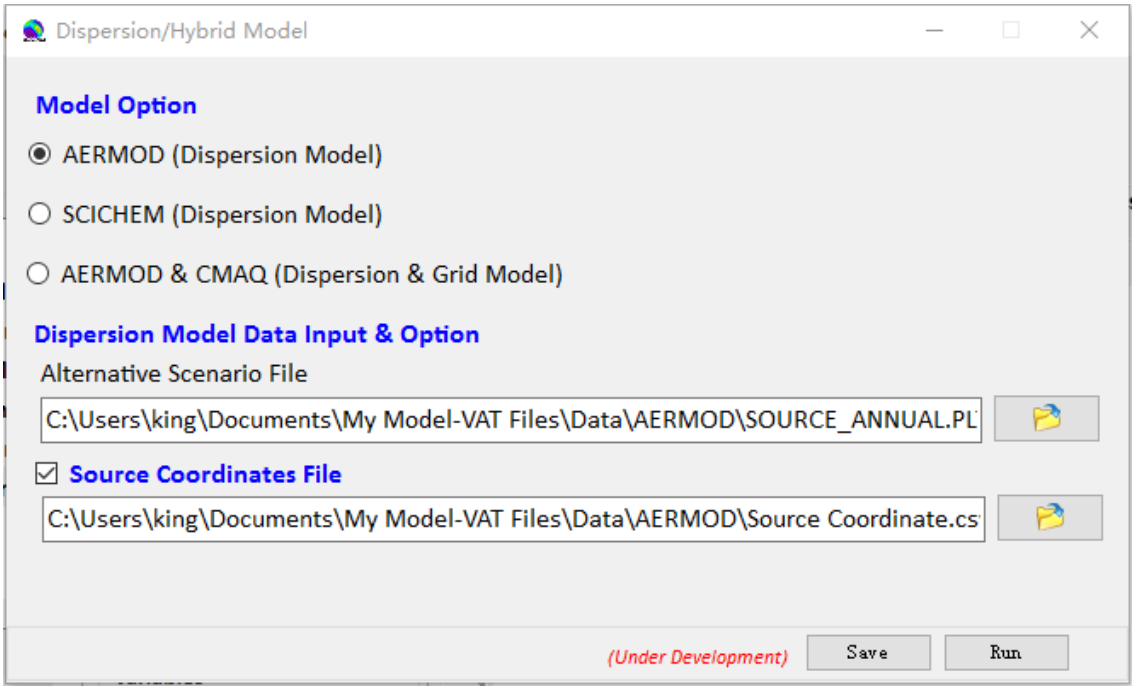


Figure 4.6-2. AERMOD Window

Then click **Run** to run the model, there is an AERMOD Plot window been opened as show as Figure 4.6-3 or Figure 4.6-4. There are two types of algorithm to draw AERMOD:

Coarse-3D: Using the biggest grid cell.

Actual-2D: Using its actual data (including small and big grid cell), only 2D plot.

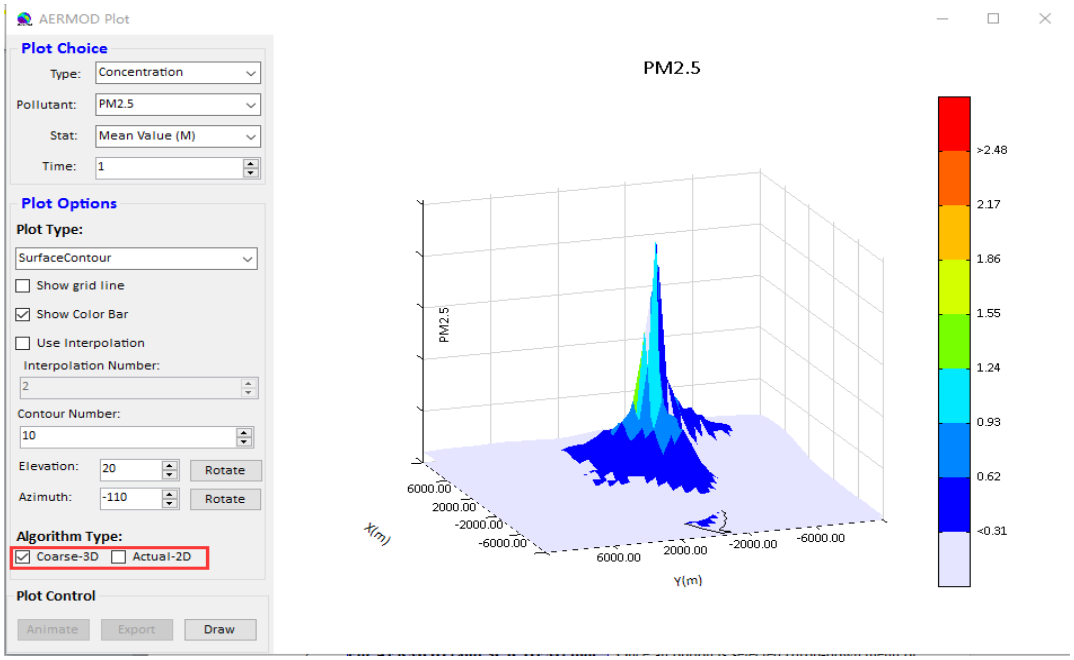


Figure 4.6-3. AERMOD Plot Window (Coarse-3D Type)

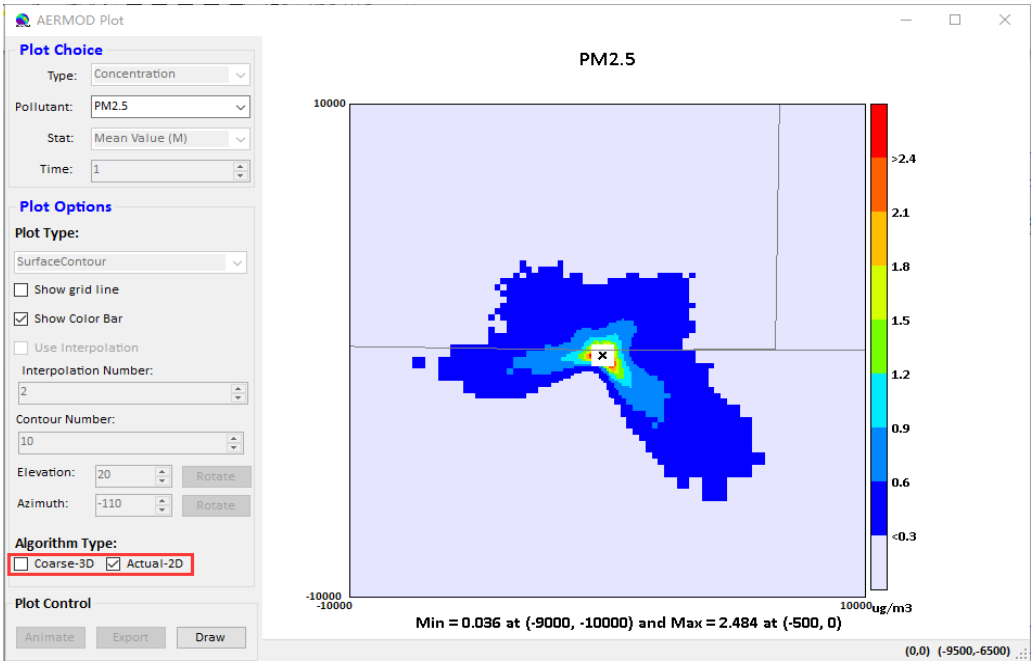


Figure 4.6-4. AERMOD Plot Window (Actual-2D Type)

SCICHEM

When users check the **SCICHEM**, users will see the window as show as Figure 4.6-5, and **Source Coordinate File** option will be disenabled.

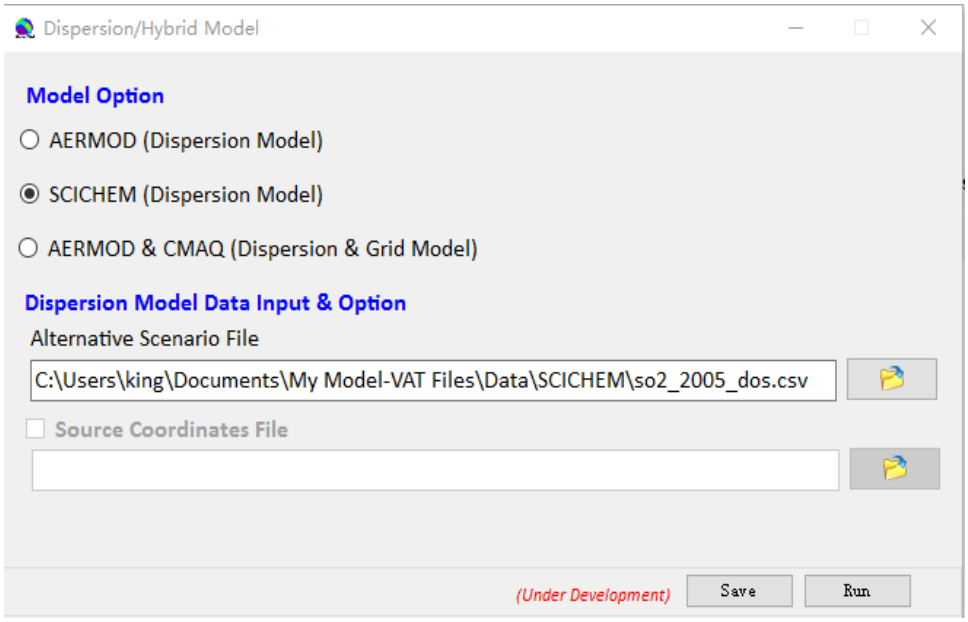


Figure 4.6-5. SCICHEM Window

Then click **Run** to run the model, there is a SCICHEM Plot window been opened as show as Figure 4.6-6.

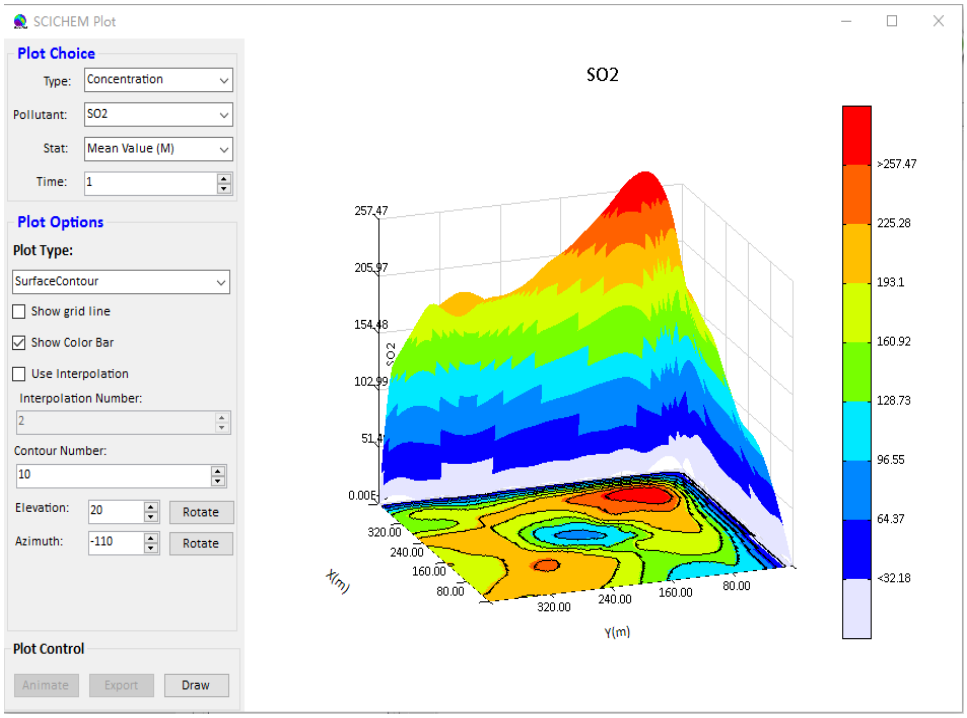


Figure 4.6-6. SCICHEM Plot Window

AERMOD & CMAQ

When users check the **AERMOD & CMAQ**, users will see the window as show in Figure 4.6-1, and users can check or uncheck **Source Coordinates File**. There are two kinds of grid model data, users can check daily or quarterly.

Then click **Run** to run the model, there is a Model Fusion Plot window been opened as show as Figure 4.6-7.

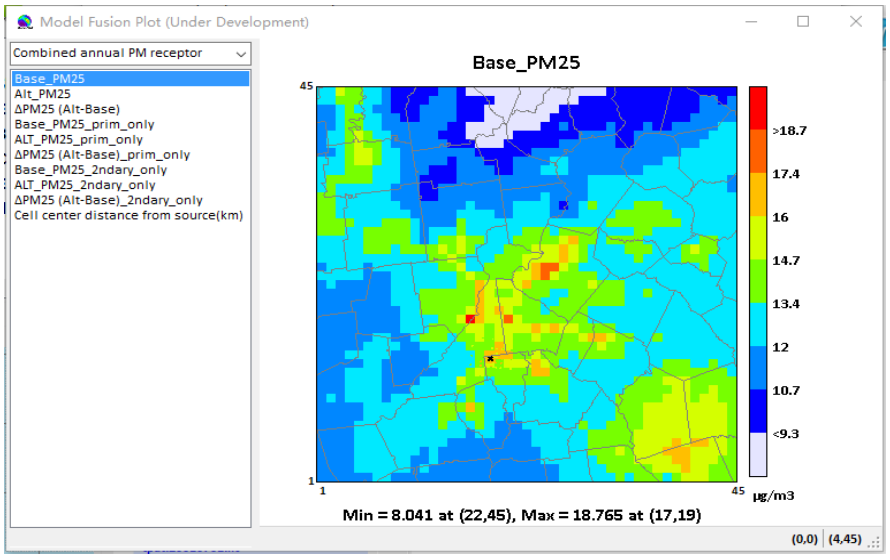


Figure 4.6-7. Model Fusion Plot Window

Note:

1. Users could zoom in and zoom out in the **AERMOD**, **SCICHEM,** and **AERMOD & CMAQ** plot.
2. Users could make the figure 3D revolve by left click and drag the mouse.
3. Users could set the **Number of Tiles** on Configure Plot to change the smooth of the plot as show as Figure 4.6-8.

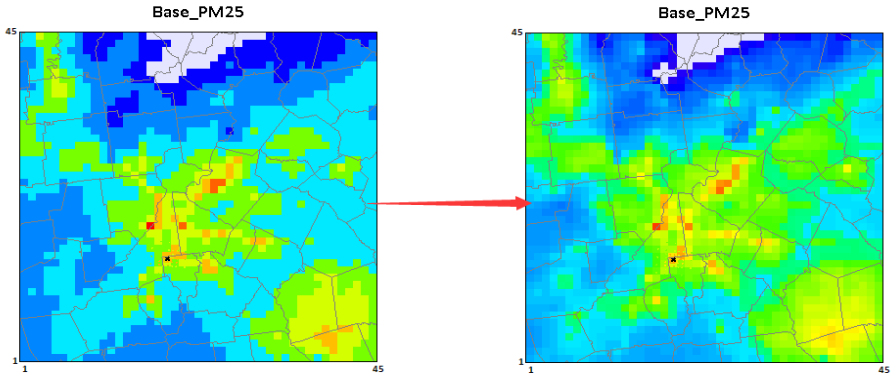


Figure 4.6-8. Set the Number of Tiles to a Lager One to Make the Plot Look Smoothly

4.7 GeoPlatform

Click **GeoPlatform** to open **GeoPlatform** window. User can enter the visual display interface of air model based on online map.

As shown in Figure 4.7-1 **GeoPlatform** window, on the left is the loading model file area. Users can preload different model files, select the pollutant to be displayed, and choose which online map to draw based on.

On the right is the display area, which includes basic GIS features, allowing users to zoom in and out, pan the map, and select layers, and so on.

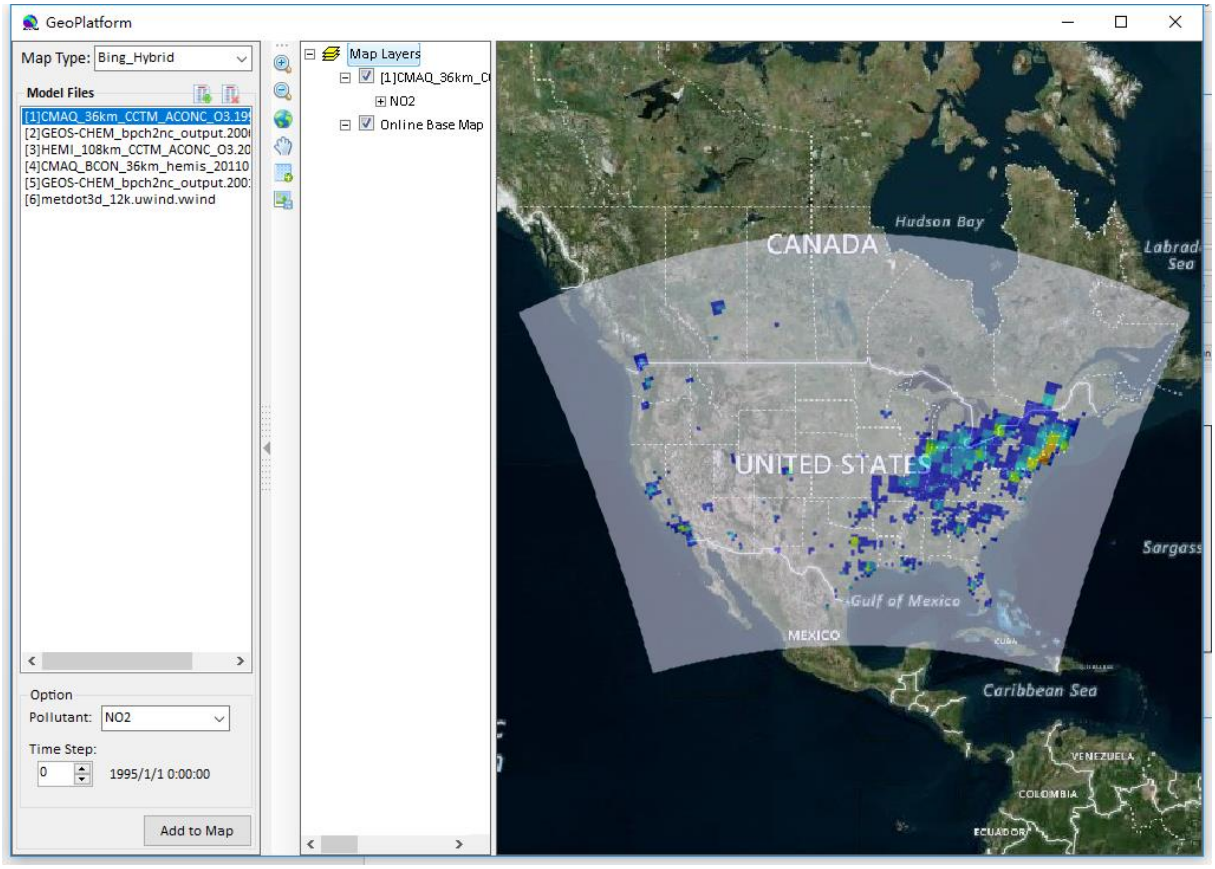


Figure 4.7-1. GeoPlatform Window

4.8 Help

Click **Help** to open the **Help** list. There are two options under the **Help**.

Model-VAT Help

Users could click **Model-VAT Help** or  button to open Quick User Guide of Model-VAT window as show as Figure 4.8-1.

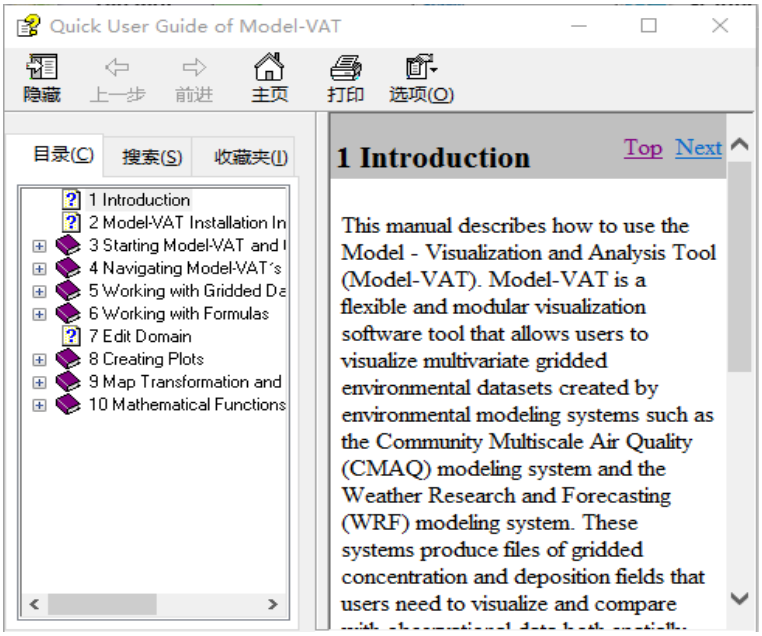


Figure 4.8-1. Quick User Guide of Model-VAT Window

About

Users could click **About** or  button to open About Model-VAT window as show as Figure 4.8-2.

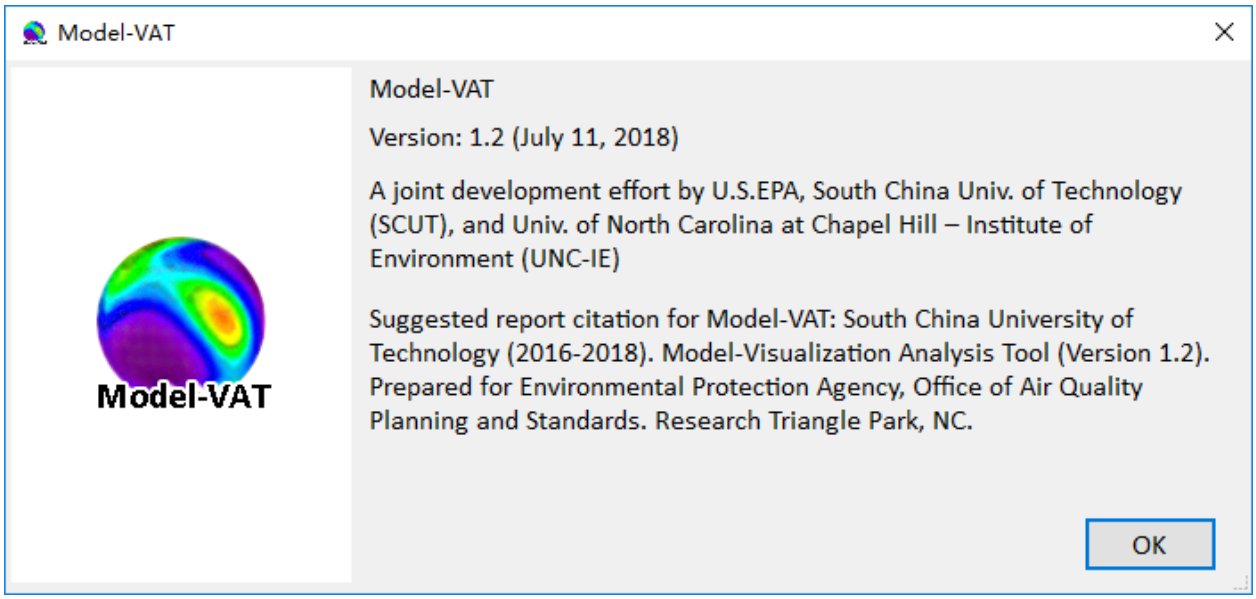


Figure 4.8-2. About Model-VAT Window

5 Working with Gridded Datasets

5.1 Gridded Input File Formats

5.1.1 Model Formats

Model-VAT currently supports visualizing files in the following file format conventions: CMAQ Input/output Applications Programming Interface (I/O API) NetCDF, WRF NetCDF, and Geos-Chem, AERMOD, SCICHEM and ASCII format (for observational data). In the future, format of CAMx and (UAM-IV) will be supported.

The CMAQ I/O API was designed as a high-level interface on top of the NetCDF library. (See https://www.cmascenter.org/ioapi/ and http://www.unidata.ucar.edu/software/netcdf/ for further information). The I/O API library provides a comprehensive programming interface to files for the air quality model developer and model-related tool developer. I/O API files are self-describing and include projection information within the gridded dataset.

NetCDF and I/O API files are portable across computing platforms. This means that these files can be read regardless of what computer type or operating system you are using. There are routines available to convert data to these formats or new code can be written and contributed to Model-VAT for use by the community.

5.1.2 Observational Data Formats

Observational data in ASCII format can be obtained from EPA s Remote Sensing Information Gateway - RSIG (http://badger.epa.gov/rsig). To use a consistent set of units for the model data and the observational data, you may need to import the ASCII data into an Excel spreadsheet and perform a unit conversion. Model-VAT doesn t allow the user to use an observational variable to create a formula, so conversions to different units should be done within an Excel spreadsheet. Import the ASCII file that is generated by RSIG into Excel, change the units to match the units found in the gridded model data file and then save using a tab delimited ASCII file format.

The observational data ASCII format recognized by Model-VAT is an ASCII file with tab-separated columns where the first four columns are provided in the order shown in and one or more additional columns are arbitrary but must have the header format 'name(units)' as shown in Figure 5.1-1. Spreadsheet programs can be used to edit and write the files by choosing ASCII output and tab as the delimiting character (instead of comma). Data within a column must be complete, as empty fields within a column will prevent Model-VAT from reading the observational data. Model-VAT allows the user to specify an alphanumeric value (either numbers and/or letters) for the fourth column (Station ID).

```
Timestamp(UTC)       LONGITUDE(deg)      LATITUDE(deg) STATION(-)    pm25(ug/m3)
2005-08-26T00:00:00-0000   -121.7842     37.6875       060010007     11.0000
2005-08-26T00:00:00-0000   -122.3991     37.7660       060750005     12.0000
2005-08-26T00:00:00-0000   -122.2034     37.4829       060811001     21.0000
2005-08-26T00:00:00-0000   -121.8950     37.3485       060850005     16.0000
2005-08-26T01:00:00-0000   -121.7842     37.6875       060010007     21.0000
2005-08-26T01:00:00-0000   -122.3991     37.7660       060750005     22.0000
2005-08-26T01:00:00-0000   -122.2034     37.4829       060811001     19.0000
2005-08-26T01:00:00-0000   -121.8950     37.3485       060850005     20.0000
2005-08-26T02:00:00-0000   -121.7842     37.6875       060010007     28.0000
```

Figure 5.1-1. Observational File ASCII Format

5.2 Example Datasets

5.2.1 Example Datasets from the Model-VAT

The example datasets are provided under the Model-VAT installation directory/data directory. For example:

- Windows: C:\Program Files (x86)\ Model-VAT\Data

5.2.2 Example Datasets from the ABaCAS

If you have installed the ABaCAS, you can find the example datasets from the ABaCAS installation directory/Model-VAT /data directory. For example:

- Windows: C:\Program Files (x86)\ABaCAS\Model-VAT\Data

These datasets may be used to recreate example plots that are provided in this user guide, including a tile plot with observational data overlay in Section 1), and the example datasets for the various dataset projections that Model-VAT supports including LCC, polar stereographic, UTM and Mercator. The data directory currently contains six subdirectories:

1. Base_map contains sample coordinates files for region of US, China and the whole world;
2. BC contains sample boundary files;
3. CMAQ contains sample WRF and CMAQ I/O API datasets;
4. Geos-Chem contains sample Geos-Chem files;
5. OBS contains an ASCII formatted observational dataset, and an observational dataset of CSV format;
6. ShapeFile contains sample shapefiles for US, China, and the whole world.

