AERMOD View™
User Guide

Jesse L. Thé, Ph.D., P.Eng.
Cristiane L. Thé, M.A.Sc.
Michael A. Johnson, B.Sc.

Lakes Environmental™

AERMOD View™

Published by:

Lakes Environmental Software
170 Columbia St. W, Suite 1
Waterloo, Ontario
N2L 3L3
Canada
Tel.: +1.519.746.5995
Fax: +1.519.746.0793

www.webLakes.com
info@webLakes.com

AERMODVW950

License Agreement

LICENSE AGREEMENT

Please carefully read the following license and warranty information. By installing, archiving copies of, or otherwise using the licensed software, Licensee agrees to be bound by the terms and conditions of this license. Lakes Environmental Software, a division of Lakes Environmental Consultants Inc., retains the ownership of this copy of software. This copy is licensed to you for use under the following conditions:

COPYRIGHT NOTICE

This software is owned by Lakes Environmental Software and is protected by both Canadian copyright law and international treaty provisions. You must treat this software like any other copyrighted material (e.g., a book or musical recording). Lakes Environmental Software authorizes you to make archive copies of the software to protect it from loss. Licensee may not distribute, rent, sub-license, lease, alter, modify, or adapt the software or documentation, including, but not limited to, translating, decompiling, disassembling, or creating derivative works without the prior written consent of Lakes Environmental Software. Licensee agrees that in case of transference of ownership of the software, the transferee must expressly accept all terms and conditions of this agreement and written notice of transference must be provided to Lakes Environmental Software.
The associated warranty is not applicable to any freely available software that may be included as part of the software installation package. Lakes Environmental Software assumes no liability for the use of any freely available software applications.

WARRANTY AND LIABILITY

Lakes Environmental Software warrants that, under normal use, the software application and its' documentation will be free of defects in materials and workmanship for a period of 60 days from the date of purchase. In the event of notification of defects in material or workmanship, Lakes Environmental Software will replace the defective material or documentation.
The above warranty is in lieu of all other warranties, whether written, express, or implied. Lakes Environmental Software specifically excludes all implied warranties, including, but not limited to, loss of profit, and fitness for a particular purpose. In no case shall Lakes Environmental Software assume any liabilities with respect to the use, or misuse, or the interpretation, or misinterpretation, of any results obtained from this software, or for direct, indirect, special, incidental, or consequential damages resulting from the use of this software.
Specifically, Lakes Environmental Software is not responsible for any costs including, but not limited to, those incurred as a result of lost profits or revenue, loss of data, the costs of recovering programs or data, the cost of any substitute program, claims by third parties, or for other similar costs. In no event will Lakes Environmental Software's liability exceed the amount of the license fee.

SOFTWARE OPTIONS AND SERVICES ENABLED ONLY FOR LICENSES UNDER CURRENT MAINTENANCE

This software application contains features and services which may be restricted to use by users with a current maintenance agreement.
In the event that this software contains one or more of the options listed below, these software options will only be available for licenses that are currently in maintenance:

1. Import of Tile Maps
2. Download of NED Terrain Data in Geophysical Processor
3. Download of SRTM1 (Version 3) Terrain Data in Geophysical Processor
4. Download of CORINE Land Use Data in Land Use Creator
5. Download of EOSD Land Use Data in Land Use Creator

License Agreement (continued)

The following services will only be available for licenses that are currently in maintenance:

1. Access to the Online Knowledgebase
2. Software Updates
3. Technical Support (support@weblakes.com)

The high resolution imagery maps, under “Lakes Satellite”, are generated and provided by large number of copyright holders (the Licensors). Licensors include satellite companies, data aggregators, and mapping companies. Therefore, Lakes Environmental is obligated to pass these copyright owner’s use restrictions:

1. The map Content is owned and/or controlled by Lakes and its respective Licensors and is protected by local and international intellectual property laws. Users cannot resale, give away, or cache these maps outside the current application.

2. Licensors grant a non-exclusive, non-transferable license, revocable at any time at Licensors’ sole discretion, to access and use the maps strictly in accordance with the terms.