5.3 Adding and Removing a Dataset from a Local File System

To load a data set from a local file system, press the green **plus** button at the top of the **Datasets** pane. A file browser (Figure 5.3-1) allows you to select a dataset for use in Model-VAT.

After you select a dataset, Model-VAT will load header information on **Dataset** pane and display the available variables on **Variables** area, and domain on **Domain** pane used by the file (Figure 5.3-2). (The actual model data are not loaded until later, when plots are created.)

Datasets can be removed by highlighting the name of the dataset in the dataset list and pressing the green **minus** button. Note that although the dataset will be removed, the number that was assigned to that dataset will not be reused by Model-VAT during the current session (unless there had been only one dataset loaded, and it was removed; in that case the next dataset that is loaded will be labeled number 1).

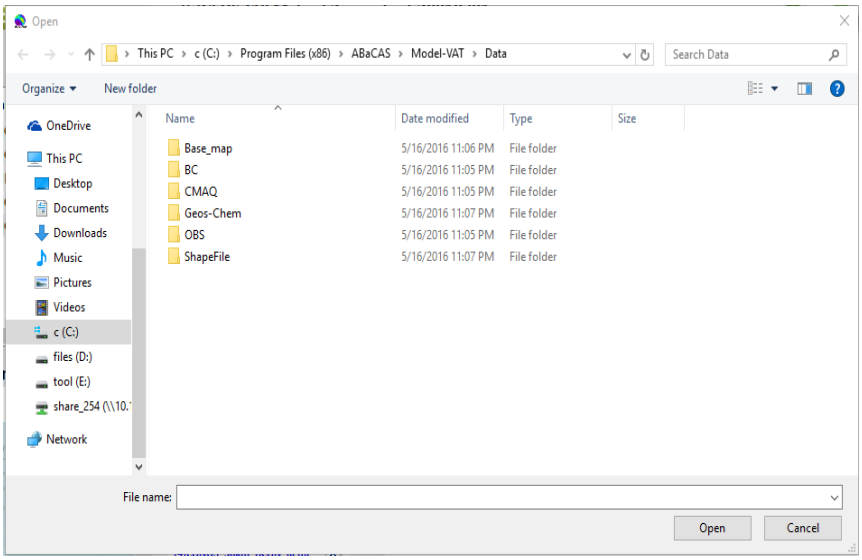


Figure 5.3-1. Open Dataset File Browser

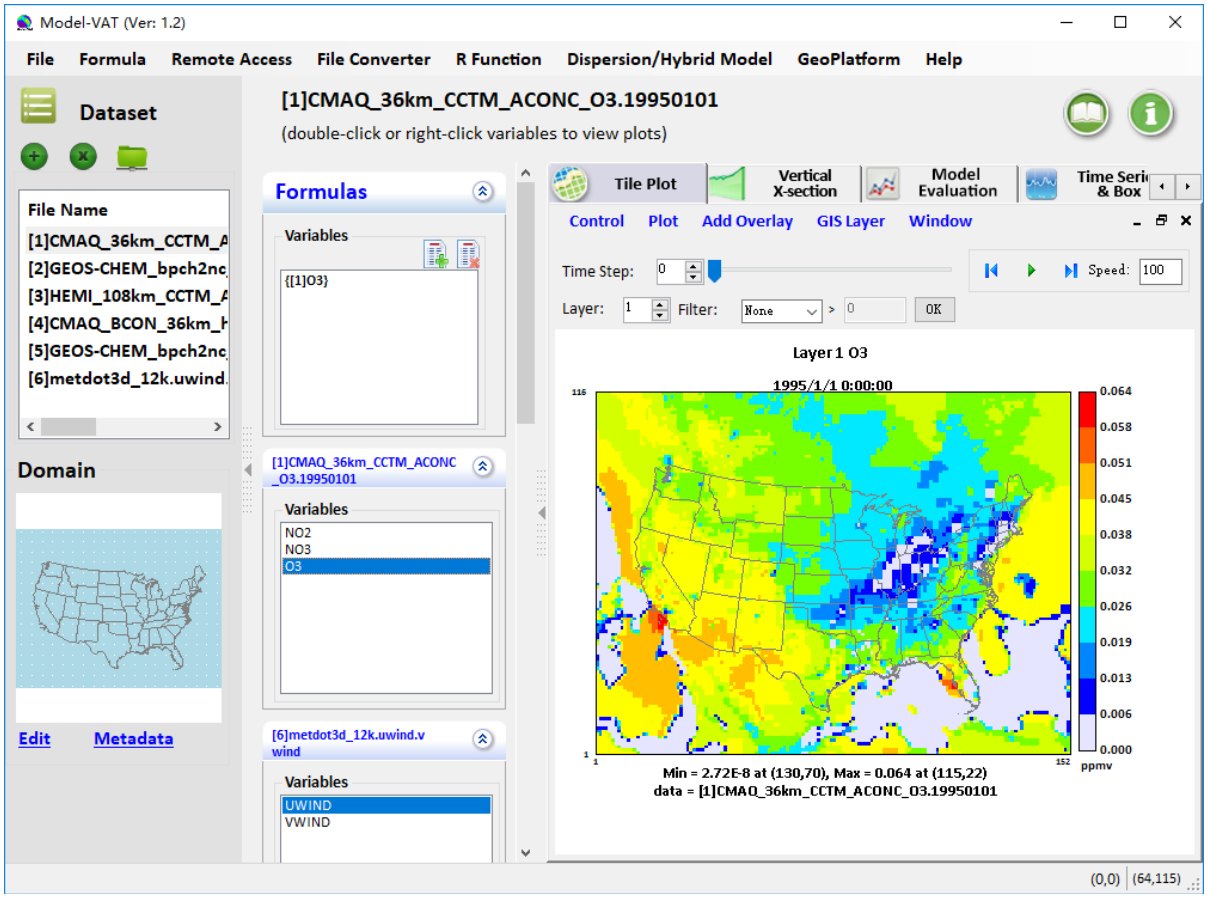


Figure 5.3-2. Datasets Pane, Domain Pane, and Variables Area

5.4 Variables List

The variables list shows all of the variables contained in a loaded dataset. Once a **Dataset** is added, a responding variable list will be added on **Variables** area. Each of the variables in the list can be used to create a formula in the **Formula Pane** that can be used to create plots. Model-VAT allows the user to automatically add a formula by double-clicking on the name of a variable. This automatically creates a formula that contains the variable for the loaded dataset and makes it the default formula for making plots. In addition, you may right-click on the name of the variable to show a popup menu as shown in Figure 5.4-1. From this menu you can add one kind of plot from **Tile Plot**, **Vertical X-section**, **Time Series & Box**, **Scatter**, and **3-D**. Formulas are described in more details in Chapter 6 and Plots are introduced in details in Chapter 8.

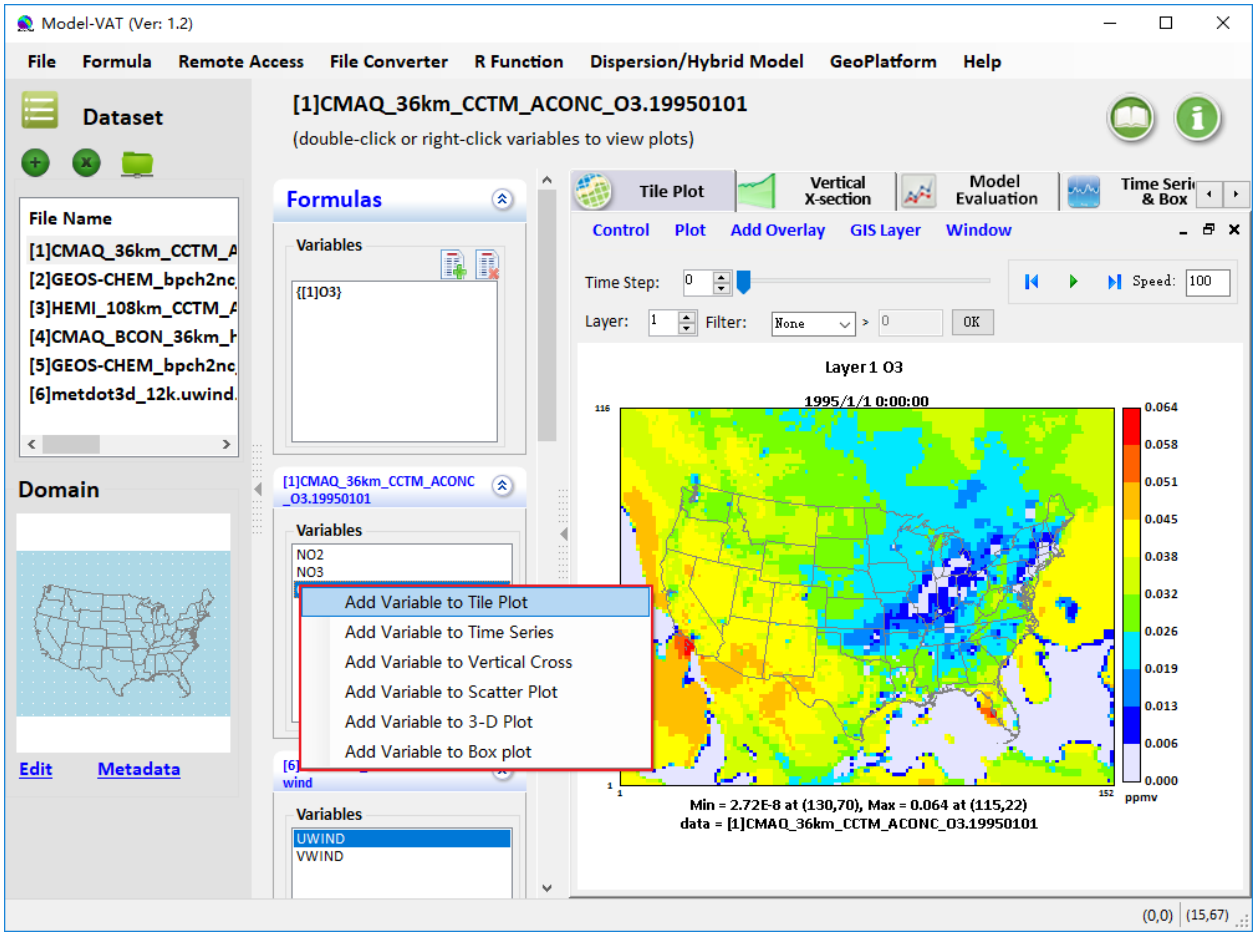


Figure 5.4-1. Right-Click on Variable List

5.5 Domain Panel

The domain panel contains an Edit button and a Metadata button. Detailed instructions for using the Edit button are provided in Chapter 7, **Edit Domain**.

To display the metadata information about a dataset, click on the **Metadata** button in the **Domain** panel. A window containing the metadata will appear (Figure 5.5-1). Each dataset includes metadata information that is part of the file header. The metadata provided include the map projection information, dataset dimensions, and dataset global attributes.

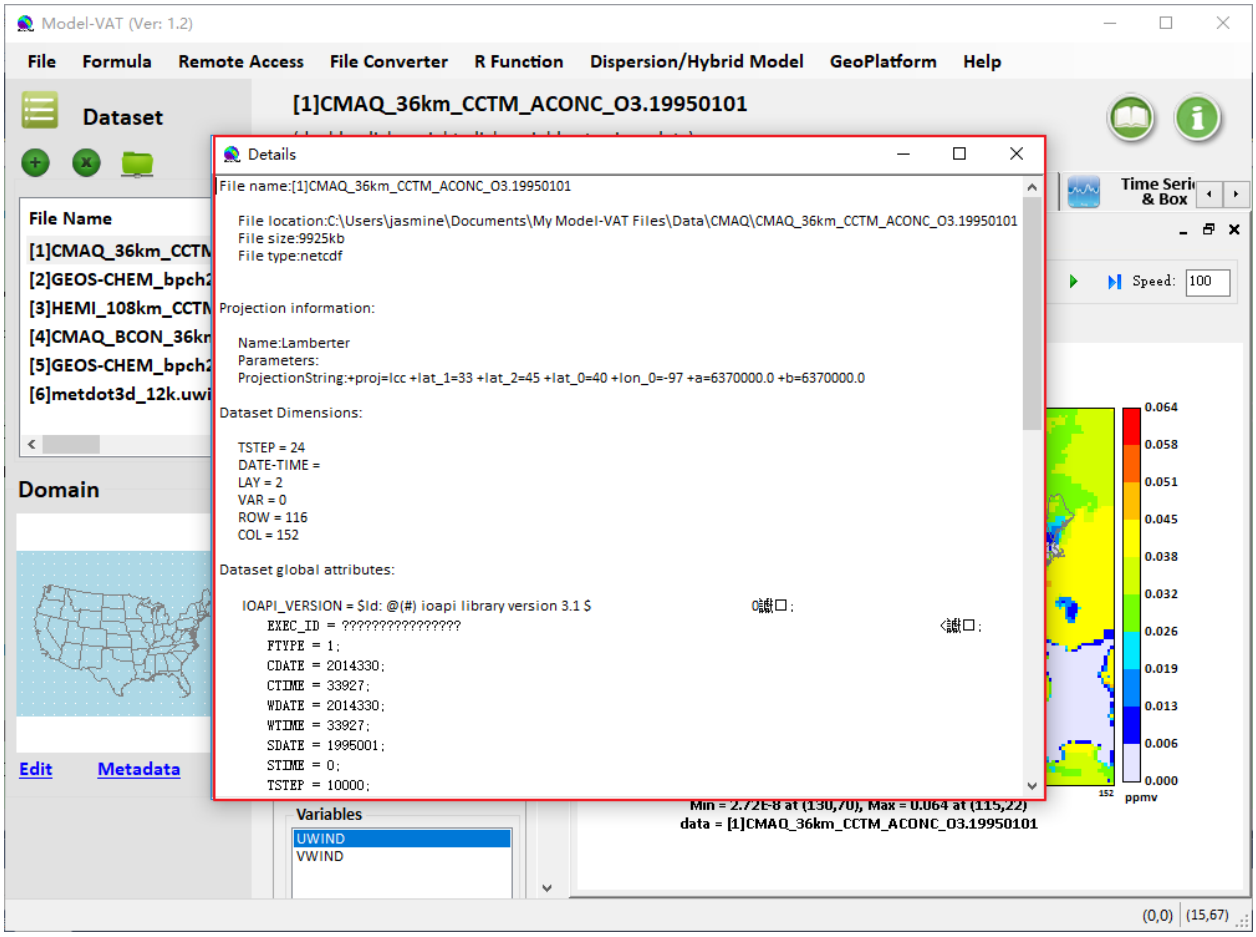


Figure 5.5-1. Dataset Metadata Information

6 Working with Formulas

All plots in Model-VAT are generated from formulas. A formula is used to compare or manipulate variables in one or more gridded datasets. A formula can be as simple as a single variable from one gridded dataset or it can be an equation that uses variable(s) from one or more gridded datasets. Formulas are used to create visualizations that can assist with model performance evaluations, for example, or that help in comparing model results with observations.

6.1 Adding and Removing a Formula

After loading the desired gridded datasets, you can use the variables in them to create formulas. To use a variable to create a simple formula, double click on the name of the variable. This will add the formula {<Dataset Number> [<Variable Name>]} to the formula list in the **Formulas** pane for example, {[1] O3}. Click the green plus button on **Formula Pane** or click **Formula** to open **Formula Calculator**. Left side of Formula Calculator is all variables of imported datasets. The right side is the **Formula Operation**. It is divided into three parts: **Function**, **Formula Editor,** and **Formula** (Figure 6.1-1).

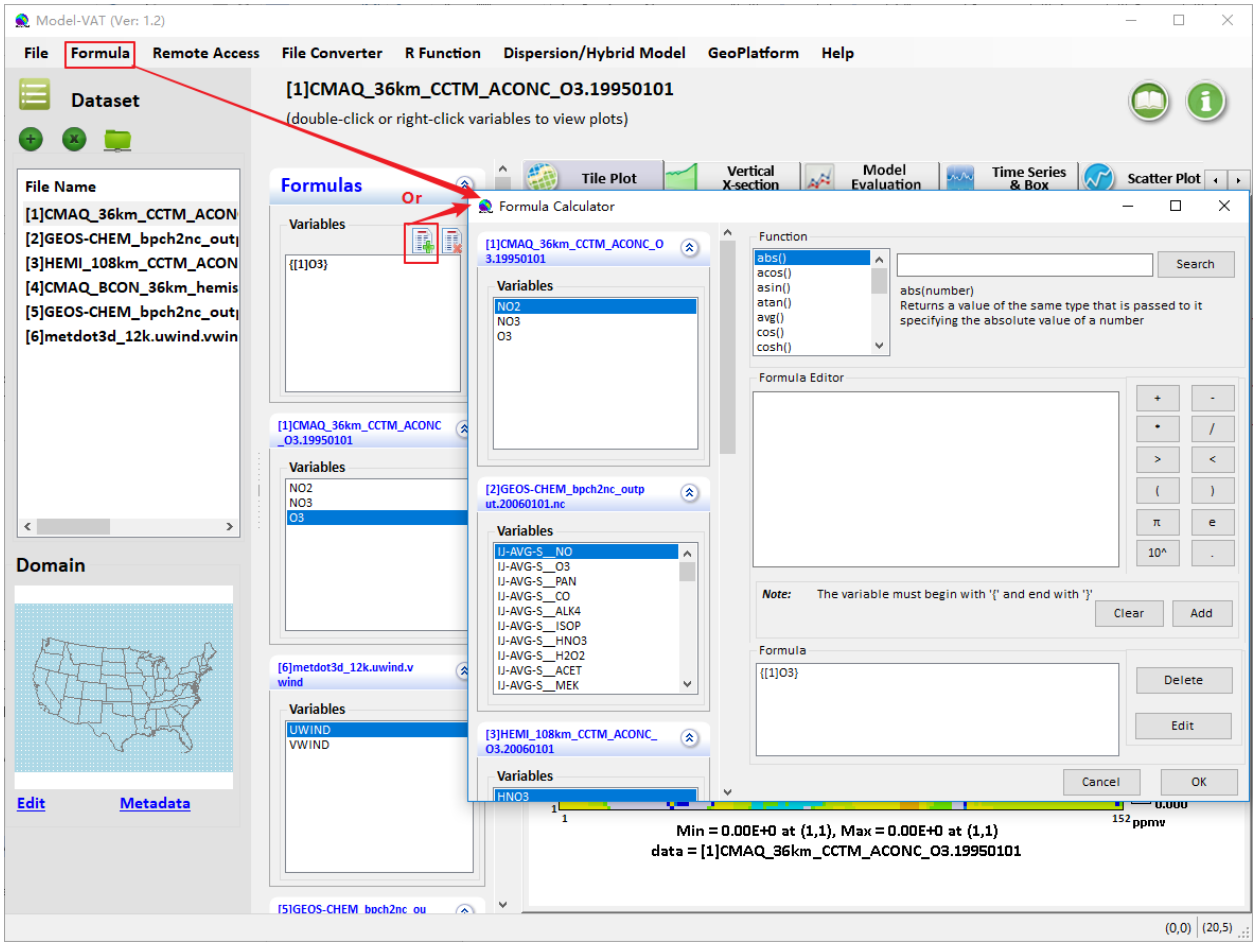


Figure 6.1-1. Adding Multiple Variables on Formula Calculator

Function provides some regular functions for convenience and you can double-click a function to add it to **Formula Editor**. To add a variable to the **Formula Editor**, double click the variable on the left side. Note that every variable must begin with {and end with}. After the variable names are added to the **Formula Editor**, use the cursor and the keyboard to type in the

mathematical operators (double-click an operator) where needed to create a valid formula (see Section 6.2 and Chapter 10). After the formula has been created in the Formula Editor, click the **Add** button, to place it in the list of formulas available in the **Formula.** Finally, all formulas created will be stored and displayed on **Formula Pane** when clicking **OK** button.

You can edit or delete an added formula by clicking **Edit** or **Delete** button on **Formula Calculator**. You can also delete an added formula by clicking  button on **Formula Pane.** Note that removing a formula from the formula list does not remove plots that were created prior to the deletion of the formula.

6.2 Example Formulas

To examine the values of ozone in dataset 1, the formula would be {[1] O3}.

To examine the difference in ozone between datasets 2 and 5, the formula would be {[2] IJ-AVG-S__O3} - {[5] IJ-AVG-S__O3}.

To calculate the percent difference in ozone between datasets 2 and 5, the formula would be ({[2]IJ-AVG-S__O3} - {[5]IJ-AVG-S__O3})*100/({[5]IJ-AVG-S__O3}) .To identify all cells where the ozone concentration exceeds a certain value, you can use the Boolean operators to screen out ranges of your data that are of particular interest. A Boolean expression will evaluate to either True = 1 or False = 0. For example, to plot the cells in which the ozone values in dataset 1 exceed 0.080 ppm, you could use the formula ({[1] O3}>0.080)* {[1] O3}. In the resulting plot, each cell where {[1] O3} exceeds 0.080 will show the value of {[1] O3} for that cell; for all other cells the value shown will be zero.

The notations that can be used in formulas to represent various mathematical functions, and the order of precedence of these functions, are listed in Chapter 10, Mathematical Functions.

6.3 Selecting a Formula for Plotting

Before creating a plot, a formula must be selected. Check to see which formula is highlighted in the **Formula Pane**. By default, Model-VAT designates the most recently added formula as the selected formula. To change the selected formula to a different one in the list, click on a formula in the list on the **Formula Pane** to highlight it.

7 Edit Domain

Datasets contain data for cells over a particular geographic area. The Model-VAT program refers to this area as a domain. By default, the entire domain contained in a dataset is used in creating plots. Use the **Edit Domain** dialog box to select a subset of this domain for plotting.

First, select a dataset on **Dataset** pane and you will see the whole domain of current dataset on **Domain** pane. To select a subset of the domain, press the **Edit** button or click the map. The **Edit Domain** dialog box then appears (Figure 7-1).

There are two ways to select domain, using **Select Region** at the top or adjusting start and end position at the bottom. If the area of interest is large, background data such as state and county outlines will be shown on the map. The **magnifying glass** buttons above the map can be used to zoom in and out on the map. To select the desired cells, click to select a starting position and then drag a box around the area of interest; selected cells will appear in red. To clear the selected area, use the **Clear Region** button, and the **Clear Region** button under the button in the blue circle as show as Figure 7-1. When you are satisfied with the domain subset you have chosen, click the **OK** button.

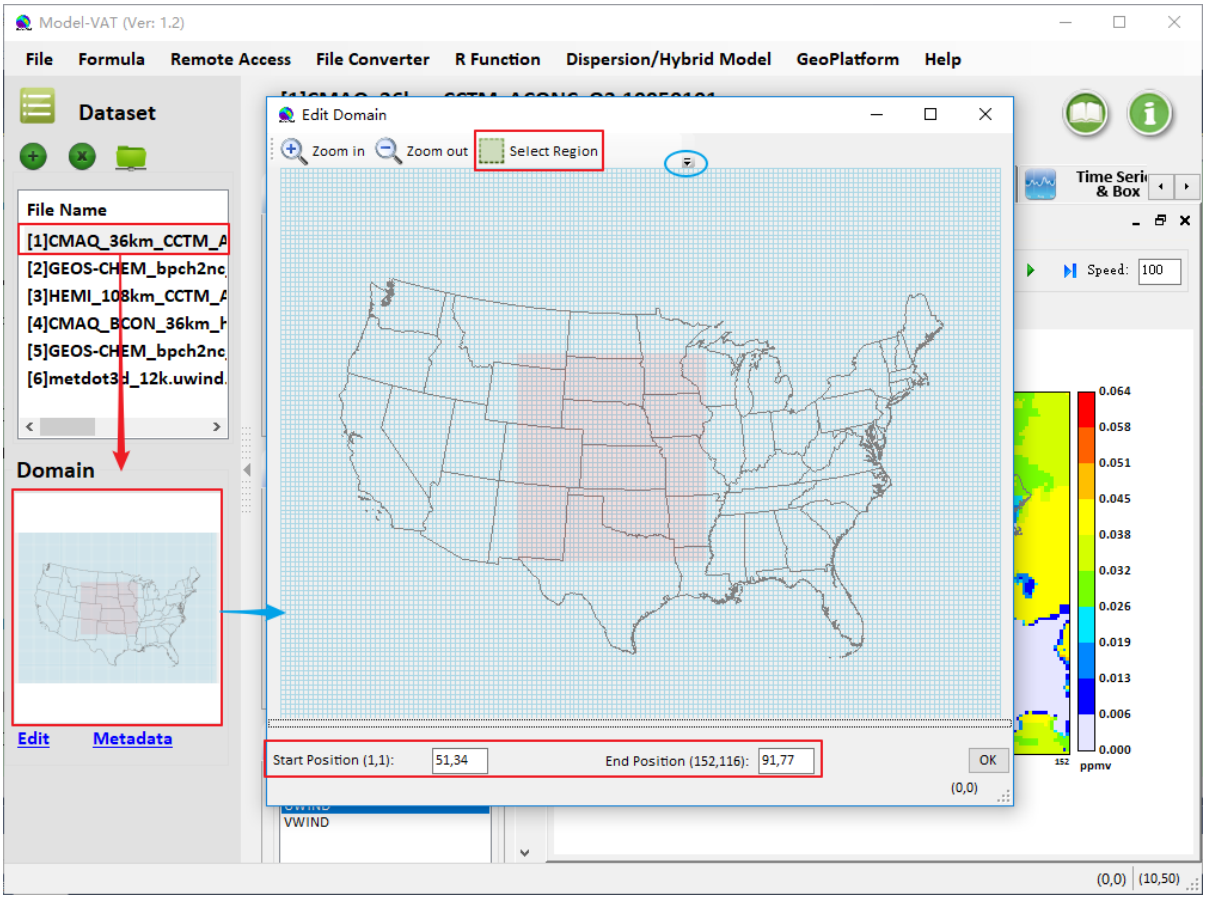


Figure 7-1. Edit Domain Dialog Box

When you finish editing domain, the selected domain will be showed on the map of **Domain.** Then you can choose a variable to create a tile plot within the selected region (Figure 7-2).