3. Users may not distribute, transfer the right to use, modify, translate, reproduce, resell, sublicense, rent, lease, reverse engineer, or otherwise attempt to discover the source code of or make derivative works of the data and maps from the application.

4. The Data is restricted for use in the specific system for which it was created. Except to the extent explicitly permitted by mandatory laws, you may not extract or reutilize any parts of the contents of the Data, nor reproduce, copy, modify, adapt, translate, disassemble, decompile or reverse engineer any portion of the Data.

5. Use the Content in accordance with the restrictions set out in the applicable local and international laws.

6. Software from Lakes Environmental may provide links to other sites. Lakes Environmental has no control over the third party content, sites, or services and assumes no responsibility for services provided or material created or published on these third-party sites or services. A link to a third-party site does not imply that Lakes Environmental endorses the site or the products or services referenced in the site.

7. The Data may contain inaccurate or incomplete information due to the passage of time, changing circumstances, sources used and the nature of collecting comprehensive geographic data, any of which may lead to incorrect results. The Data does not include or reflect information on - inter alia - travel time and may not include neighborhood safety; law enforcement; emergency assistance; construction work; road or lane closures; road slope or grade; bridge height, weight or other limits; road conditions; or special events depending on the navigation system brand that you possess. Users must assess suitability of the maps for the specific purpose of use.

8. To the extent permitted by local law, Lakes Environmental and/or Licensors, including their respective map and data suppliers, disclaim any warranties, express or implied, of quality, performance, merchantability, fitness for a particular purpose, or non-infringement.

9. Users may not use any automated systems or means, except for those provided by Lakes Environmental Software, for the selection or downloading of the Content.

License Agreement (continued)

10. You agree to use the Data in compliance with all applicable laws, rules and regulations, including local laws, rules and regulations of the country or region in which you reside or in which you obtain or use the Data. You agree not to export from anywhere any part of the Data or any direct product thereof, except in compliance with, and with all licenses and approvals required under, applicable export laws, rules and regulations, including but not limited to the laws, rules and regulations administered by the Office of Foreign Assets Control of the U.S. Department of Commerce and the Bureau of Industry and Security of the U.S. Department of Commerce. To the extent that any such export laws, rules or regulations prohibit your, your company, Lakes Environmental, and Licensors, or your supplier from complying with any of its obligations to deliver or distribute the Data, such failure shall be excused and shall not constitute a breach of this Agreement.

11. The Map Service is provided on "AS IS" and "AS AVAILABLE" basis. Lakes Environmental and Licensors do not warrant that the Map Service will be uninterrupted or error free. No warranty of any kind, either express or implied, including but not limited to warranties of title, non-infringement, merchantability, or fitness for a particular purpose, is made in relation to the availability, accuracy, reliability, information or content of the Service. You expressly agree and acknowledge that the use of the Map Service is at your sole risk and that you may be exposed to content from various sources.

WEB LICENSE

In the event this software was distributed with a web activated license, the software application must be able to periodically establish connectivity to the web license server, maintained by Lakes Environmental, in order for the software application to properly function. Specifically, the software application must successfully communicate with the web license server at a minimum of once every two weeks. Otherwise, software applications are not able to verify security and licensing credentials, which are necessary to maintain an active license status for the software application and registered user.

GOVERNING LAW

This license agreement shall be construed and enforced in accordance with the laws of the Province of Ontario, Canada. Any terms or conditions of this agreement found to be unenforceable, illegal, or contrary to public policy in any jurisdiction will be deleted, but will not affect the remaining terms and conditions of the agreement.

ENTIRE AGREEMENT

This agreement constitutes the entire agreement between you and Lakes Environmental Software.

Acknowledgements

The authors would like to thank all those who have helped with reviews and suggestions for the successful completion of AERMOD View.

Authors:

- Jesse L. Thé, Ph.D., P.Eng.
- Cristiane L. Thé, M.A.Sc.
- Michael A. Johnson, B.Sc.

Our development effort was enormously facilitated by our team members:

- Valeriy Smotrikov
- Igor Raskin
- Oleg Shatalov
- Yurai Núñez-Rodríguez
- Michael Hammer
- Alexandra Torrens

Disclaimer

This document and accompanying software follow the AERMOD, ISCST3, and ISC-PRIME model to the best of our understanding. The user is responsible for checking the input data and the results for consistency.

Technical Support

Lakes Environmental is dedicated to providing full technical support to users under current maintenance. If you need any assistance please contact the Lakes Environmental technical support staff. Our technical support hours are from 9:00 a.m. to 5:00 p.m. EST, Monday through Friday. Please have your serial number and version number ready when sending us an email.

Lakes Environmental Software

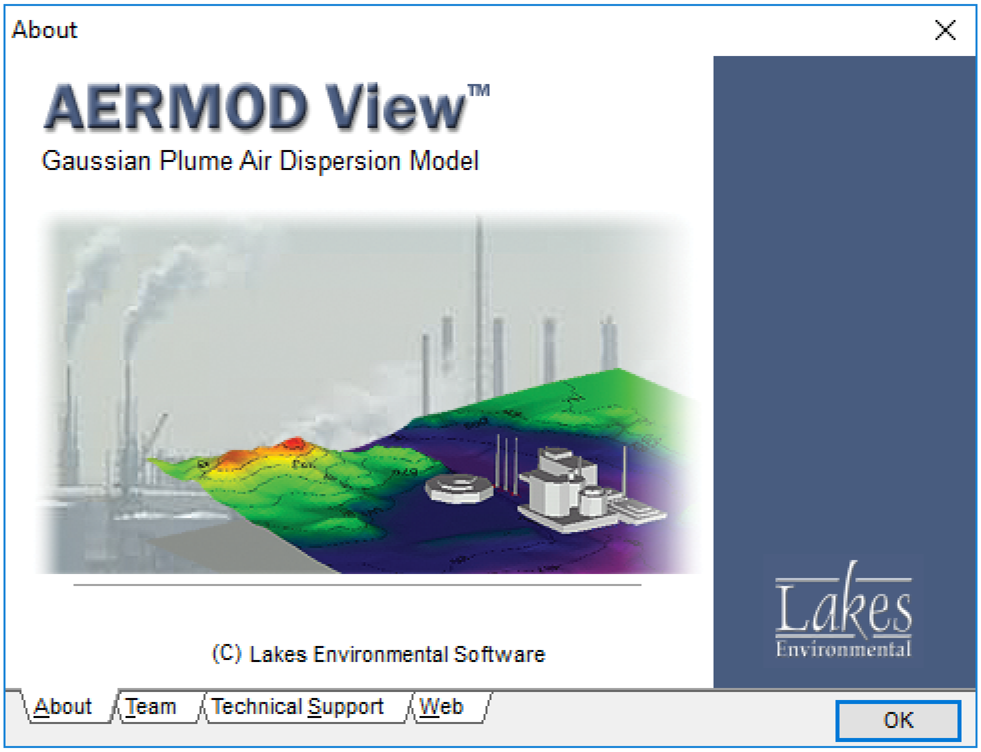
Tel.: +1.519.746.5995
Fax: +1.519.746.0793

e-mail: support@webLakes.com
Website: www.webLakes.com

Table of Contents

About AERMOD View

AERMOD View is a complete and powerful air dispersion modeling package that seamlessly incorporates the popular U.S. EPA models, AERMOD, ISCST3, and ISC-PRIME into one interface without any modifications to the models. These models are used extensively to assess pollution concentration and deposition from a wide variety of sources.



Features

- Create impressive presentations of your model results with the easy and intuitive graphical interface of AERMOD View. You can customize your project using display options such as transparent contour shading, annotation tools, various font options, and specify compass directions.

- Specify model objects such as sources, receptors and buildings graphically. After defining an object graphically you automatically have access to the related text mode list, in which you can further modify parameters.

- Automatically eliminate receptors within the facility property line.

- Import base maps in a variety of formats for easy visualization and source identification.

- Use the major digital elevation terrain formats - USGS DEM, NED, GTOPO30 DEM, UK DTM, UK NTF, XYZ Files, CDED 1-degree, AutoCAD DXF.