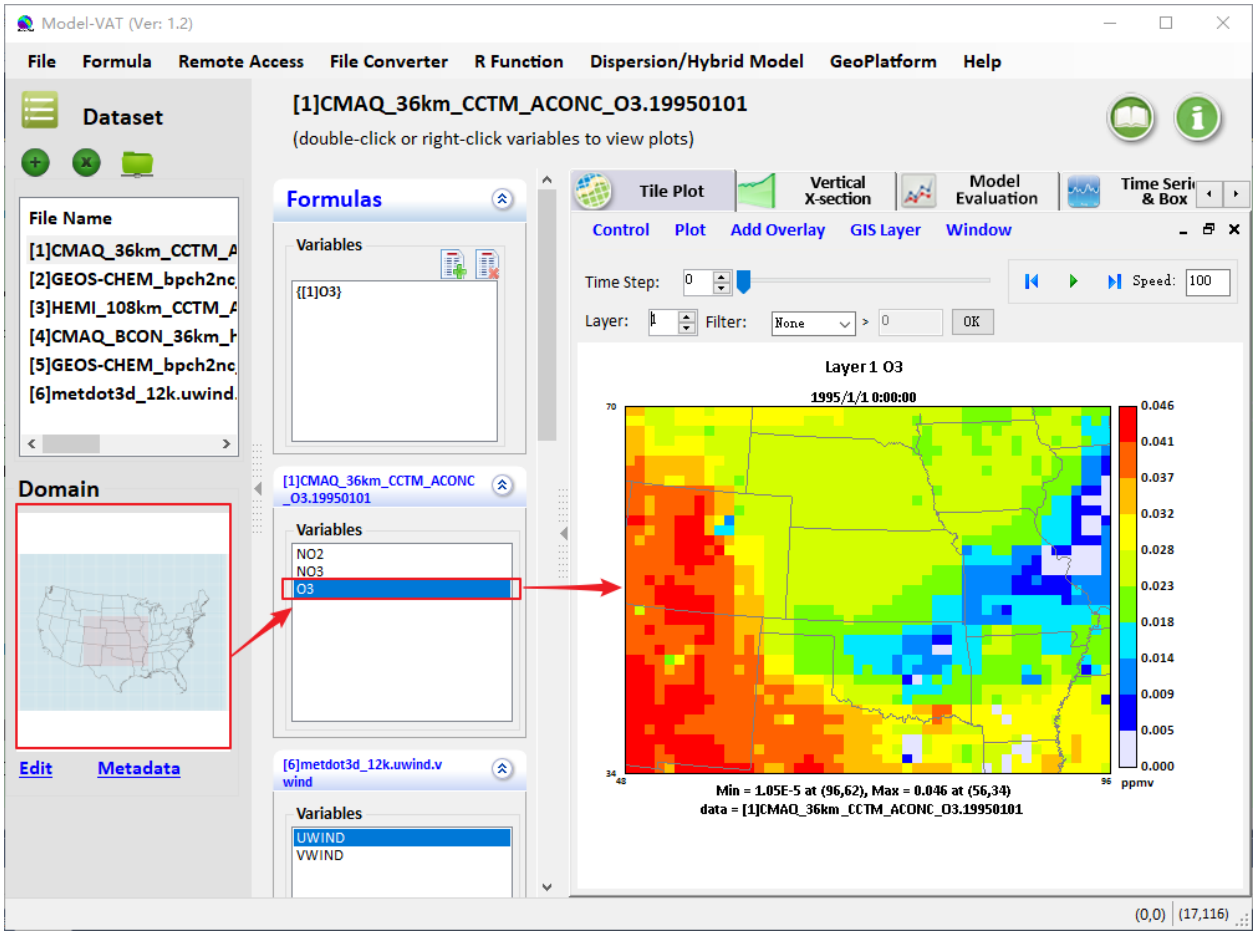


Figure 7-2. Create a Plot within Selected Region

User are also allowed to zoom in domain by clicking mouse button and dragging to the right-low, and go back to previous zoom domain by clicking mouse button and dragging to the left-top or right-clicking or clicking the Control to click the **Zoom out to previous domain** or **Reset to default domain**.

8 Creating Plots

After creating a formula, you are ready to create and view some plots. The available plot types are shown on the Plot Area: **Tile Plot**, **Vertical X-section**, **Time Series & Box**, **Scatter Plot**, **3-D Plot**, and **Data Plot**. All of these are described in this chapter. Note that not all datasets are appropriate for all plot types.

To generate a plot first highlight a formula in the list of formulas you have created in the **Formula Pane** or a variable in the list of corresponding **Variables** area. Next, right click to choose the plot style to generate the plot.

Each plot contains its own menu bar at the top of its window with options for configuring and exploring that type of plot. The menus may include **Configure, Controls, Plot, GIS Layers, Add Overlay, and Window**.

8.1 Tile Plot

The **Tile Plot** allows the user to display the time steps and vertical layers contained in a gridded dataset and to display grid cell time aggregate statistics. An example of the Tile Plot window is shown in Figure 8.1-1.

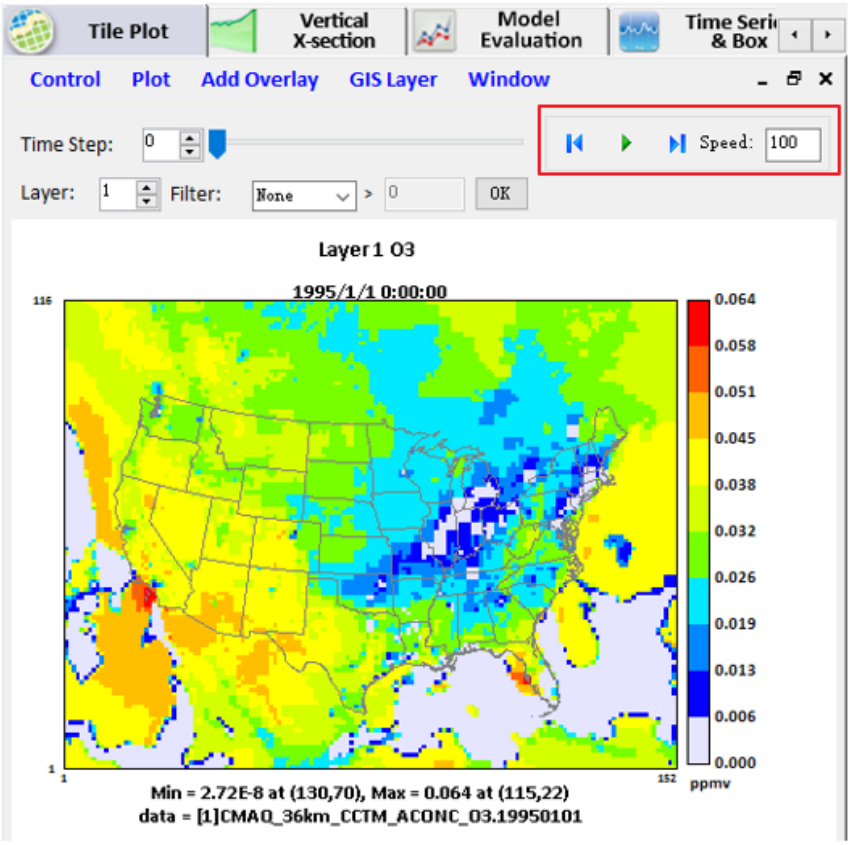


Figure 8.1-1. Tile Plot

There are three ways to add tile plot: Double-clicking a variable in the list box of variable, mouse right-clicking a variable in variable list box, and select **Add Variable to Tile Plot** and double-clicking a variable in the formula list box.

Users can also see the BCON file in the tile plot (Figure 8.1-2).

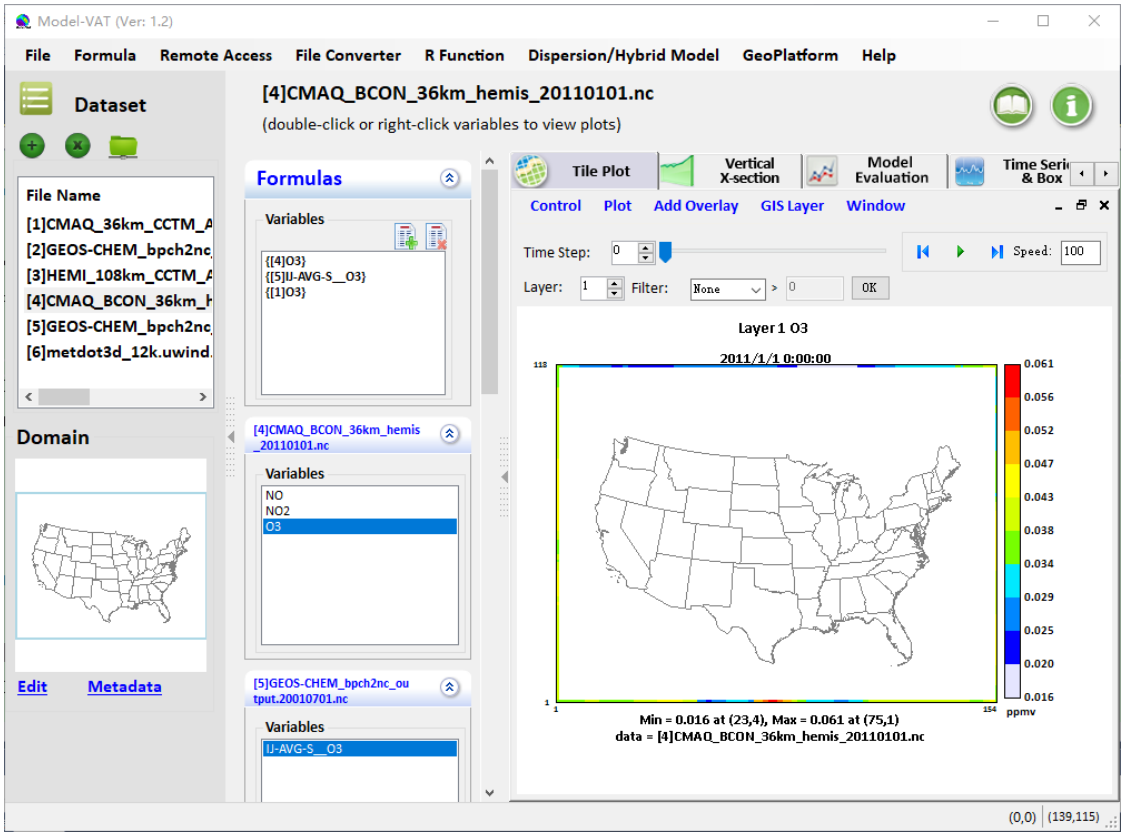
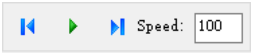


Figure 8.1-2. BCON File Showed in the Tile Plot

8.1.1 Time Selection and Animation Controls

At the top is a track bar and its settings. To change the time step, you can drag the cursor to the position you want on the bar.



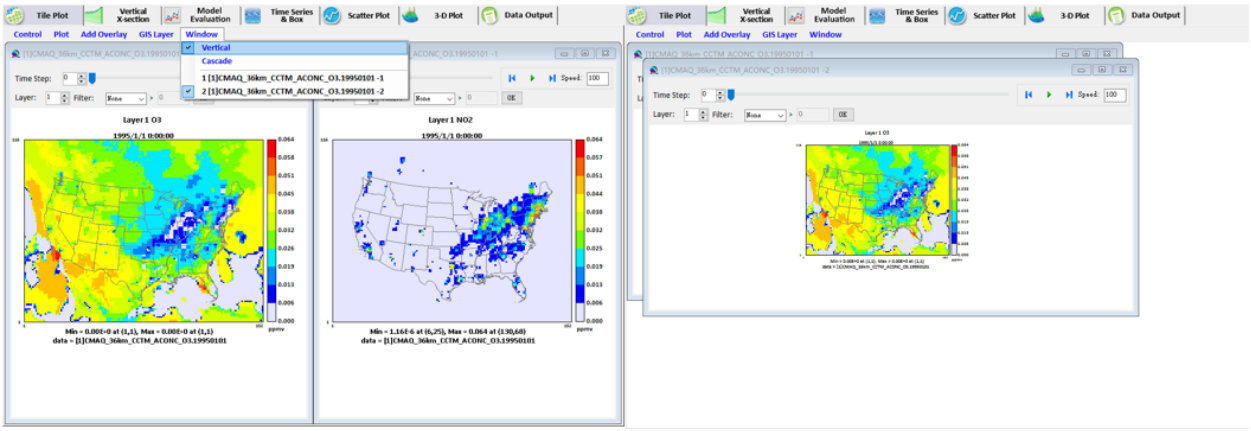
Buttons in the top right corner of the plot allow you to use play/stop, reverse, forward, and speed options to control the animation of the plot (Figure 8.1-1). Control the speed of the animation through the text box labeled **Speed**; the default delay is 100 milliseconds between frames. If that text box is not visible, expand the plot window s width by clicking with the left mouse button on the right edge of the window and dragging to the right. Enter a number in the box for the length of the delay and then press the **Enter** key. A larger plot with multiple map layers may require a shorter delay between frames than a small zoomed-in plot with few map layers. There are some kinds of basic statistical functions in the **Filter**.

User can right-click **Probe** to use **Probe** function. When **Probe** is invoked (display moving + sign), show **Column & Row** value below when moving mouse over map; When clicked, show cell value (concentrations, etc., below) and record the probed cells; When double-clicked, show a time series of probed cells.

8.1.2 Arrange Window and Select Active Window

Arrange Window:

There are two ways used to arrange tile plot window: **Vertical** and **Cascade** (Figure 8.1-3).



a. Vertical Style b. Cascade Style

Figure 8.1-3. Arrange Style of Tile Plot Window

Select Active Window:

To select an active window, you can click the title of child window or select a child window from the menu named **Window** under **Tile Plot** tab page. The checked item is the current active child window (Figure 8.1-4).

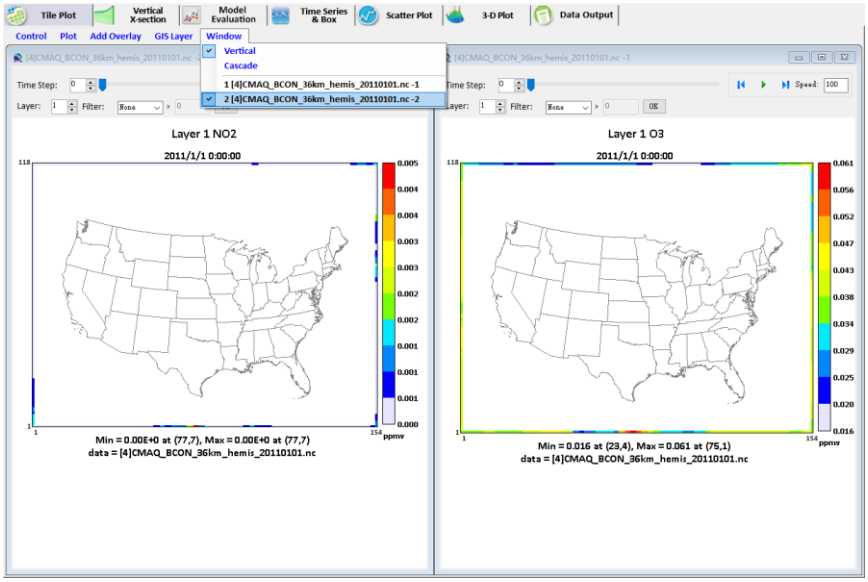


Figure 8.1-4. Select an Active Window

8.1.3 Layer Selection

The **Layer** displayed for the plot can be controlled by clicking on the up or down arrow for the Layer spin control in the top center of the plot.

8.1.4 Add Overlay

Model-VAT supports two types of overlays observations and vectors. For both types of overlays, you may need to add data from another data file onto that in the underlying plot.

1) Observational Data Overlays

It is useful to visually compare the results contained in model output datasets with the data points in observational datasets. You can do this by creating a Tile Plot of the model output and then overlaying it with observational data points. The observational dataset needs to be in a csv or txt observational data format.

Sample observational data are provided in the directory ABaCAS install directory/Model-VAT /data/OBS so you can create a sample Observational Data Overlay Plot. Follow these instructions to create your plot.

- Load a model output dataset to Tile Plot.

- Select a child window as current active window. And then select Add Overlay>Observations from the tile plot s **Plot** menu to open Add observation data window on a tile plot.

- An **Observation** dialog box (Figure 8.1-5) appears containing observation datasets to load an observational dataset, and the variables that are available in the observational dataset. Once selecting a dataset, **Available Variables list** will show all variables of added dataset. Select the observational variable to overlay on the Tile Plot from the Observation Details list. Multiple observational dataset variables can be overlaid on a Tile Plot.

- You can control the appearance of the symbols representing the observational data. The stroke size controls the thickness of the line used to draw the symbols; the shape size controls their diameter. You can use up to six different open-area shapes circle, diamond, square, star, sun, and triangle to distinguish among multiple observational datasets. A circle is the default symbol shape.

- Click **OK** to overlay the observational data on the tile plot (Figure 8.1-6).

Repeat the above process to add multiple variables. To reset the symbols size, shape, or stroke thickness, reopen the **Observation** dialog by using **Add Overlay>Observations**, select the observational variable you want to adjust, and then change its stroke size, shape size, or symbol. You can also remove a variable in the list. When you are finished click the **OK** button.

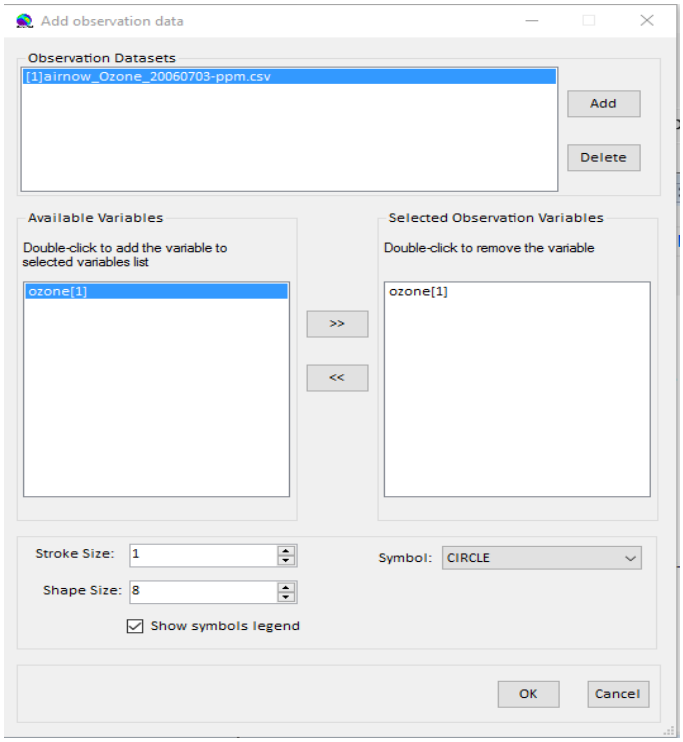


Figure 8.1-5. Tile Plot Observation Dialog

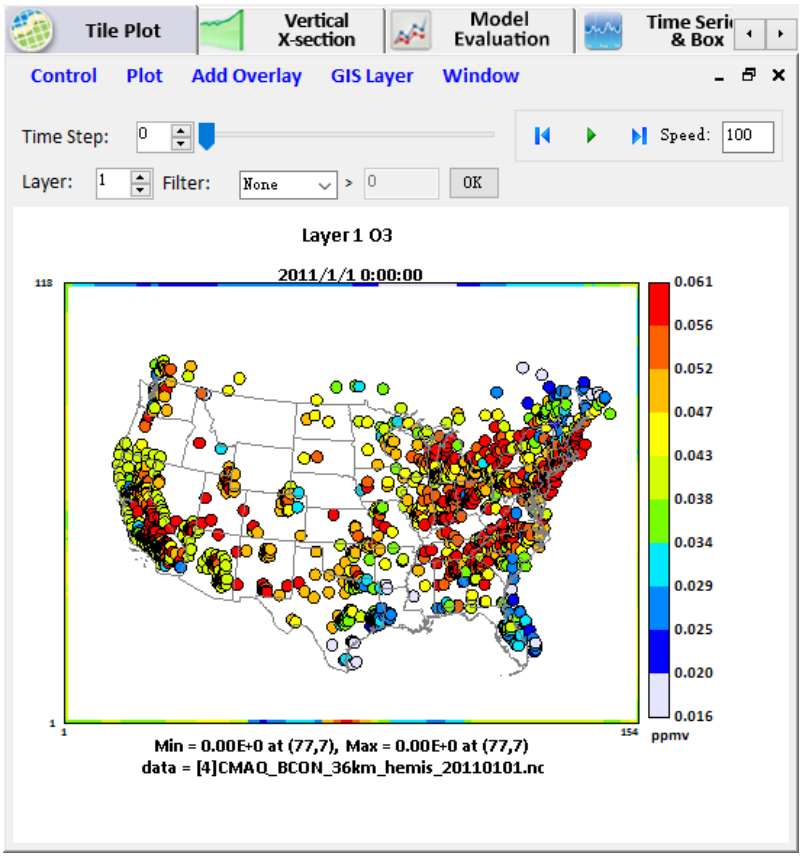


Figure 8.1-6. Tile Plot with Observational Data Overlay

2) Vector Overlays

Follow these instructions to add a vector overlay to a tile plot. Typically, these are created to show wind speed and direction on a plot of gridded air quality data. The length of the calculated vectors is proportional to their magnitude.

- Create your Tile Plot.
- Select the Add Overlay>Vectors option from the tile plot s **Plot** pull-down menu.
- Select the two components of your vector in the **Vector Overlay** dialog box (Figure 8.1-7).
- Click the **Plot button** and the vector overlays are displayed on the plot.

NOTE: At this time you cannot control how the vectors are displayed, and there is no option to remove the vectors from the plot. If you need to make a change, you must start again with your Tile Plot.

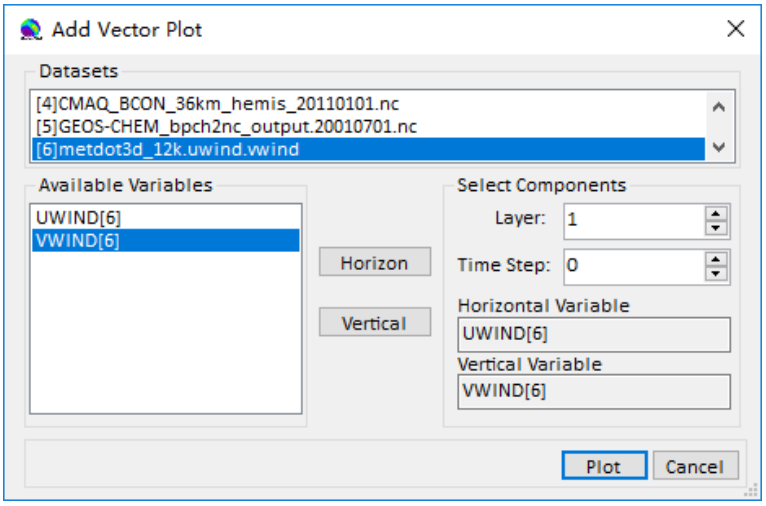


Figure 8.1-7. Vector Overlay Dialog Box

An example of an ozone concentration Tile Plot with a wind vector overlay is shown in Figure 8.1-8. The length of each vector is proportional to its length. The direction of the vector is calculated from the direction and magnitudes of its two components. This figure illustrates how the wind changes speed and direction in this portion of the modeling domain for layer 1, time step 0.

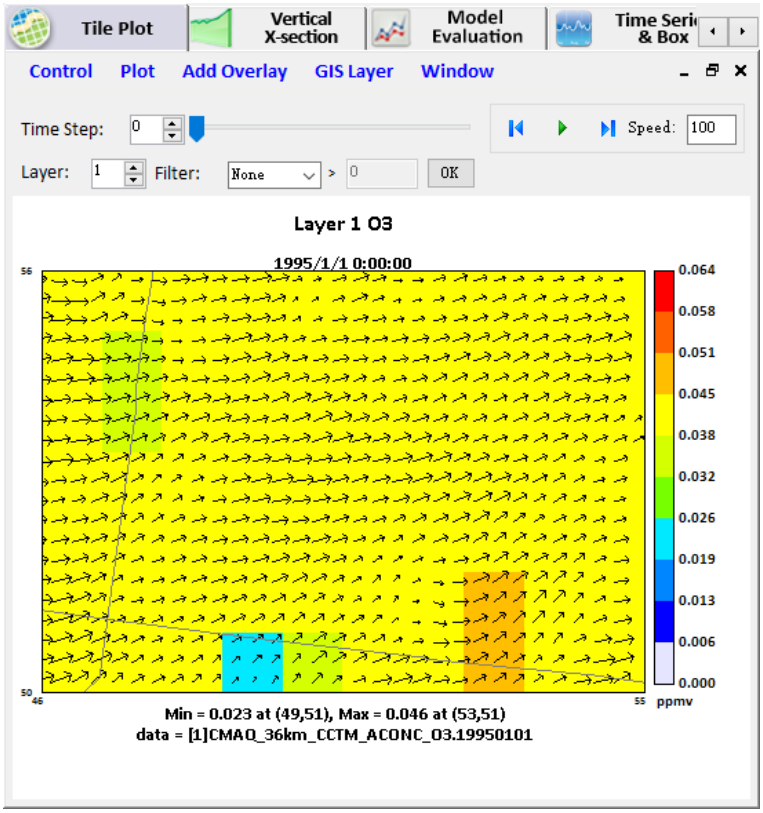


Figure 8.1-8. Wind Vector Overlay on an Ozone Tile Plot

3) Draw Wind Rose Plot

We developed the Wind Rose Plot on the basis of Vector File. Follow these instructions to add a Wind Rose Plot to a tile plot.

- Selected the **Wind Rose Plot** under the right-clicking menu or **Plot** to open Add Wind Rose Plot dialog box as show as Figure 8.1-9.
- Select the two components of your vector in the Add Wind Rose Plot dialog box (Figure 8.1-9).
- Click the **Plot button** and the vector overlays are displayed on the plot (Figure 8.1-10).