- Interpret the effects of topography by displaying your model results with 3D terrain using the powerful 3D visualization built right into the interface .

- Complete your building downwash analysis effectively and quickly using the necessary tools that AERMOD View provides.

- Prepare your meteorological data quickly and accurately using the step-by-step meteorological pre-processing interface.

- Take advantage of AERMOD View's integrated post-processing with automatic contouring of results, automatic gridding, blanking, shaded contour plotting and posting of your results.

- Compare models rapidly.

- Summarize your modeling input in professionally designed reports using report-ready formats.

- Use AERMOD View to its full potential and your best advantage by accessing context-sensitive “Help that really helps”, which provides you with a clear explanation of the modeling requirements.

About AERMOD MPI

The current US EPA AERMOD model executable is not capable of using multiple processors, now the norm with new computers, which come with at least a dual-core CPU (Central Processing Unit). However, the US EPA AERMOD source code is structured in such a way that it can be modified to take advantage of computers with multiple processors.

Lakes Environmental has adjusted the US EPA AERMOD source code and recompiled the model to parallelize the processing of receptors. The modifications to the source code have been kept as minimal as possible, with the intention of achieving identical results to those of the original US EPA AERMOD executable.

Processing speed gains indicate that as the number of processors in the computer increases:

- The reduction in processing time reduces approximate linearly with the number of processors.
    - For a 4-processor computer the gain in processing speed is approximately 3.75 faster.
    - For an 8-processor the total processing time is reduced by almost 8 times.

How to Select and Run AERMOD MPI (AERMOD Parallel)

1. Make sure you have installed MPICH2 that can be downloaded from the AERMOD View Update site.

MPICH2 installation is available in the AERMOD View Update Site for users in current maintenance.

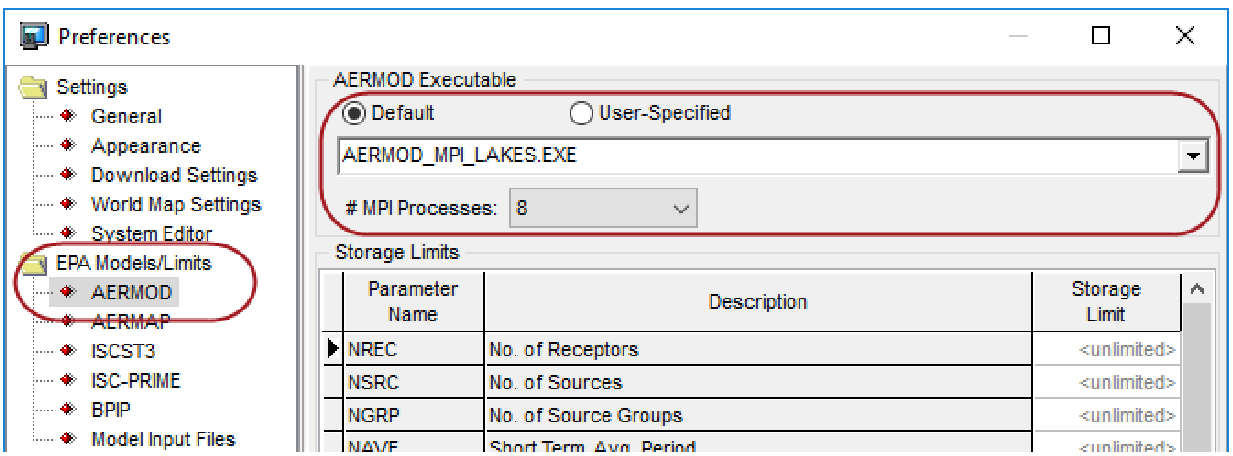
2. Open the Preferences dialog by clicking **File | Preferences** from the menu.

3. In the **Preferences** dialog, select **EPA Models/Limits | AERMOD**. The Preferences - EPA Models/ Limits - AERMOD dialog displays.

4. Make sure the **Default** option is selected, and select **AERMOD_MPI_LAKES.EXE** from the drop-down list.

5. From the **# MPI Processes** drop-down list box, select the number of **MPI processes (cores)** you want to use. The number of cores in your computer will be automatically detected in the list up to maximum of 8).