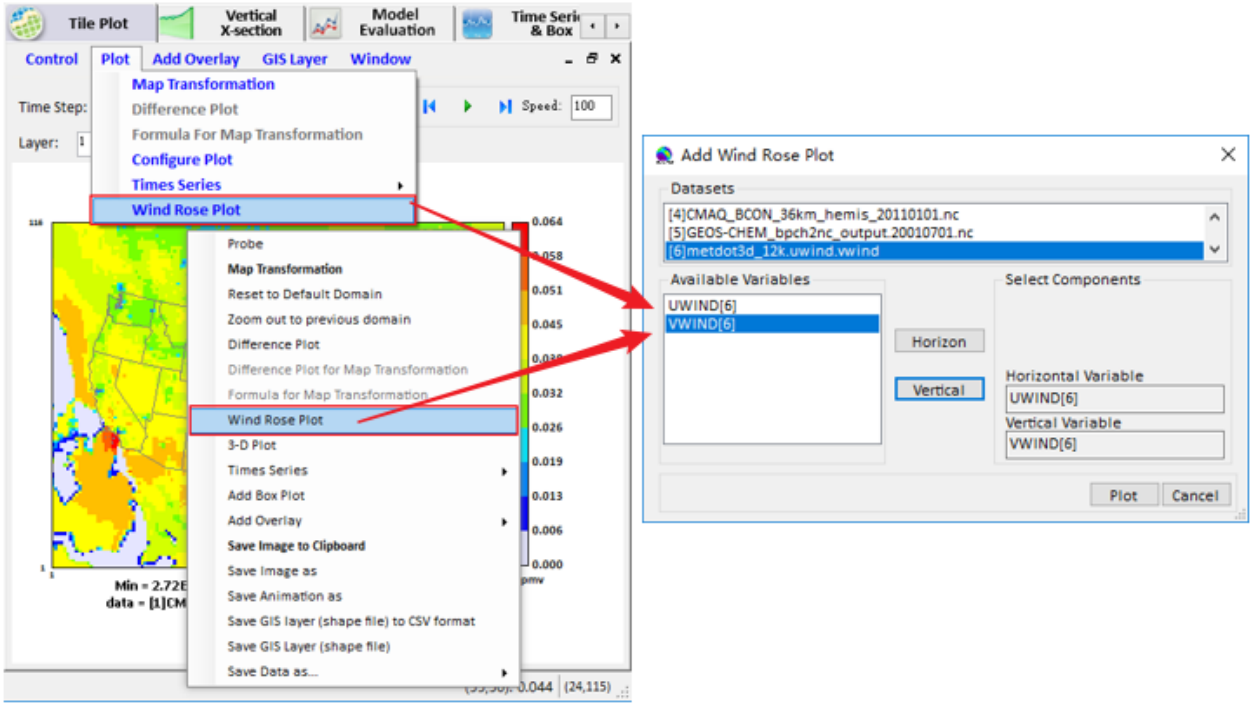


Figure 8.1-9. Open Add Wind Rose Plot

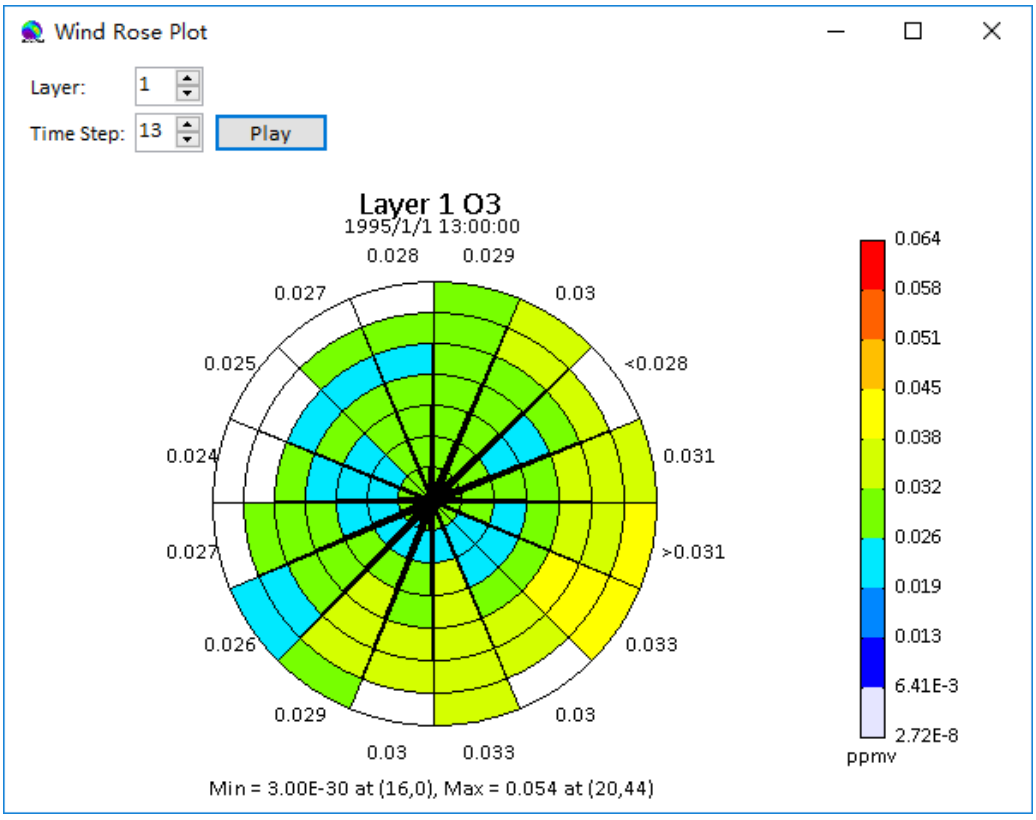


Figure 8.1-10. Wind Rose Plot

8.1.5 **Add Map Layers**

Use the **Configure Layers** option in the **GIS Layers** menu to open **Add Layer** window to add maps to a Tile Plot (Figure 8.1-11).

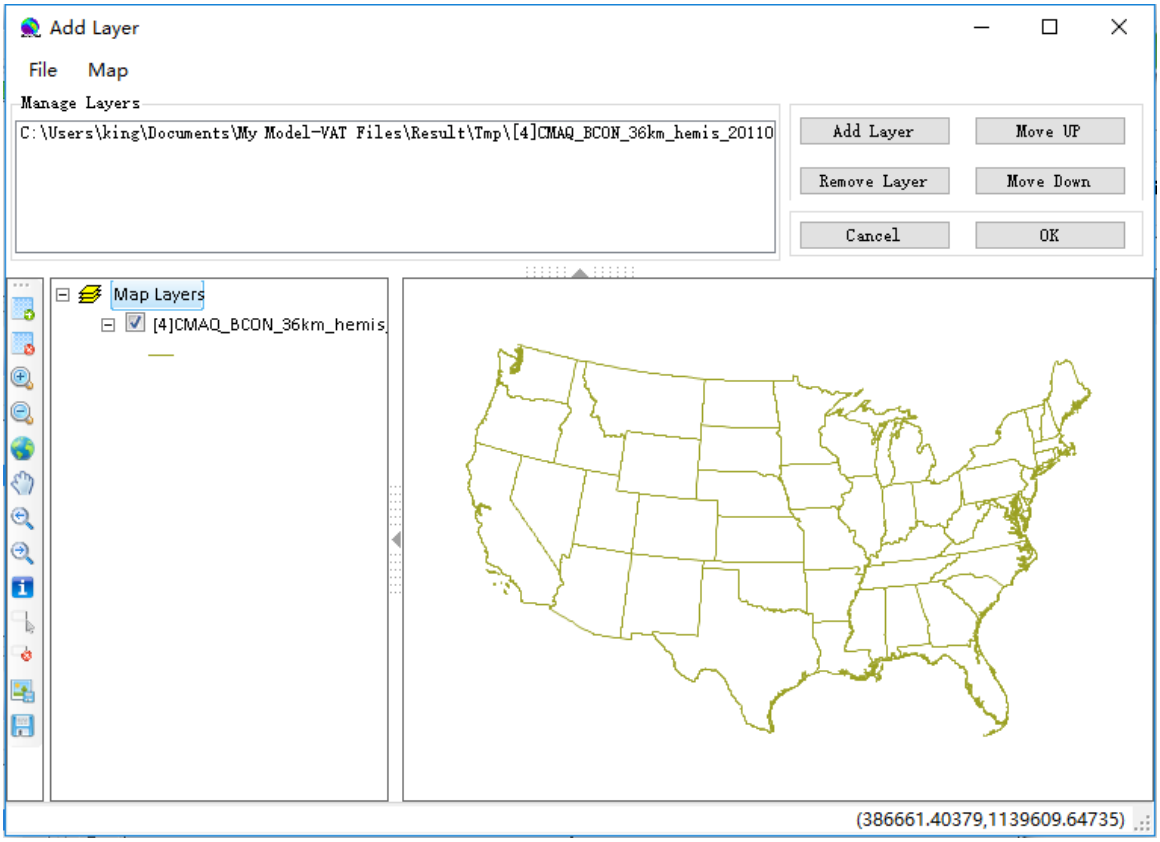


Figure 8.1-11. Add Layer Window

The upper part of the window is used for adding/removing a layer file (**Add Layer**/ **Remove Layer** button) or modifying orders of layer files (**Move Up**/ **Move Down** button)(Figure 8.1-12). To save layer file as csv format, click to highlight a file in the Manage Layers list, and then mouse right-click it (Figure 8.1-13). The popup save file dialog appears and you can decide a saving path to save it as a csv file.

The lower part of the window is used for display all added layers on a map. On the left side is **Map Tools,** which are used for manipulate the map, e.g., zoom in, zoom out, and full extent (Figure 8.1-12). To change projection, right-click **Map Layers** to select **Projection**. On the popup **Map Projection** window, click **Change Projection** at the left bottom to enter into **Select Projection** window. You can select a projection and then click **OK** to finish changing projection (Figure 8.1-14).

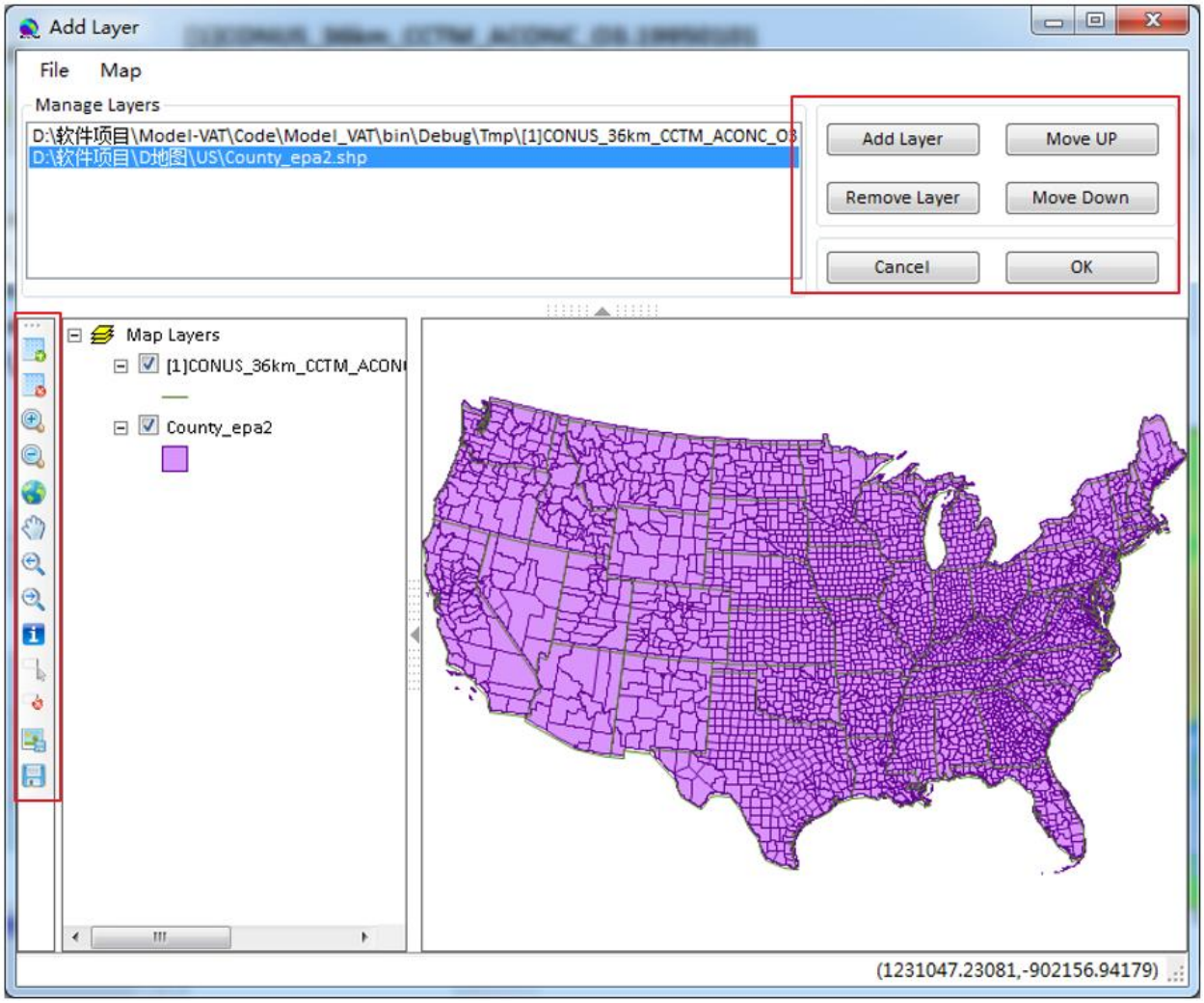


Figure 8.1-12. Add Layer Window

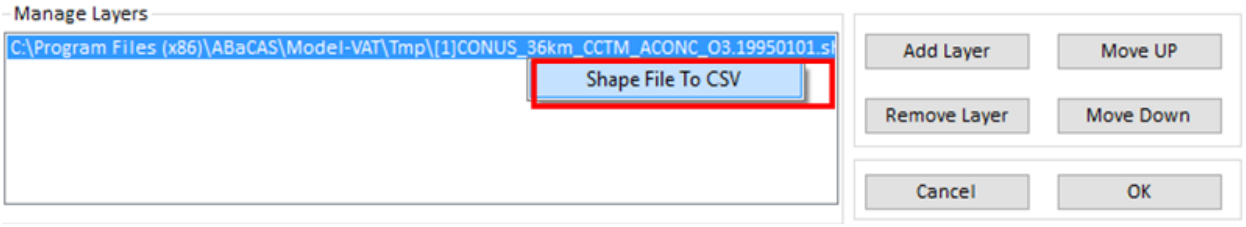


Figure 8.1-13. Save Layer File as CSV File

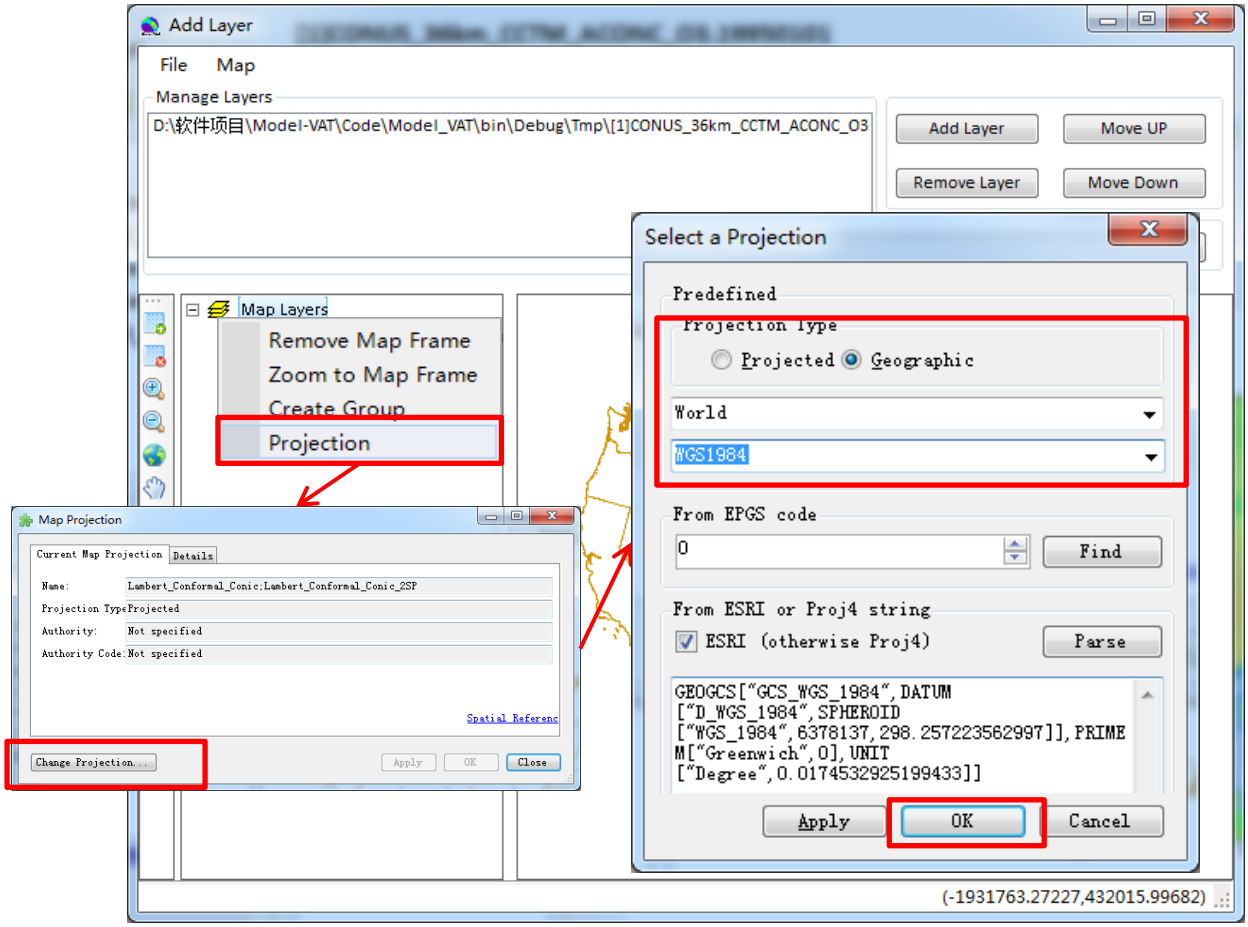
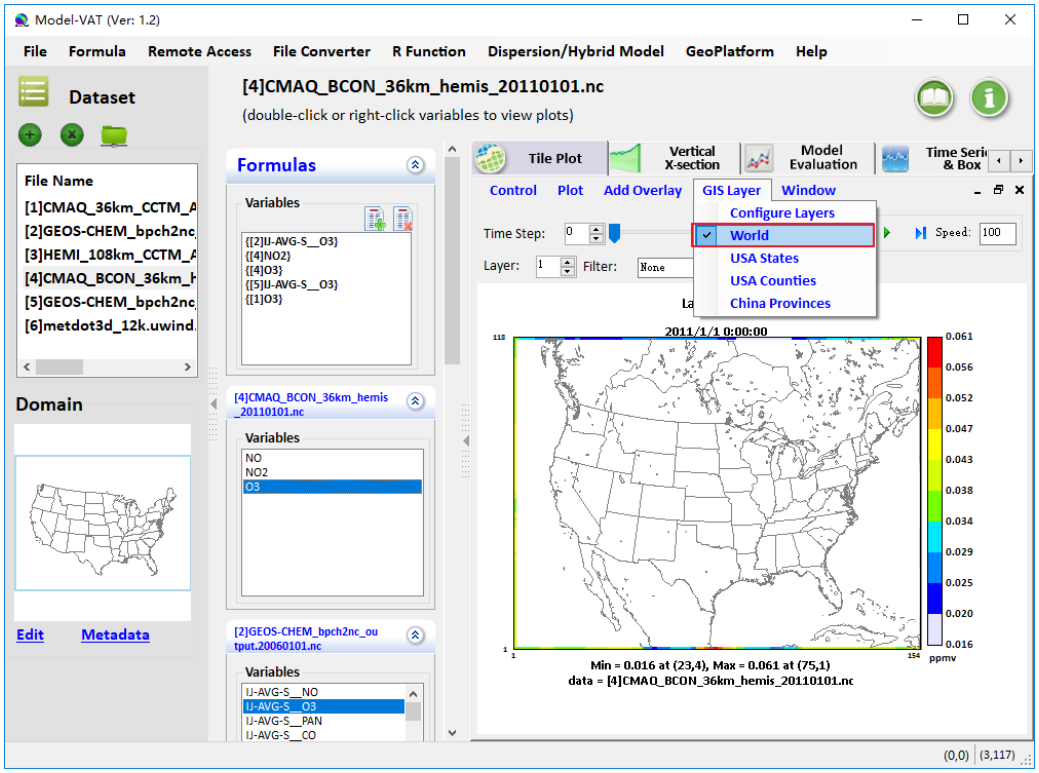
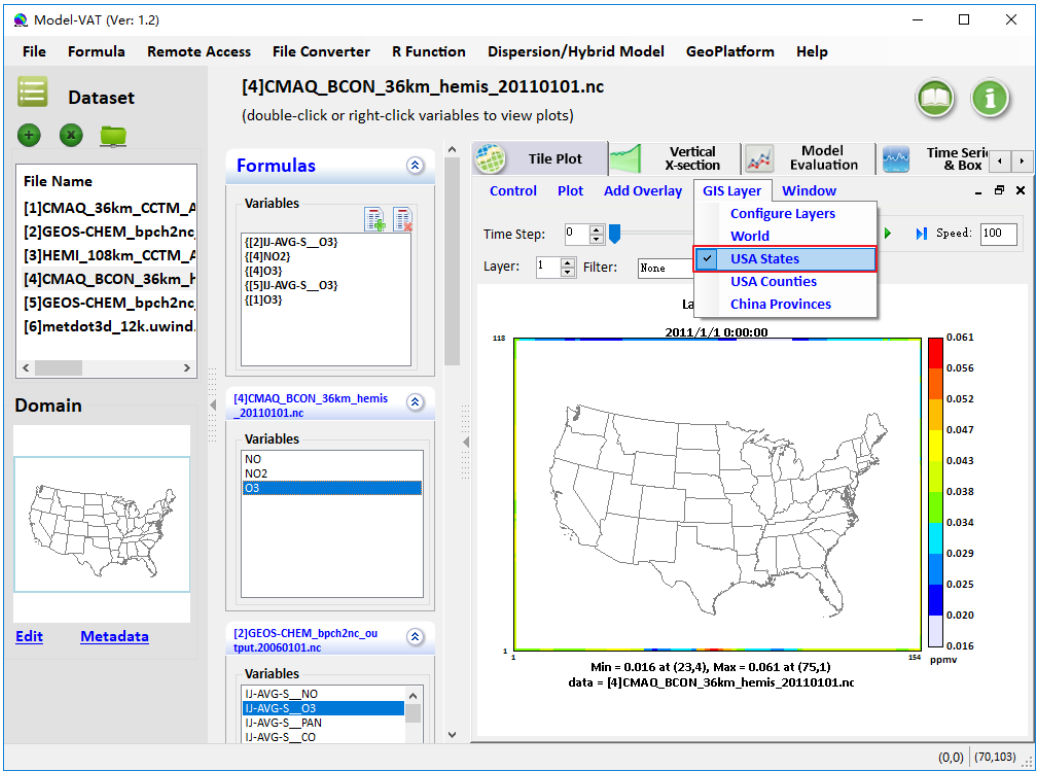


Figure 8.1-14. Change Projection

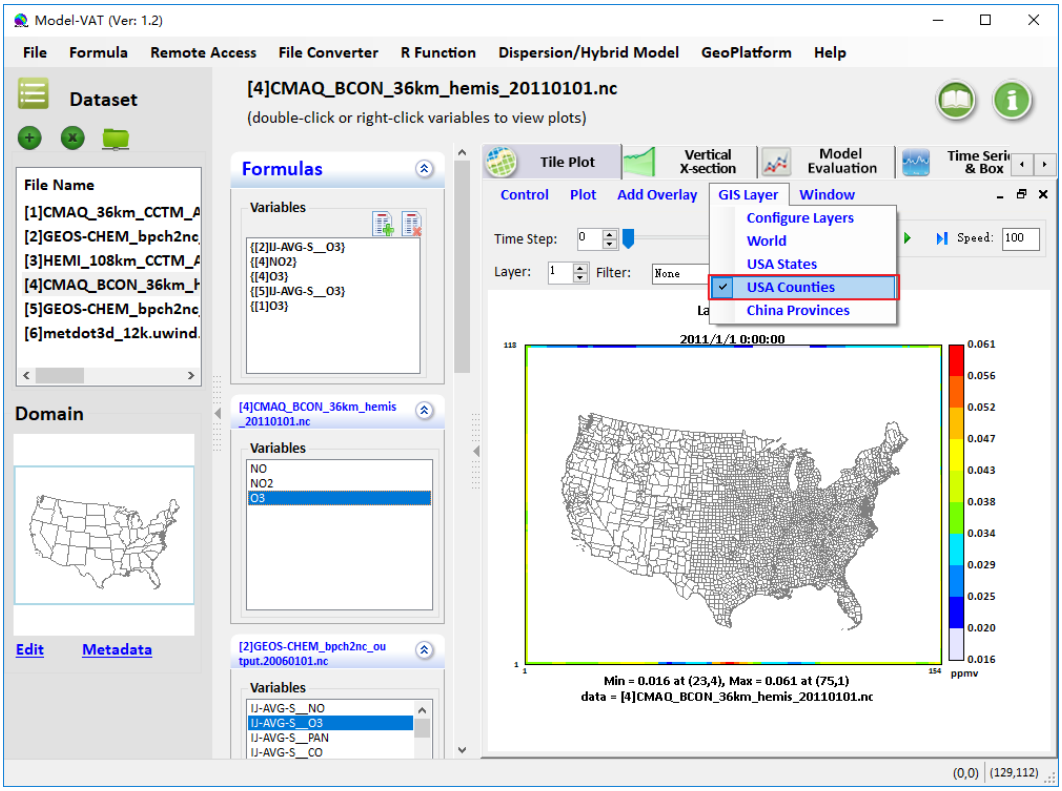
Use **World, USA States, USA Counties**, and **China Province** in the GIS Layer to rich the map as show as Figure 8.1-15.



a. World



b. USA States



c. USA Counties

Figure 8.1-15. GIS Layer

8.2 Vertical X-section

The **Vertical X-section** allows you to show a slice of data. There are two ways to add vertical cross section plot:

1) Mouse right-click a variables in the list box of variable, and select **Add Variable to Vertical Cross** (Figure 8.2-1).

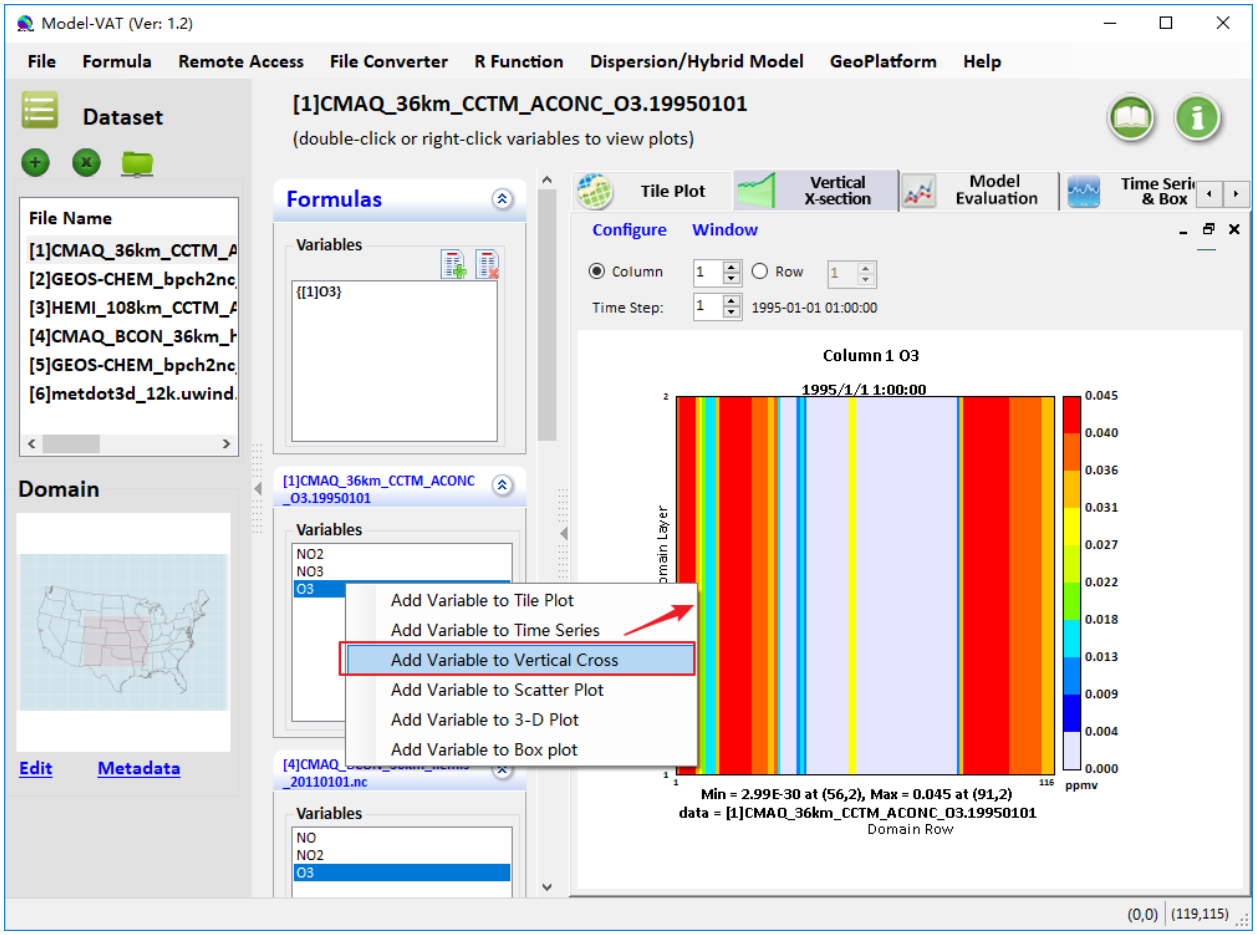


Figure 8.2-1. Select a Variable to Add Vertical Cross Section Plot

2) Switch to **Vertical X-section** tab page and double-click a variables in the list box of variable (Figure 8.2-2).

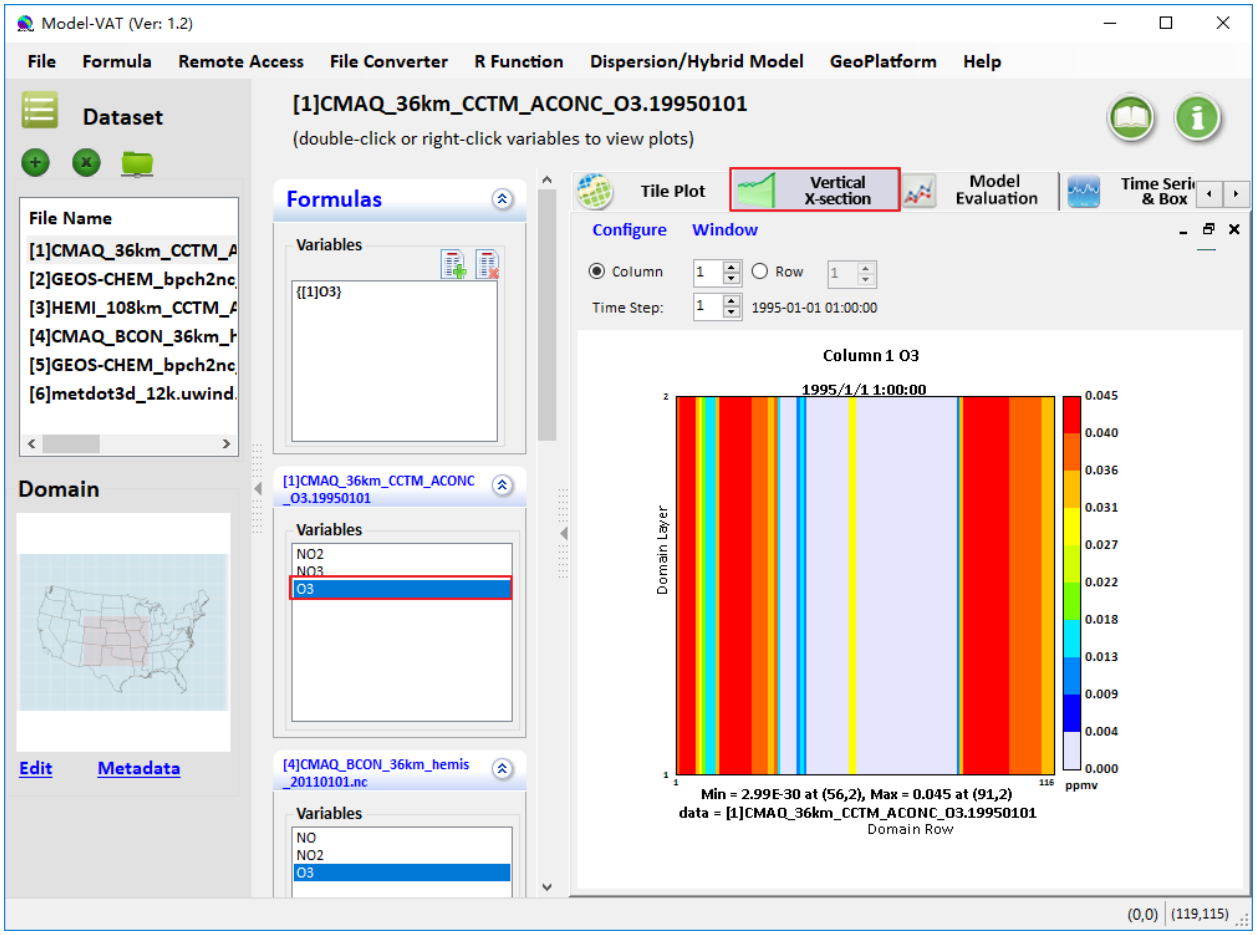


Figure 8.2-2. Add Vertical Cross Section Plot under Vertical Cross Section Plot Tab Page

Enter either The current time step on the plot can be changed. Using the **Time Step** spinner control above the plot, which also changes the date and time shown in the bottom of the plot. There is also a **Column** (or **Row**) spinner control to change the column number (or row number). The column is used for an *x*-axis cross section and the row is used for a *y*-axis cross section. The cross-section column number (or row number) is included in the title of the plot and changes as you change the spinner control.

8.3 Model Evaluation

The **Model Evaluation** can verify the reliability of the model simulation results. The comparison between the simulated value and the observed value is generally divided into two aspects. On one hand, the difference between the two values is compared, and on the other hand, the difference between the two trends in the concentration change is compared. Besides, it provides a visualization display of the results in the form of tile plots, time series or plots scatter plots. An example of the Model Evaluation window is shown in Figure 8.3-1.

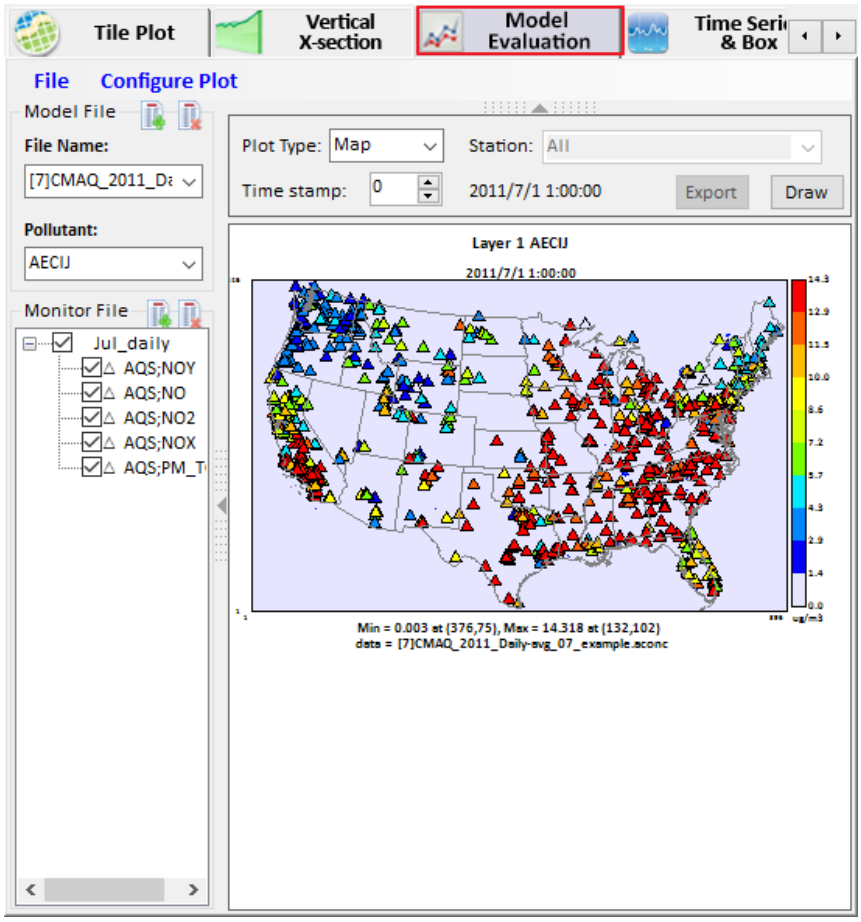


Figure 8.3-1. Model Evaluation Window

There are main 4 steps to use **Model Evaluation** module (Figure 8.3-2):

1) Add and select model files/pollutants
2) Add and select monitor or site compare (sitecmp) files/pollutants
3) Set drawing option
4) Draw plot

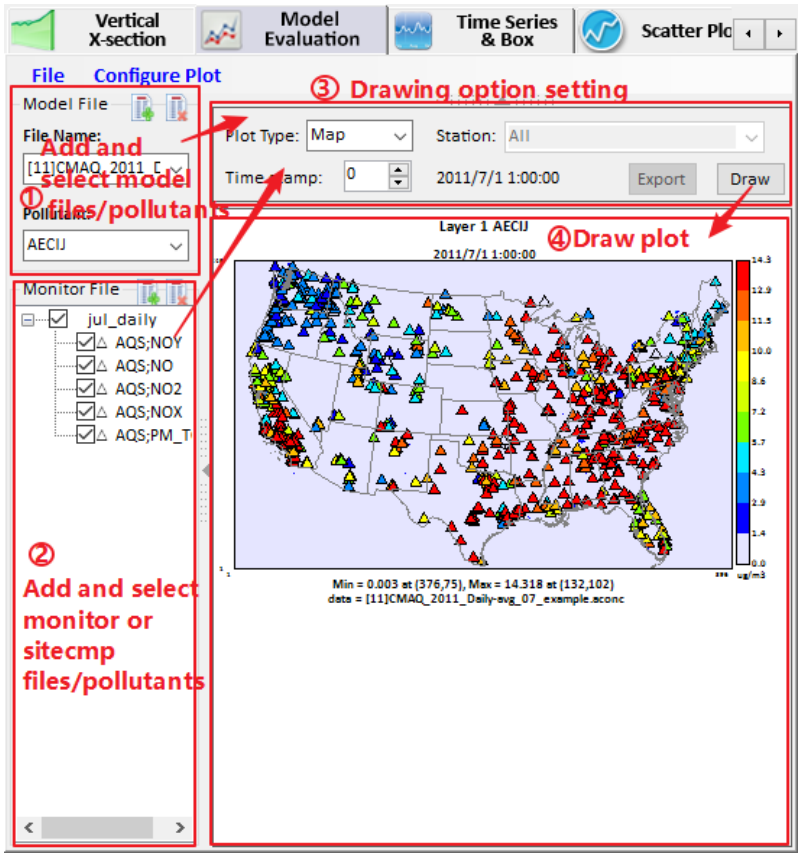
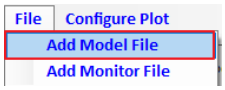


Figure 8.3-2. The process of using Model Evaluation module

8.3.1 Add and select model files/pollutants

Click  or click 

Add Model File to add and select model files/pollutants (Figure 8.3-3).

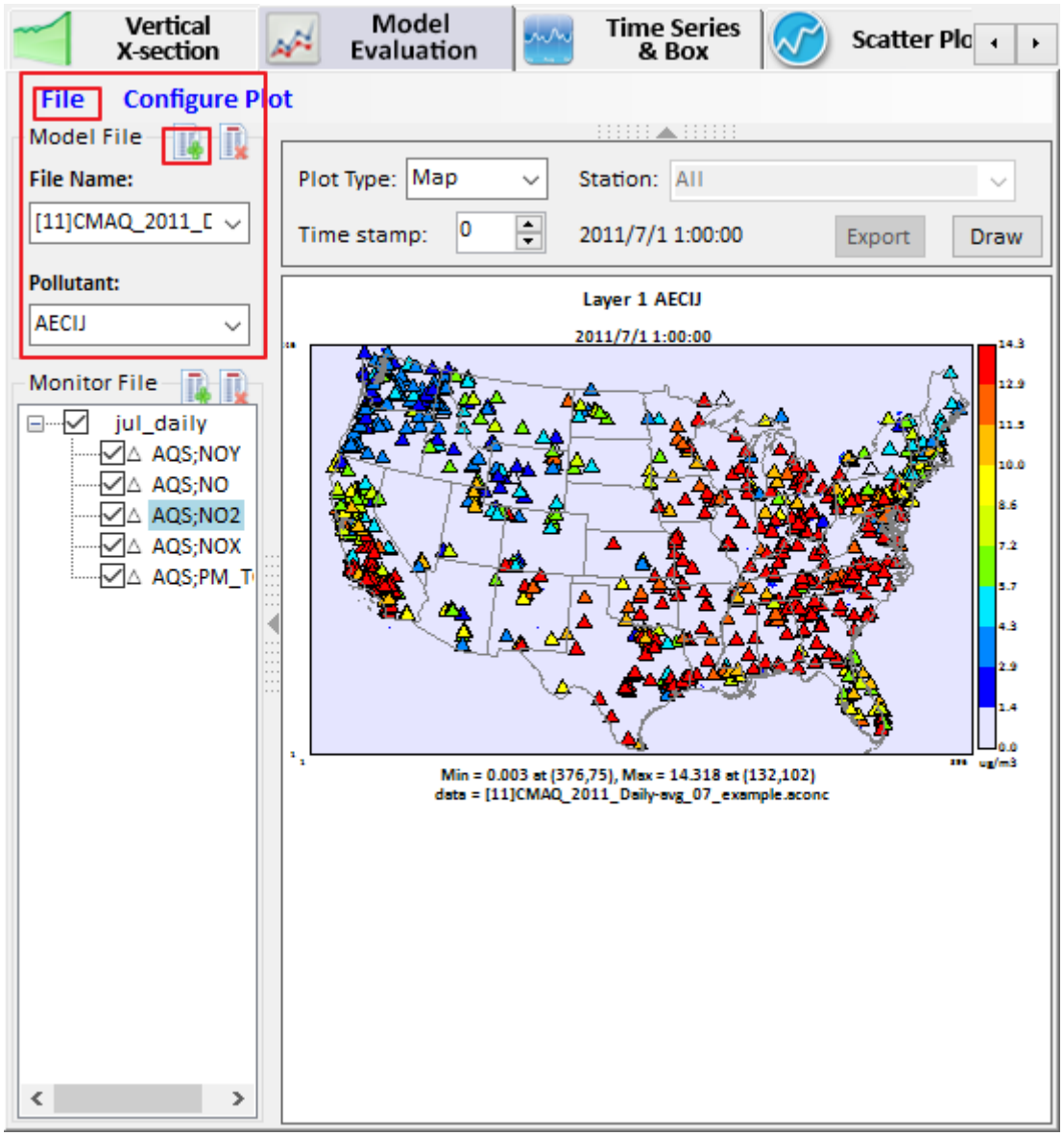
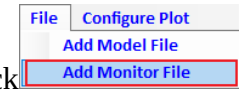


Figure 8.3-3 Choose model file/pollutant

8.3.2 Add and select monitor or site compare files/pollutants

Click  or click 

Add Monitor File to open **Model Evaluation Input Editor** Window to add and select monitor or site compare files/pollutants (see Figure 8.3-4).

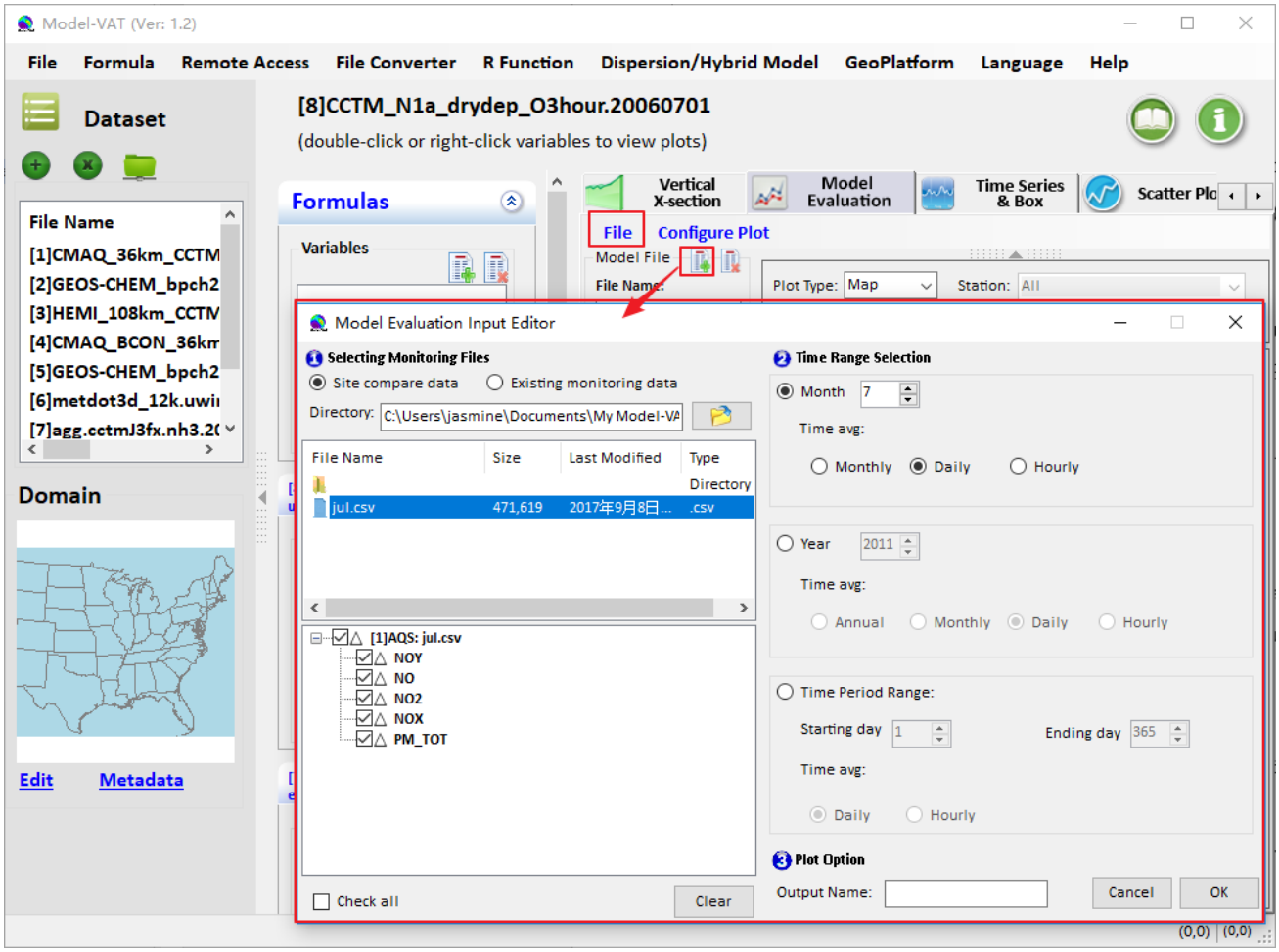


Figure 8.3-4 Open Model Evaluation Input Editor Window

Notes:

The initial directory is C:\Users\xxx\Documents\My Model-VAT Files\Data\Sitecmp\AQS.

8.3.3 Set drawing option

1) Choose **All**: average value of all stations in the sitecmp files at each time stamp (Figure 8.3-5).
2) Single Station: value of a single station at each time step, as Figure 8.3-6 shows. Only choose **Time Series plot** can use **Single Station**.

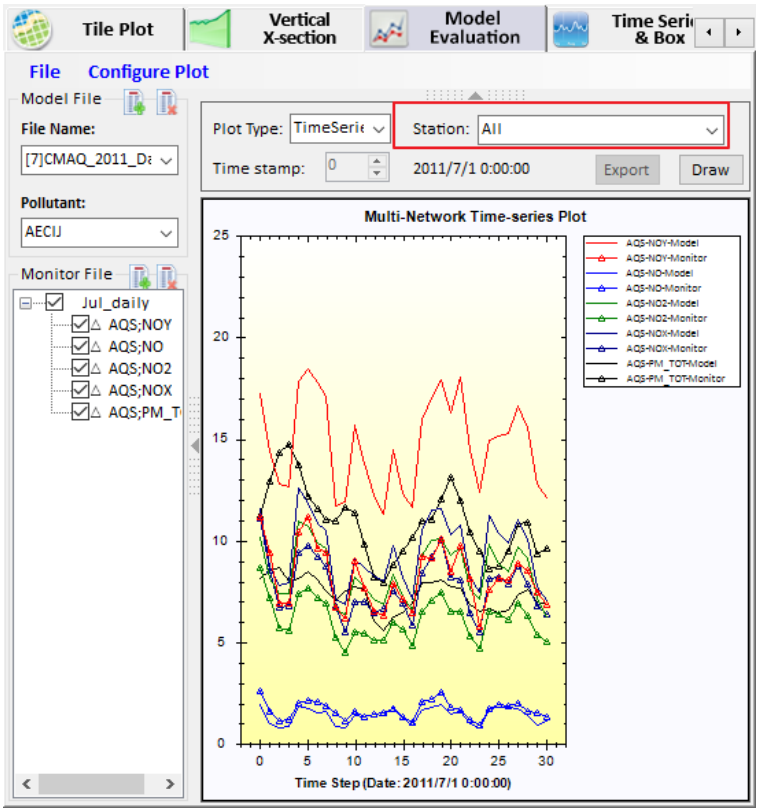


Figure 8.3-5 Time series plot (all stations)

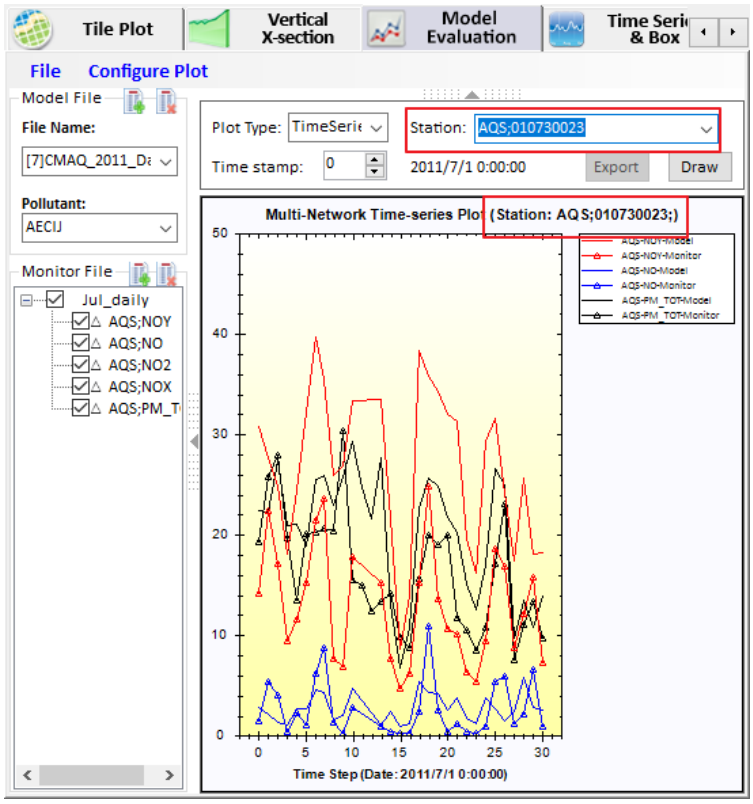


Figure 8.3-6 Time series plot (single station)

8.3.4 **Draw plot**

Legend has been changed to the right as default. You can choose **Change title and legend** to change the legend position, as Figure 8.3-7 shows.

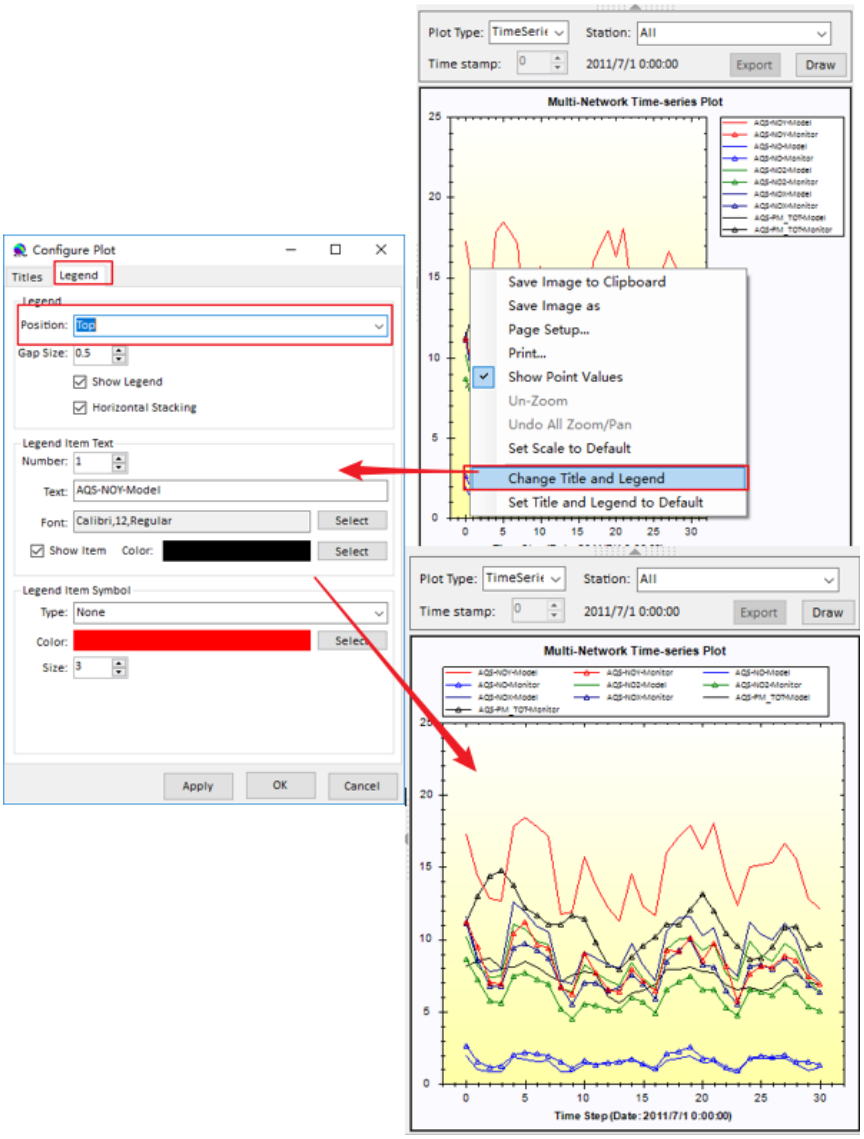


Figure 8.3-7 Select the legend position

8.4 Time Series & Box

8.4.1 **Basic Operation**

The **Time Series & Box** can show a line graph with the average values over time. The plot is made for the formula s selected domain and layer range. Each time step s data are averaged linearly to produce that time step s data point.

There are three ways to add **time series plot**:

1) Select a grid cell at the map of Tile Plot and then click **Plot** Time **Series** Time **Series of Probed Cell(s)** (Figure 8.4-1).

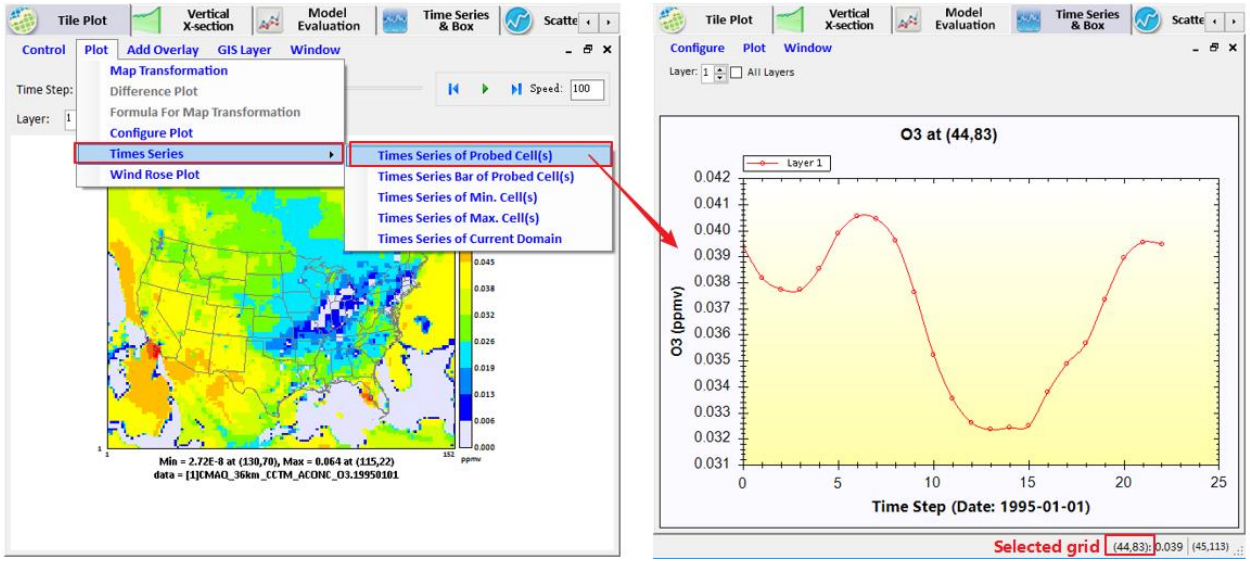


Figure 8.4-1. Select a Grid Cell to Add Time Series Plot

2) Mouse right-click a variable in the list box of variable, and select **Add Variable to Time Series** (Figure 8.4-2).

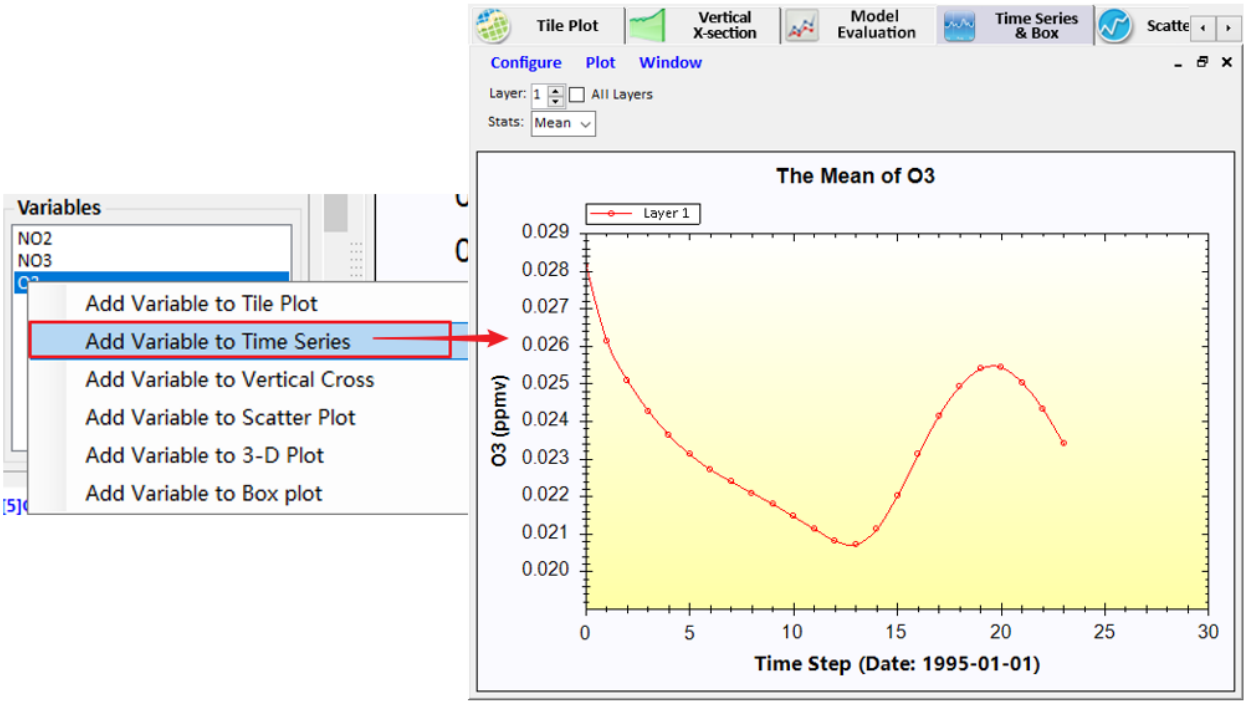


Figure 8.4-2. Select a Variable to Add Time Series Plot

3) Switch to **Times Series** tab page and double-click a variables in the list box of variable (Figure 8.4-3).

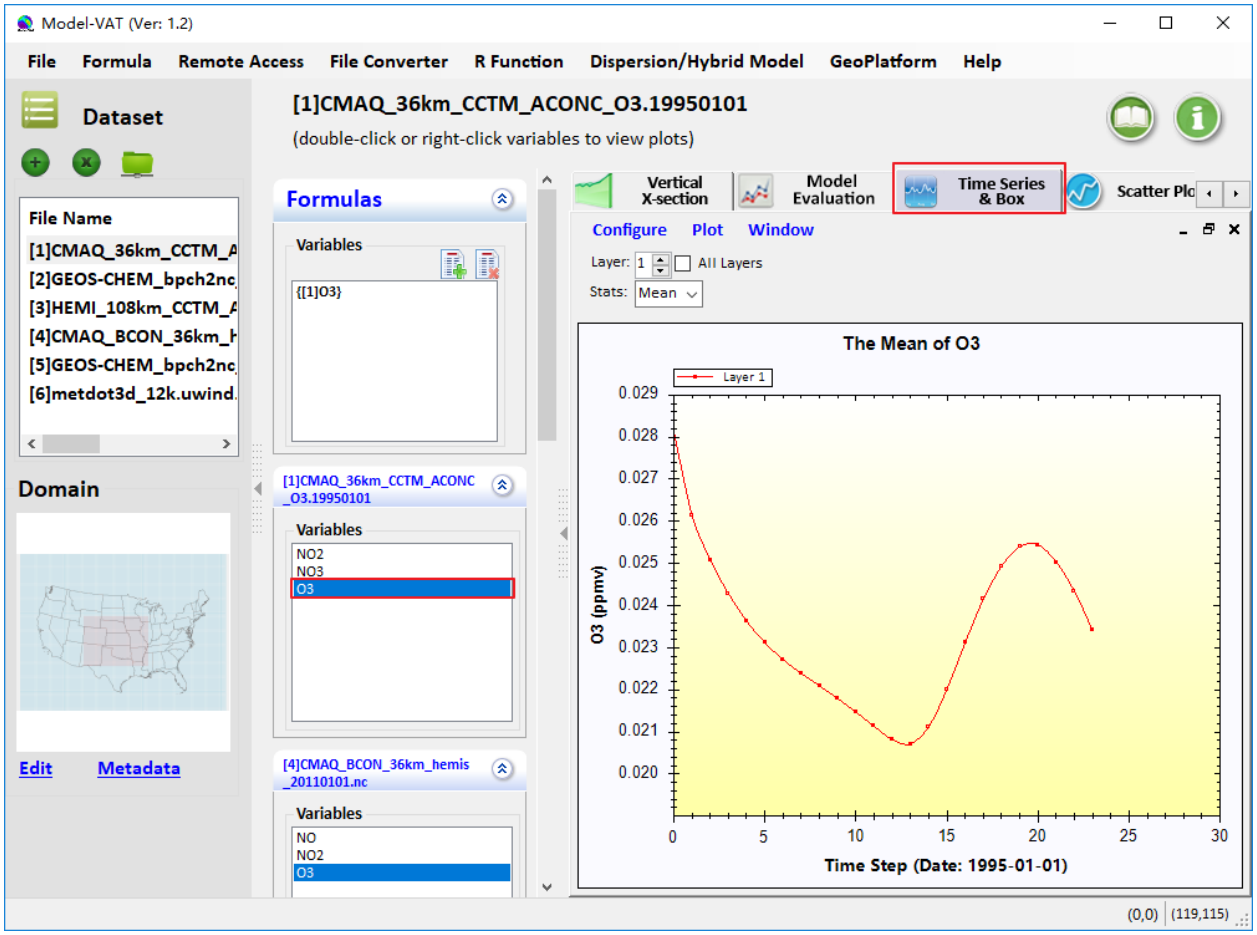


Figure 8.4-3. Add Time Series Plot under Time Series & Box Tab Page

There are four kind of styles of time series plot: **smooth line**, **step line**, **non-step line**, and **bar chart**. They can be found on the popup menu of **Plot** menu button at the top.

Smooth Line: Connect points with smooth curve, as shown in Figure 8.4-4.

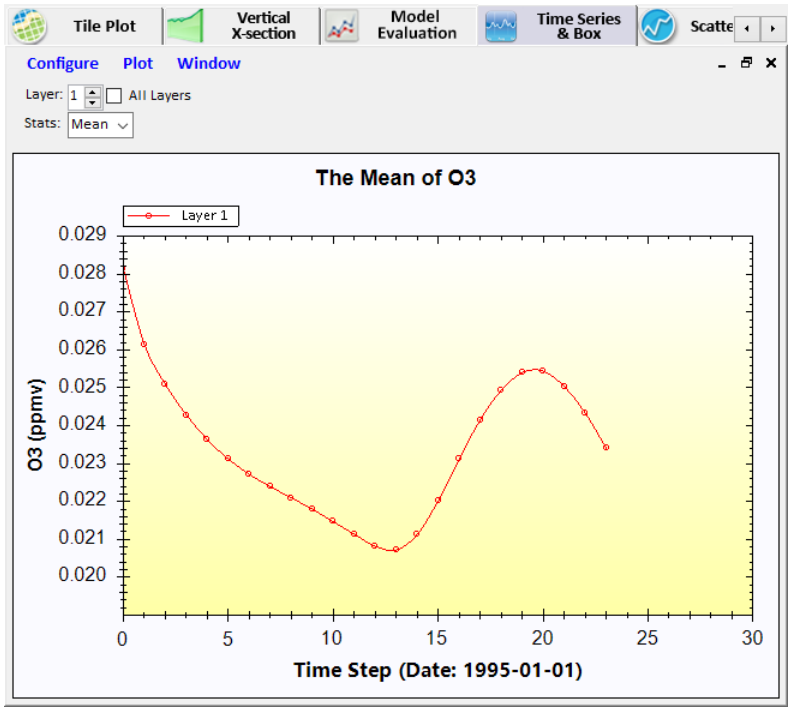


Figure 8.4-4. Smooth Line

Step Line: It is divided into **forward step line** and **rearward step line** (Figure 8.4-5).

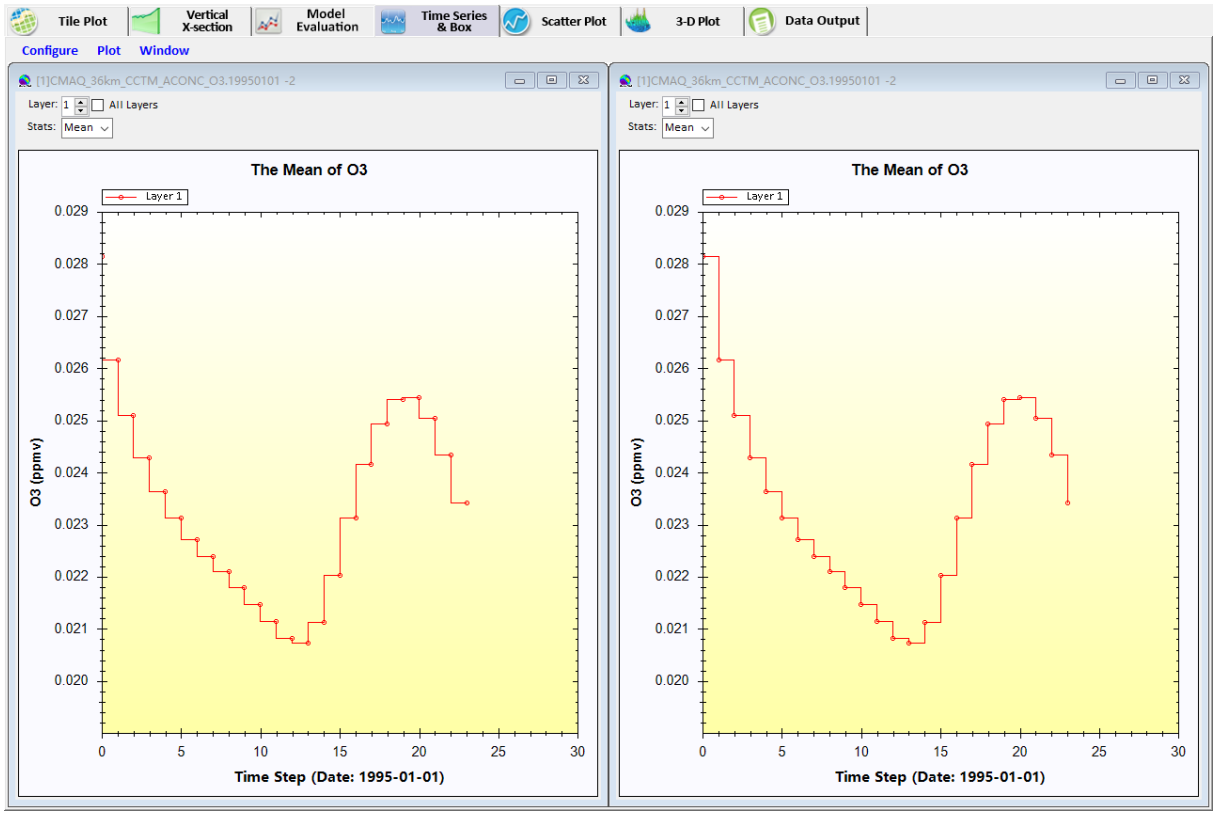


Figure 8.4-5. Forward Step Line (left) and Rearward Step Line (right)

Non-Step Line: Connect points with line, as shown in Figure 8.4-6.

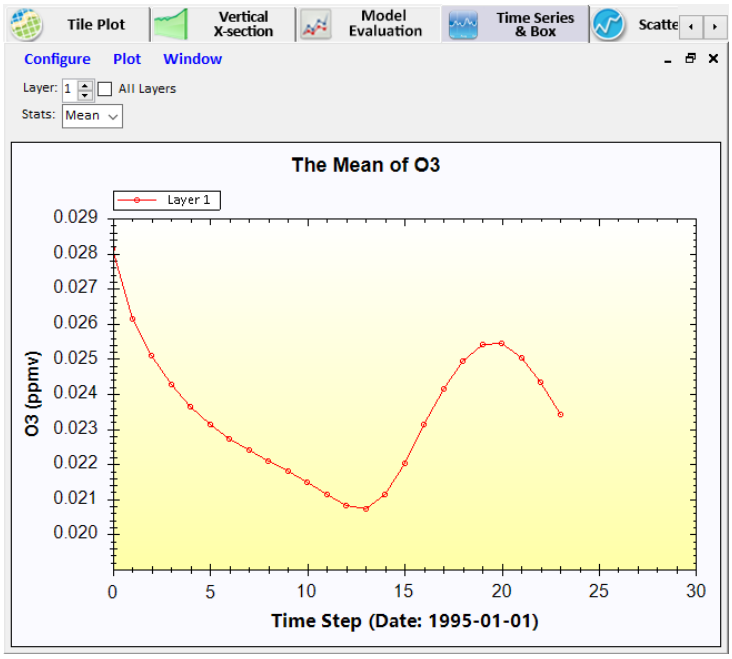


Figure 8.4-6. Non-Step Line

Bar Chart:

The time series bar chart shows average values over time in a bar plot format (Figure 8.4-7) rather than a line format. Other than that, the description of this plot type is the same as for the time series line plot.

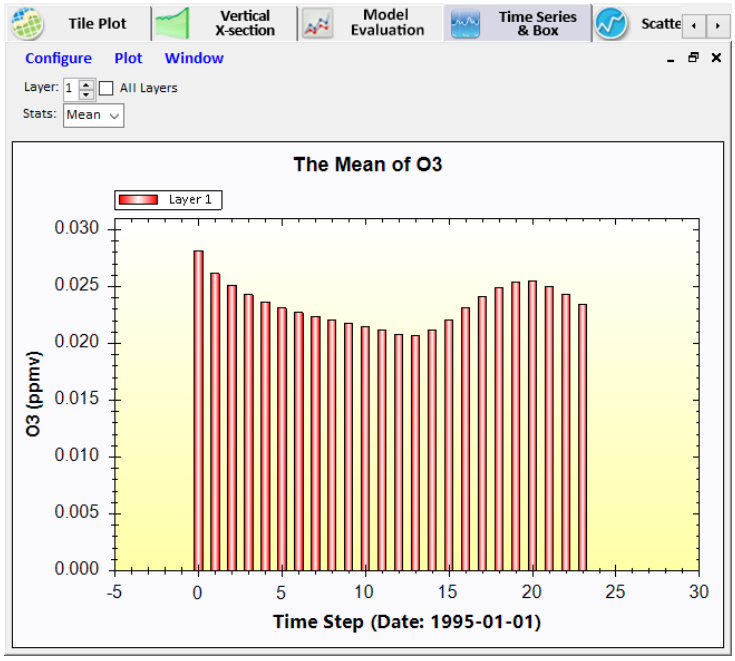


Figure 8.4-7. Non-Step Line

There are two ways to add **Box Chart**:

1) Select a grid cell at the map of **Tile Plot** and then click **Plot** Time **Series** Time **Series Bar of Probed Cell(s)** (Figure 8.4-8).

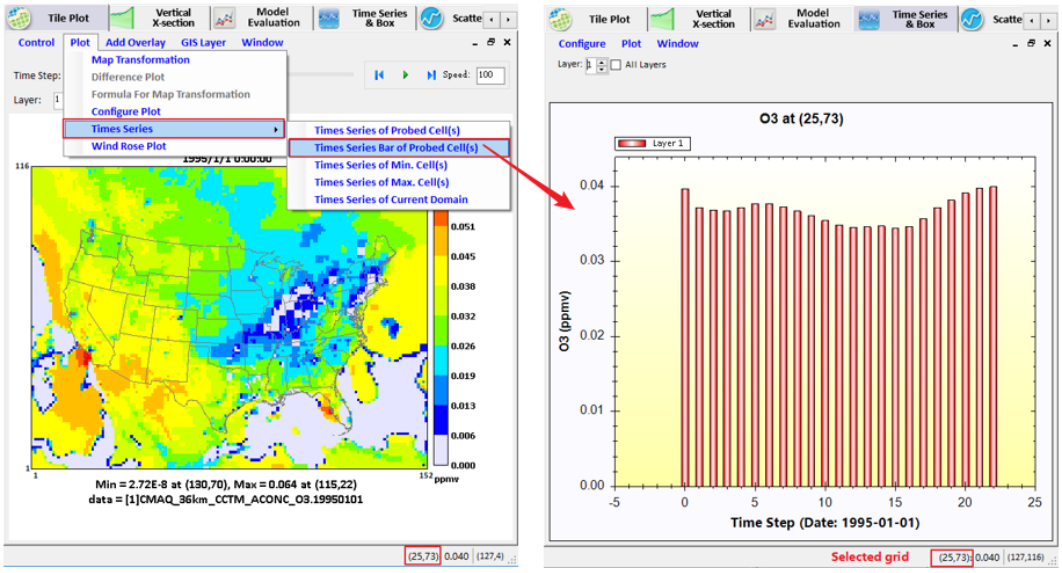


Figure 8.4-8. Select a Grid Cell to Add Bar Chart

2) Transform Time Series Plot to Box Chart by Clicked the **Bar Chart** under **Plot** in the **Time Series & Box** (Figure 8.4-9).

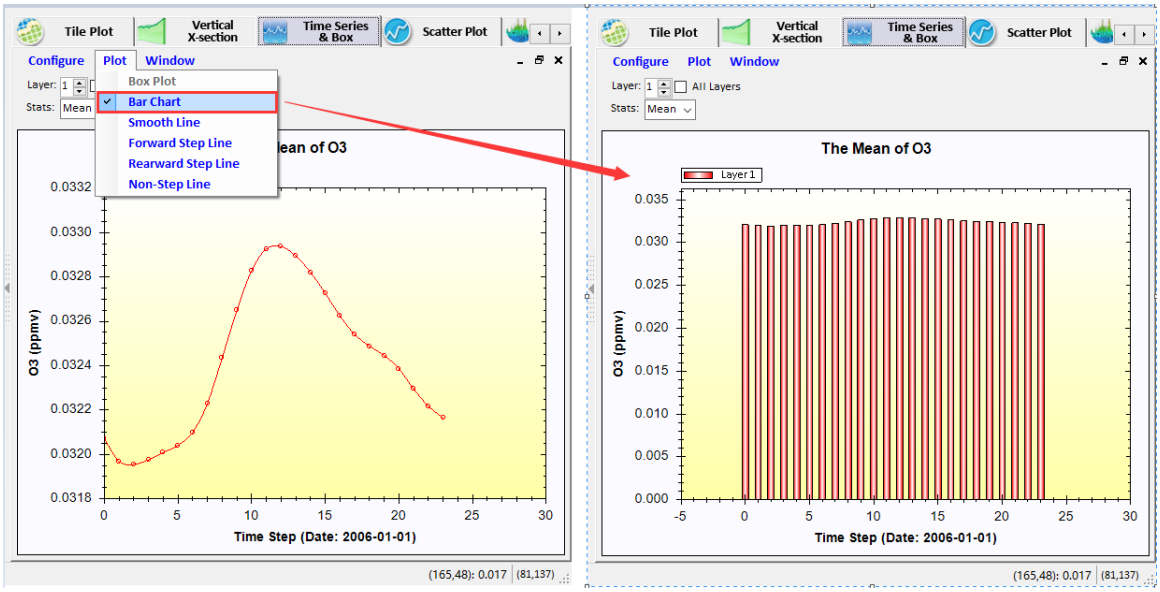


Figure 8.4-9. Transform Time Series Plot to Bar Chart

There are two ways to add **Box Plot**:

1) Mouse right-click a variable in the list box of variable, and select **Add Variable to Box Plot** (Figure 8.4-10).

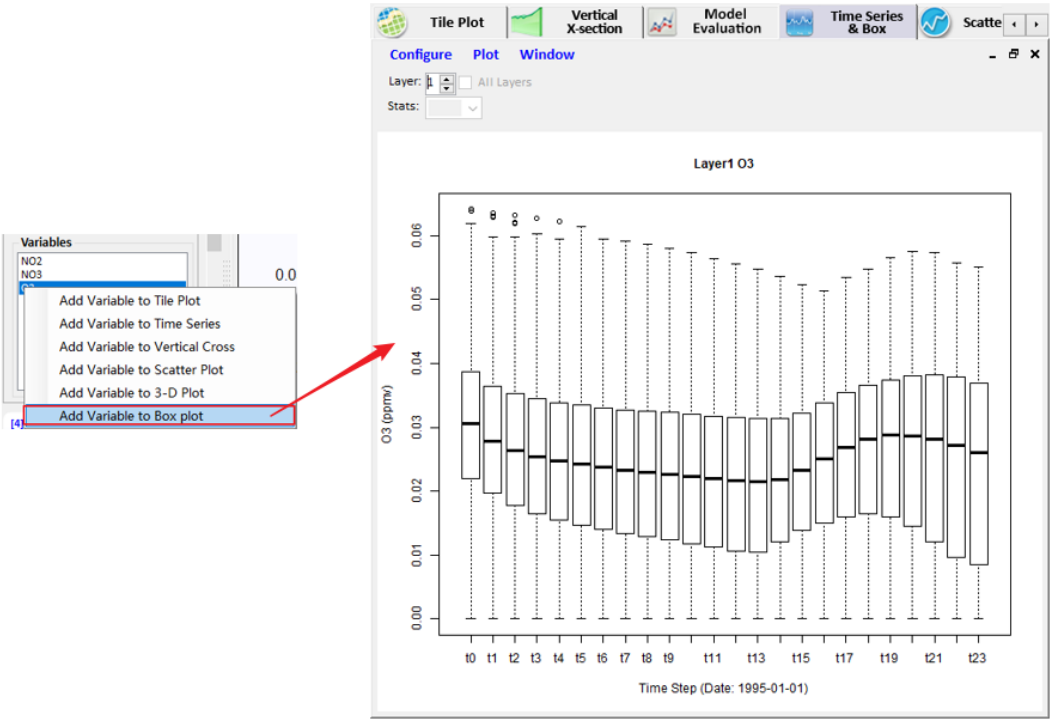


Figure 8.4-10. Select a Variable to Add Box Plot

2) Transform time series plot to box plot by Clicked the **Bar Plot** under **Plot** in the **Time Series & Box** (Figure 8.4-11).

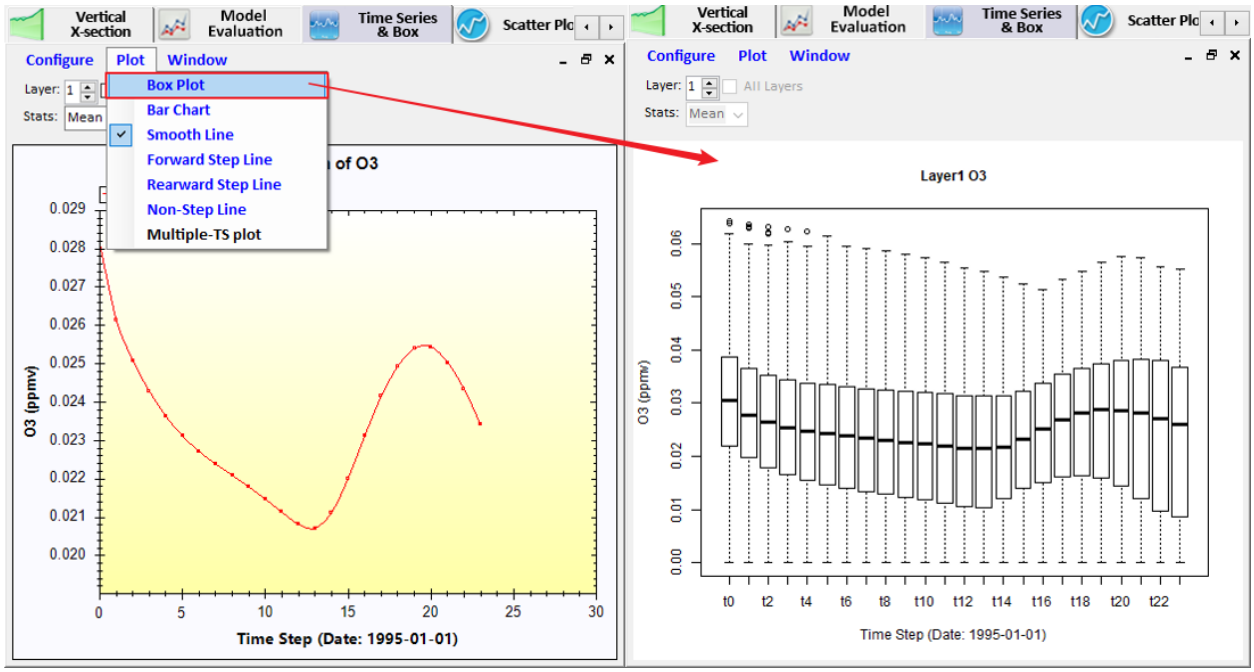


Figure 8.4-11. Transform Time Series Plot to Bar Chart

The current layer can be changed using the **Layer** spinner control above the plot. The layer value listed in the title is updated when you change the layer. If you want to see all layers plots, you can tick the **All Layers** (Figure 8.4-12).

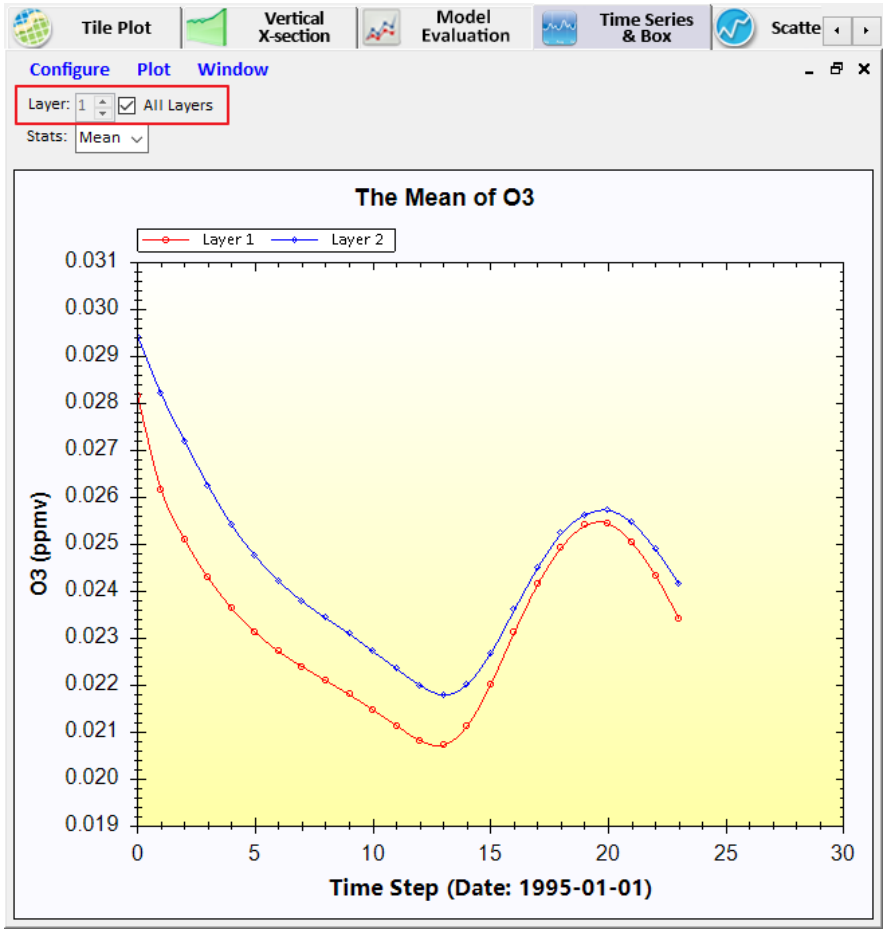


Figure 8.4-12. Plot Shows All Layers

8.4.2 Grid Cell Time Aggregate Statistics

The pull-down menu option labeled **Stats** provides the option to display grid cell time-aggregate statistics. In this version, the function is only enabled in time series plot (Figure 8.4-13).

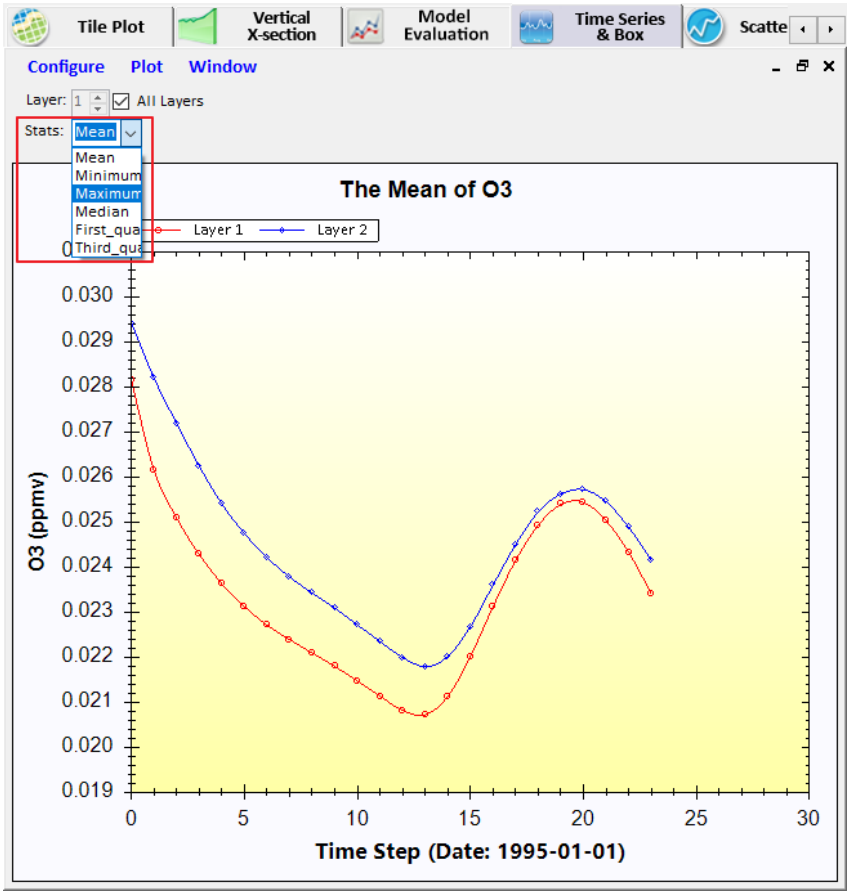


Figure 8.4-13. Plot Shows Different Stats

Model-VAT calculates the grid cell time aggregate statistics as follows: For each cell (*i,j,k*) in the currently selected domain (independent of neighboring cells), the aggregated statistical value is calculated over the currently selected time steps. In other words, the aggregated statistical value is calculated for the plotted formula for cells (*i,j, k,tmin*...tmax), with the number of time steps *n*, where *n*= (tmax-tmin+1).

- MINIMUM: min (var(i,j,k,tmin), var(i,j,k,tmin+1), ..., var(i,j,k,tmax))
- MAXIMUM: max (var(i,j,k,tmin), var(i,j,k,tmin+1), ..., var(i,j,k,tmax))
- MEAN: SUM / n
- MEDIAN: value at 50th percentile of (sorted {var(i,j,k,tmin), var(i,j,k,tmin+1), ..., var(i,j,k,tmax)})
- FIRST_QUARTILE: value at 25th percentile of(sorted (var(i,j,k,tmin), var(i,j,k,tmin+1), ..., var(i,j,k,tmax)))
- THIRD_QUARTILE: value at 75th percentile of(sorted (var(i,j,k,tmin), var(i,j,k,tmin+1), ..., var(i,j,k,tmax)))

8.5 Scatter Plot

The **Scatter Plot** shows the relationship between two formulas using dots (Figure 8.5-1). There are two ways to add a scatter plot:

1) Mouse right-click a variable in the list box of variable, and select **Add Variable to Scatter Plot** (Figure 8.5-1).

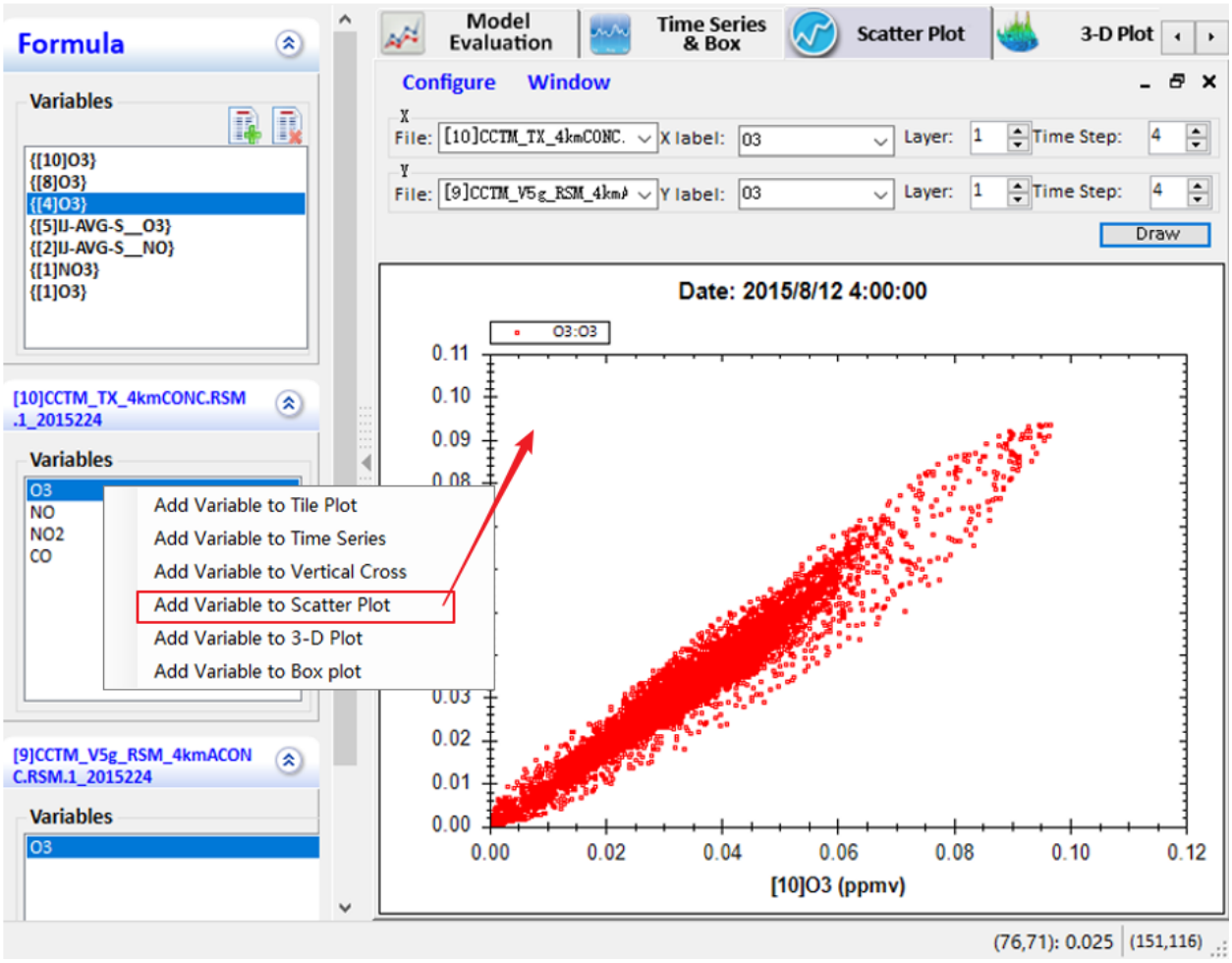


Figure 8.5-1. Select a Variable to Add Scatter Plot

2) Switch to **Scatter Plot** tab page and double-click a variable in the list box of variable (Figure 8.5-2).

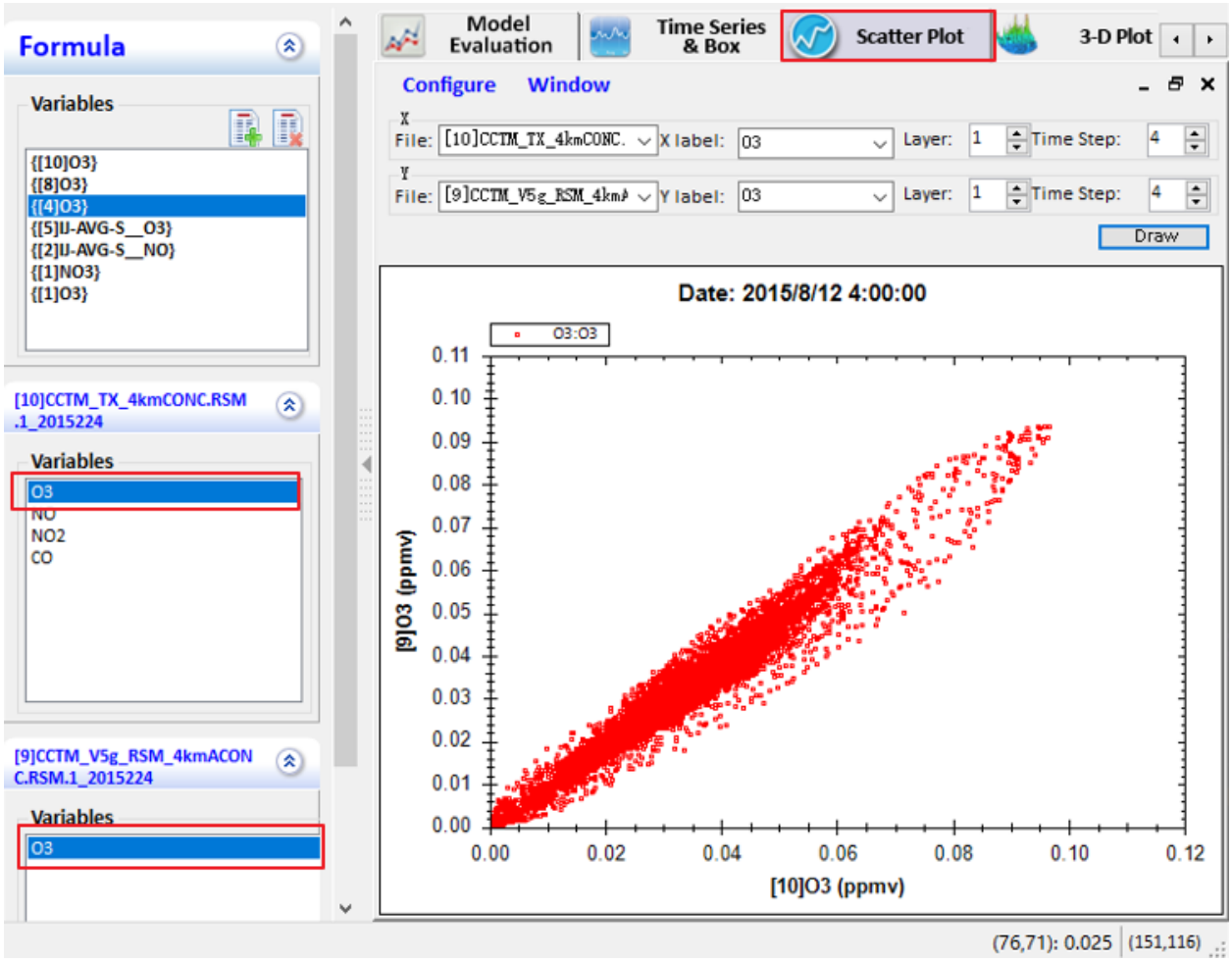


Figure 8.5-2. Add Scatter Plot under Scatter Plot Tab Page

You can specify X File, Y File, **X label, Y label, Layer and Time Step** at the top of the plot. The X label (or Y label) can be adjusted by the pull-down menu. The current time step and layer can be adjusted using the spinner controls.

8.6 3-D Plot

The **3-D Plot** displays the data with stereogram (Figure 8.6-1). There are two ways to add a scatter plot:

1) Mouse right-click a variable in the list box of variables, and select **Add Variable to 3-D Plot** (Figure 8.6-1).

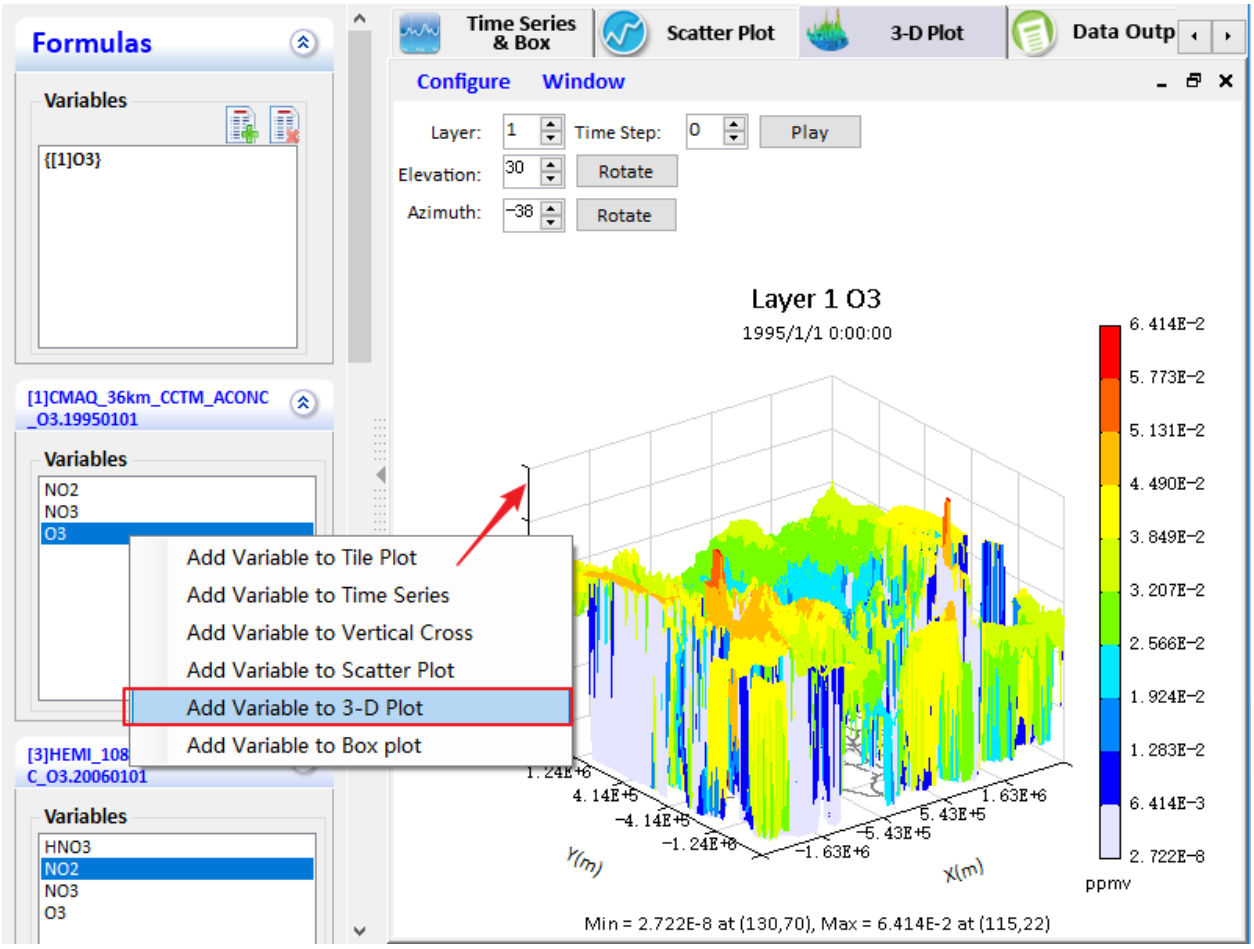


Figure 8.6-1. Select a Variable to Add 3-D Plot

2) Switch to **3-D Plot** tab page and double-click a variable in the list box of variable (Figure 8.6-2).

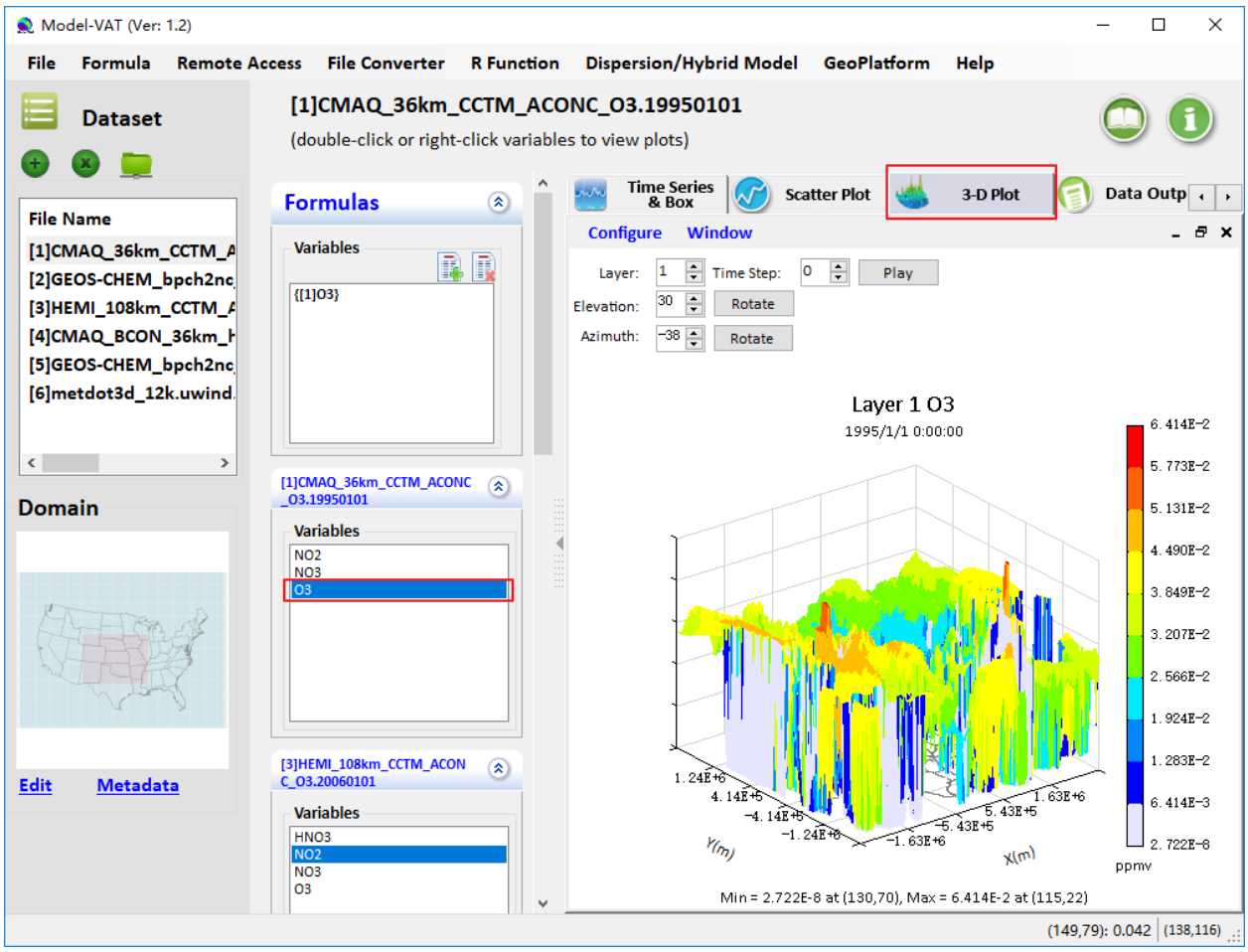


Figure 8.6-2. Add 3-D Plot under 3-D Plot Tab Page

8.7 Data Output

The **Data Output** shows detailed data of variables of selected dataset. In the **Data** tab page**,** click a dataset on **Dataset** pane and its variables will be loaded (Figure 8.7-1). You can check one or more variables and click **Apply**, and the values of these variables will be shown in the right data table. Time step and layer can be adjusted using spinner controls below the list of variables. Users can export selected data by clicking **Export** button.

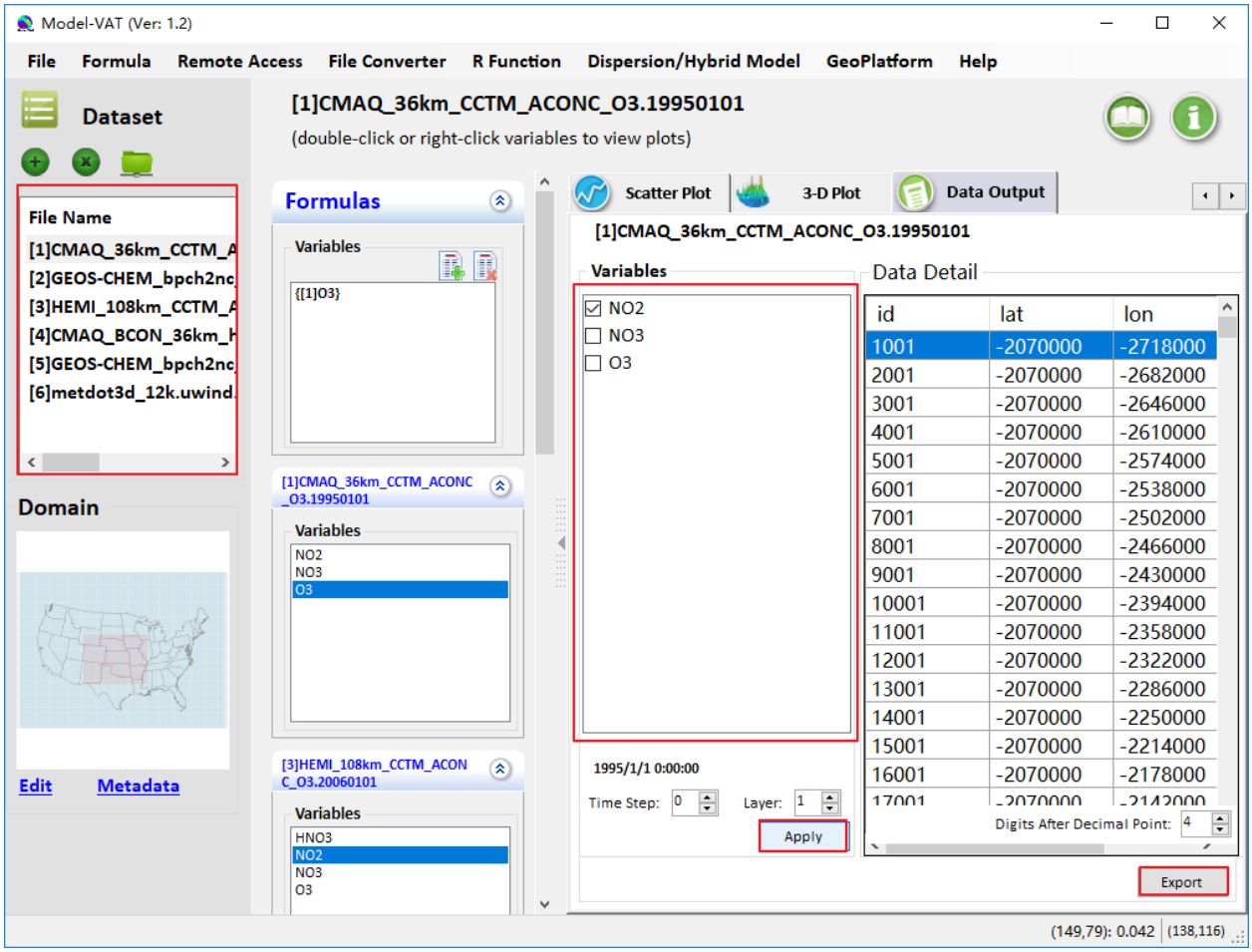


Figure 8.7-1. Data Output Tab Page

8.8 Color Map

This tab provides many widgets for you to configure your legend colors, break points, range, etc. Users can select the number of tiles, the palette type to be used, the color interval, the number format, and the scale. Double-click the legend or click the **Configure Plot** under the plot to open Configure Plot window (Figure 8.8-1).

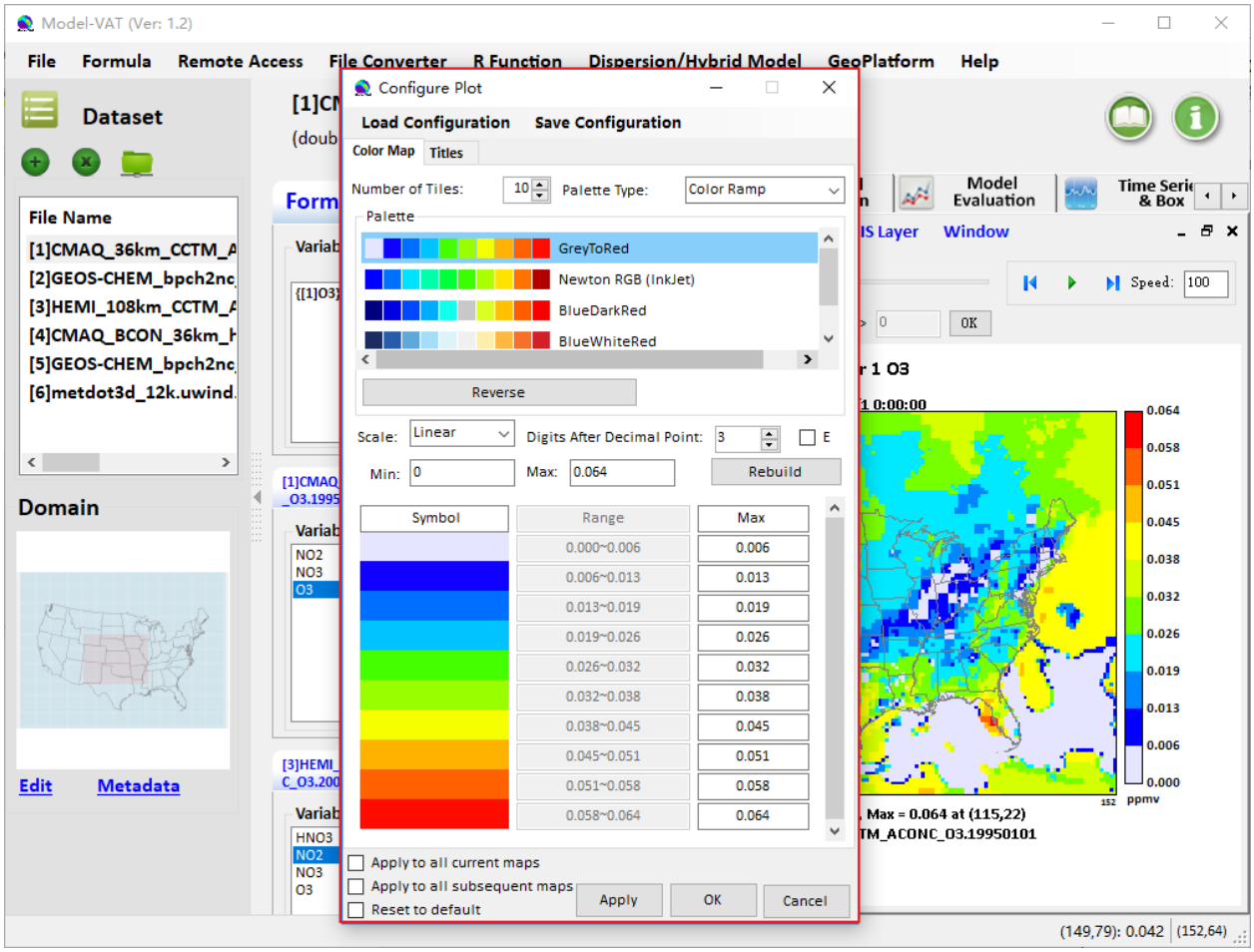


Figure 8.8-1. Configure Plot, Color Map Tab

- **Load Configuration:** Click to open an existing configuration in user's computer.
- **Save Configuration:** Click to overwrite the old configuration.
- **Number of Tiles:** Start by selecting the number of tiles. Options in some other selections, such as the available palettes and break points, change as you vary the number of tiles.
- **Palette Type:** Four palette types are available: Color Ramp, Single Sequential, Double Sequential, and R Scale. Color Ramp has a mixture of colors. Single Sequential has a dark color on one end (typically the maximum) with lighter shades of the same color proceeding to the lightest shade at the minimum. Double Sequential has two dark colors at the maximum and minimum of the range and light colors in the middle. R Scale be printed in grayscale and color-blind friendly which are available in an R package. The color palette that is frequently associated with air quality modeling results is one of the Newton RGB palettes of the Sequential type.
- **Reverse:** The reverse button reverses the order of the colors in the selected palette. For example, if a sequential palette is used, the reverse button changes the color intensities such that the darkest color is at the minimum instead of the maximum of the scale.
- **Scale:** Select either Linear or Logarithmic. The interval start values are automatically adjusted when you change the scale.
- **Digits after Decimal Point:** Adjust the digits after decimal point.

- **Min** and **Max:** The minimum and the maximum values are computed for the data to be plotted. If you set the Interval to Automatic, you can change the values for the minimum and maximum. Then, press the Rebuild button and the Apply button to see your changes both in the legend and on the map. You cannot edit the minimum or the maximum value if the Interval is set to Custom. Instead, directly edit the Interval Start values; the Interval Start value for the lowest interval is the same as the minimum value.
- **Rebuild:** The Rebuild button is either active or inactive (i.e., grayed out) depending upon what other widgets are active. If you make changes and the Rebuild button is active, press it before continuing.
- **Apply for all maps:** Check to apply the old configuration for all maps.

9 Map Transformation and Resolution

Generally, different air quality models are in different output file formats, grid resolutions, model fields, coordinate systems, etc. It is difficult to make the comparison of the differences between these heterogeneous model data. Different types of air model data must be consistent in these aspects in order to make a comparative analysis. The **Map transformation** of Model-VAT is developed to convert two or more different model output into the same projection and resolution to make comparison. In this module, it supports to compare with multi-scale models such as CMAQ, H-CMAQ, CAMx, WRF, and Geos-Chem, and hybrid modeling of CMAQ/AERMOD. Note that those binary model files (e.g., Geos-Chem) which are not in the Input/output Applications Programming Interface (I/O API) and netCDF format need to be converted to netCDF format first using the **File Converter** module.

9.1 Map Projections

On **Tile Plot** tab page, you can use **Map transformation** to unify two or more plots into the same projection and resolution. This contributes to comparison between plots with different projections.

To open map transformation window, there are two ways as follows:

1) Choose the active window and then click **Plot** to select map transformation. (Figure 9.1-1)

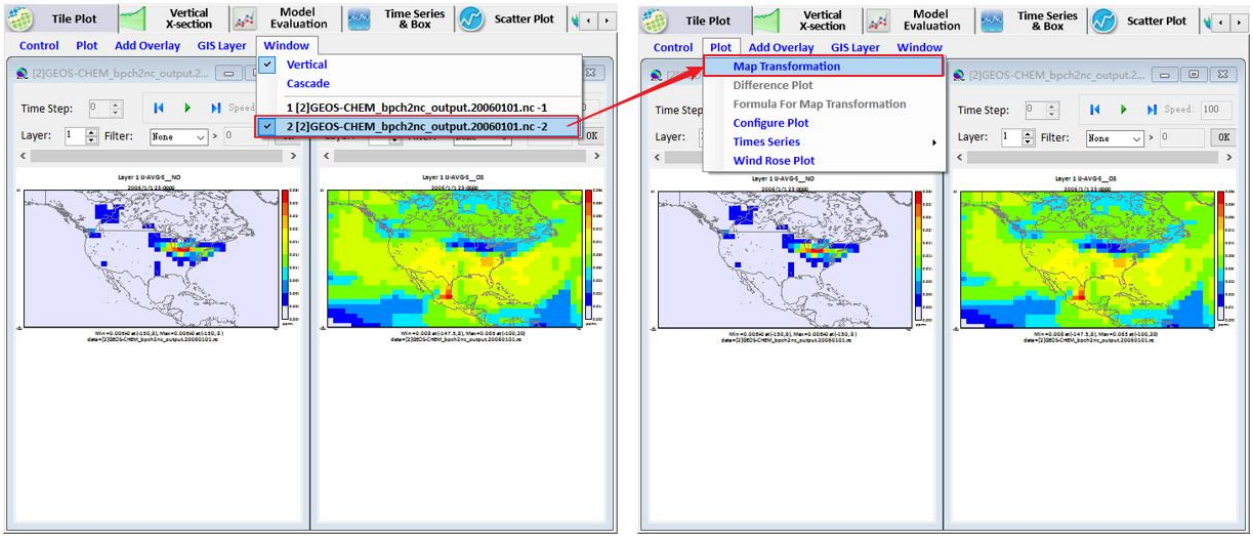


Figure 9.1-1. Choose an Active Window to Open Map Transformation

2) Right-click a plot and select map transformation.

On the popup Map Transformation window, it is divided into **Current Projection** and **Target Projection**. Both of them display their grid type, projection name, and details. Choose a target projection and click **OK** button (Figure 9.1-2).

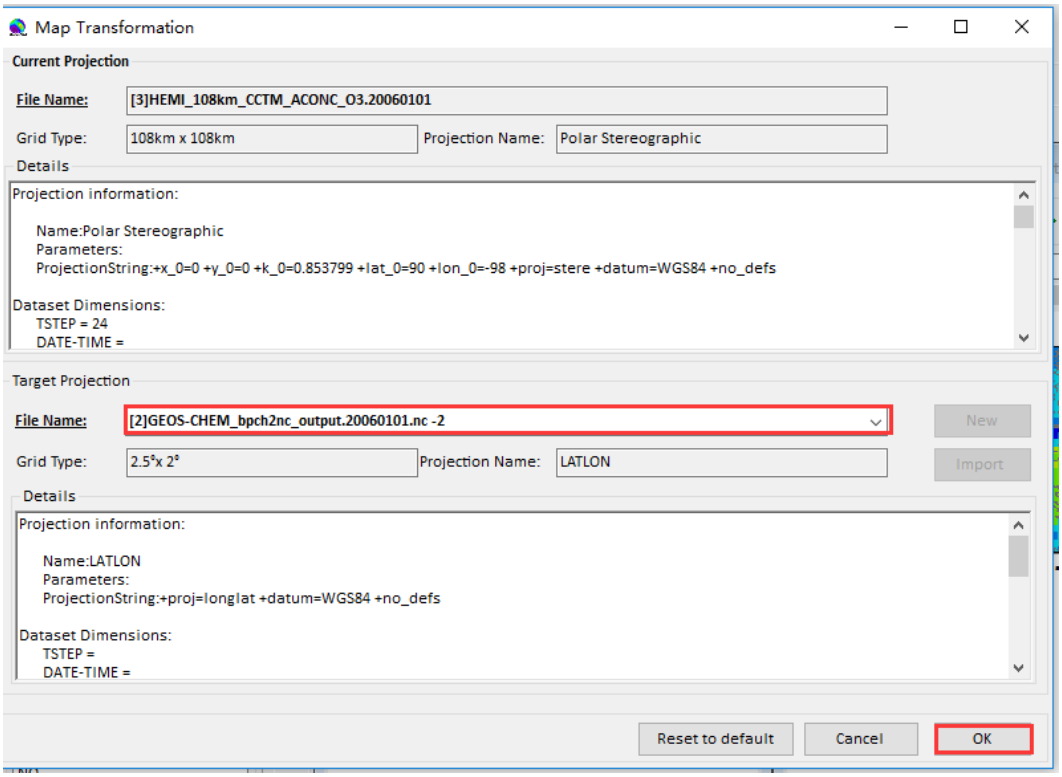


Figure 9.1-2. Choose Target Projection

Figure 9.1-3 and Figure 9.1-4 shows the plots before and after map transformation.

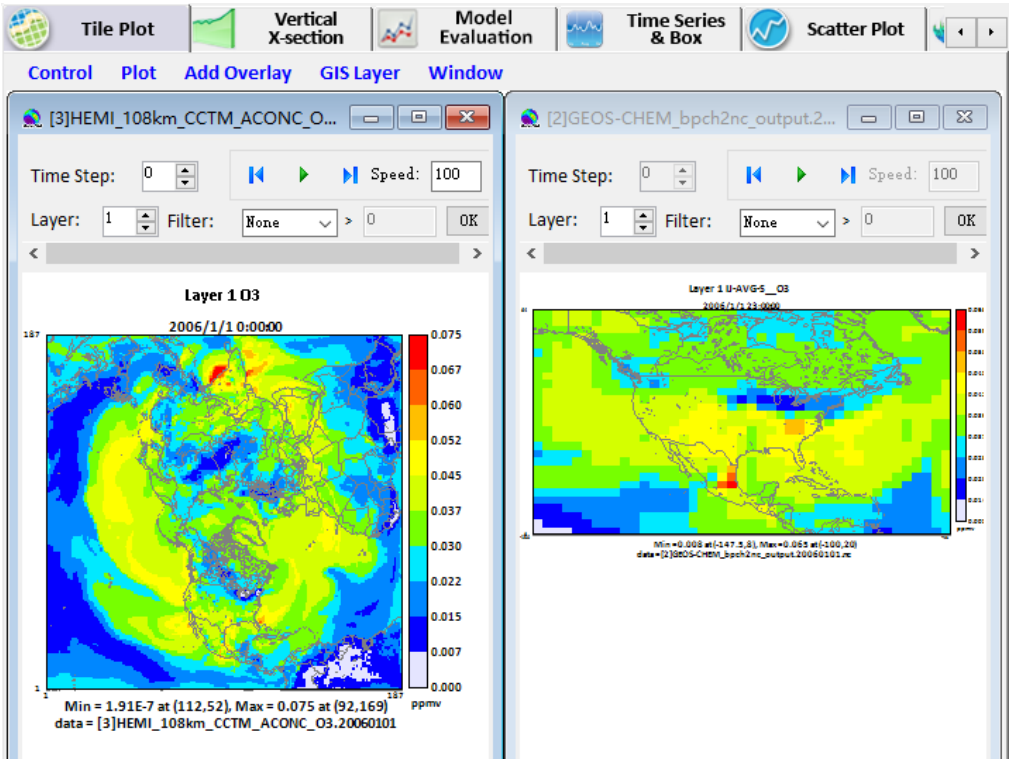


Figure 9.1-3. Plots before Map Transformation

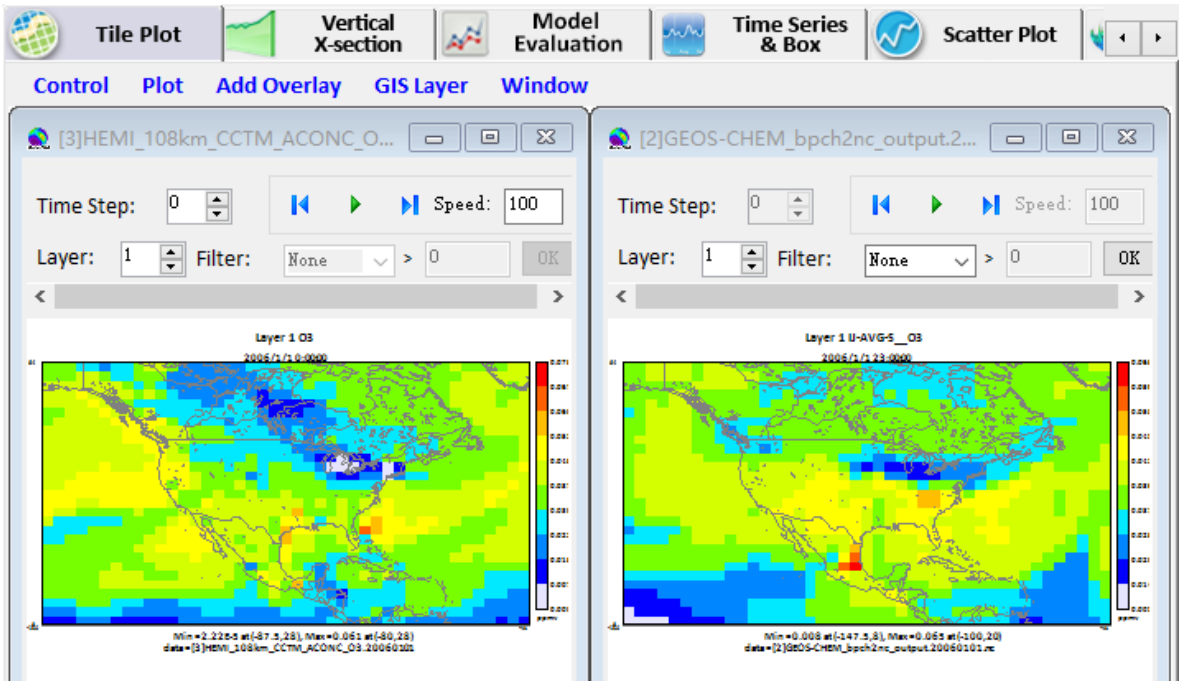


Figure 9.1-4. Plots after Map Transformation

9.2 Different Plot for Map Transformation

There are two ways to show different plot for Map Transformation:

1) First select target projection plot as active window and then click **Plot** to choose **Different Plot for Map Transformation**.

2) In target projection plot, right click to select **Different Plot for Map Transformation**.

Figure 9.2-1 shows the differences among different plot, target projection plot, and source projection plot.

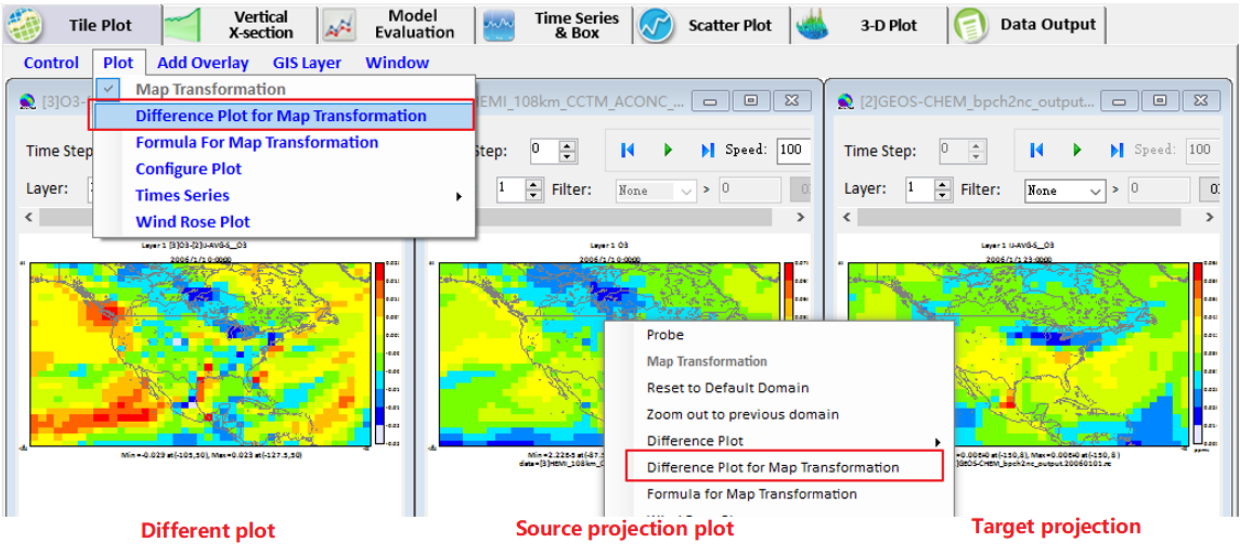


Figure 9.2-1. Different Plot, Source Projection Plot, and Target Projection Plot

9.3 Reset to Default

If you want to reset the plot to default projection and resolution, you can use **Reset to Default**.

You can reset to default domain and grid type in two ways:

1) Mouse right-click the transformed plot, and select **Reset to Default Domain**. This way only works for current window (Figure 9.3-1).

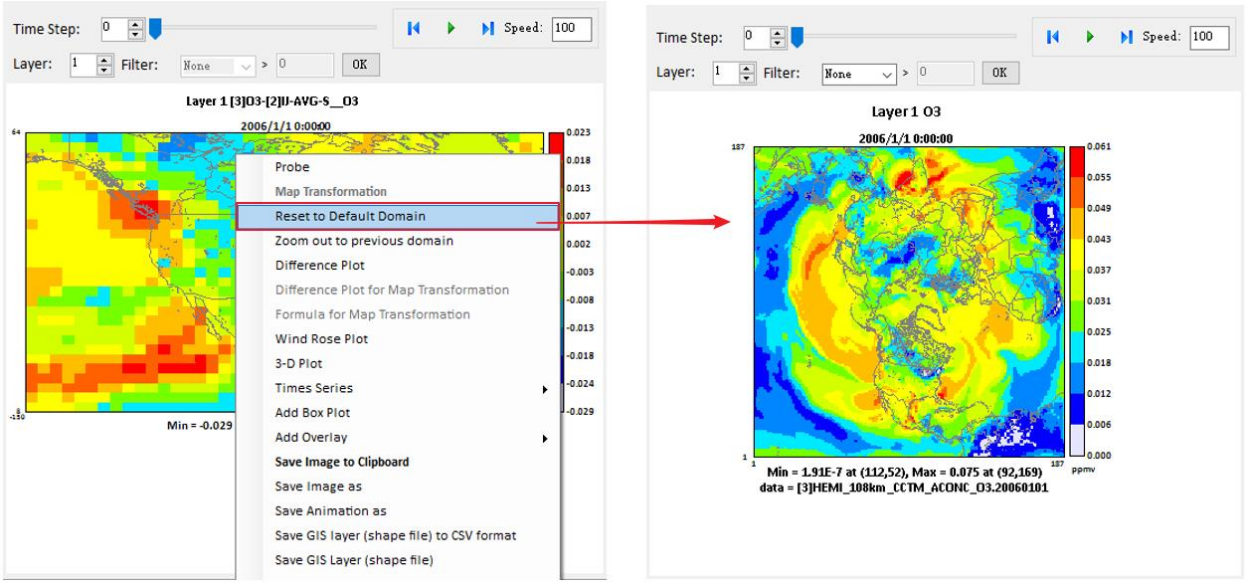


Figure 9.3-1. Reset to Default for Current Window

10 Mathematical Functions

All Model-VAT visualizations are the result of a formula evaluation. Formulas operate on the variables provided by the datasets. The simplest valid formula consists of a single variable; for example, {[1] O3} is the parameter O3 from current dataset 1. Using infix notation, you can construct more complicated formulas using the mathematical operators and functions listed below.

Listed in order of precedence, the functions and operators are:

1. abs, sqrt, exp, log, log10, ln, sin, cos, tan, asin, acos, atan, sinh, cosh, tanh, avg, sum, min, max
2. 10^ (power)
3. **/, ***
4. **+, -**
5. **<, <=, >, >=**
6. ==, !=
7. &&
8. ||

Model-VAT also supports the following constants:

1. E 2.7182818284590452354
2. PI 3.14159265358979323846

10.1 Unary Functions

Unary functions are passed a single argument. Depending on the argument and the function type, the function returns a single value or a matrix of data by performing the function on each cell of the arguments array. For example:

- **sqrt ({O3 [1]}):** Returns a matrix containing the square root of each value in the O3 [1] variable s array.

The following functions return a matrix when passed a dataset variable:

- **abs:** Returns the absolute value of the argument
- **sqrt:** Returns the square root of the argument.
- **log:** Returns the base any logarithm of the argument.
- **log10**: Returns the base 10 logarithm of the argument.
- **exp:** Returns Euler s number raised the power of the argument.
- **ln:** Returns the natural logarithm of the argument.

- **sin:** Returns the sine of the argument. The argument is in **radians**.
- **cos:** Returns the cosine of the argument. The argument is in **radians**.
- **tan:** Returns the tangent of the argument. The argument is in **radians**.
- **asin**: Returns the arcsine of the argument.
- **acos**: Returns the arccosine of the argument.
- **atan**: Returns the arctangent of the argument.
- **sinh**: Returns the hyperbolic sine of the argument.
- **cosh**: Returns the hyperbolic cosine of the argument.
- **tanh**: Returns the hyperbolic tangent of the argument.

The following functions return a single number in all cases when passed a dataset variable:

- **avg:** Average cell value for all cells in currently selected domain.
- **sum:** Sum of all cell values in currently selected domain.
- **min:** For each cell (*i*,*j*,*k*) in the currently selected domain, this calculates the minimum value for that cell over the currently selected time steps. In other words, the minimum value in cells (*i*,*j*,*k*,tmin tmax).
- **max**: For each cell (*i*,*j*,*k*) in the currently selected domain, this calculates the maximum value for that cell over the currently selected time steps. In other words, the maximum value in cells (*i*,*j*,*k*,tmin tmax).

10.2 Binary Operators

Binary operators are not passed a value but operate on the operands to their left and right. Typically, they return a matrix of data by performing the operation on each cell of the operand s arrays. If both of the operands are single numbers then these binary operators return a single number. For example:

- **{[1]O3} * 2:** multiplies each item in the {[1] O3} array by 2 and returns the result.
- **{[1]O3} * {[3] O3}:** multiplies each item in the {[1] O3} array by the corresponding item in the {[3] O3} array and returns the result. (Note that this assumes that the arrays are of equivalent shape.)
- **3 * 2:** multiplies 3 by 2.

The binary operators:

- **+** Returns the sum of the operands
- **-** Returns the difference of the operands
- ***** Returns the product of the operands
- **/** Returns the ratio of the operands

10.3 Boolean Operators

Boolean binary operators return either 1 or 0 in each cell of the resulting matrix. If the operands are single numbers, then a single 1 or 0 is returned. The Boolean binary operators:

- < Returns 1 if the left operand is less than the right operand, else 0
- <= Returns 1 if the left operand is less than or equal to the right operand, else 0
- > Returns 1 if the left operand is greater than the right operand, else 0
- >= Returns 1 if the left operand is greater than or equal to the right operand, else 0
- **!=** Returns 1 if the left operand is not equal to the right operand, else 0
- == Returns 1 if the left operand is equal to the right operand, else 0
- **&&** Returns 1 if both operands are nonzero, else 0
- || Returns 1 if either operand is nonzero, else 0