6. Click the **OK** button to save the changes. AERMOD View will automatically recognize this as a parallel version of the AERMOD executable and use it accordingly.



About Risk Generator

The AERMOD/ISC Risk Generator (**RiskGen**) allows you to set up all the required input files needed for your IRAP-h View project following the requirements of the 1998 U.S. EPA - OSW Human Health Risk Assessment Protocol (HHRAP). RiskGen runs on Microsoft® Windows® platform.

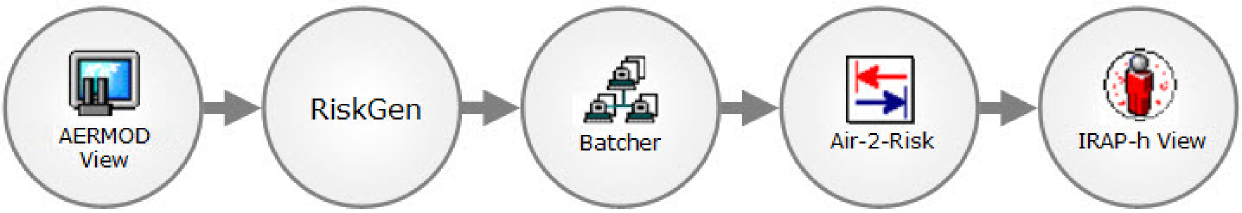
To use the **RiskGen**, you need to set up your ISCST3/AERMOD Risk Project using **AERMOD View**. Your **AERMOD View Risk Project** should contain the following information:

- Source parameters
- Building downwash information for all Point sources (if applicable)

- Particle Information Scavenging Coefficients for all sources (vapor phase, particle phase, and particle-bound phases)

- The Risk Grid

The following flow chart shows you the steps required to set up a complete Risk Assessment project:



- **AERMOD View**: Creates the Risk Project containing all the information necessary to model your site such as: sources, receptors, source emission data, etc.

- **RiskGen**: Reads your **Risk Input File** and creates all the necessary Input Files for your risk assessment project.

- **Batcher**: Takes the **Risk Input Files** generated by the **RiskGen** module and runs them using the appropriate model. Since most risk assessment projects require numerous Risk Input Files, the AERMOD Batcher can be distributed in various computers.

- **ISC 2 IRAP**: Takes the **Plotfiles** generated as a result of the runs and converts them into a format that can be used into your IRAP-h View project.

- **IRAP-*h* View**: Takes the risk data generated under the ISC 2 IRAP module, and allows you to prepare a human health risk assessment analysis following the 1998 U.S. EPA - OSW Human Health Risk Assessment Protocol (HHRAP).

The AERMOD Model

The **A**MS/**E**PA **R**egulatory **Mod**el (AERMOD) was specially designed to support the EPA's regulatory modeling programs. AERMOD is a regulatory steady-state plume modeling system with three separate components: AERMOD (AERMIC Dispersion Model), AERMAP (AERMOD Terrain Preprocessor), and AERMET (AERMOD Meteorological Preprocessor). The AERMOD model includes a wide range of options for modeling air quality impacts of pollution sources, making it a popular choice among the modeling community for a variety of applications. AERMOD contains basically the same options as the ISCST3 model:

- AERMOD requires two types of meteorological data files, a file containing surface scalar parameters and a file containing vertical profiles. These two files are provided by the U.S. EPA AERMET meteorological preprocessor program.

- PRIME building downwash algorithms based on the ISC-PRIME model have been added to the AERMOD model;
- Use of allocatable arrays for data storage;
- Incorporation of EVENT processing for analyzing short-term source culpability;
- Explicit treatment of multiple-year meteorological data files and the ANNUAL average;
- Options to specify emissions that vary by season, hour-of-day and day-of- week.
- For applications involving elevated terrain, the user must also input a hill height scale along with the receptor elevation. The U.S. EPA AERMAP terrain preprocessing program can be used to generate hill height scales as well as terrain elevations for all receptor locations.
- Deposition algorithms have been implemented in the AERMOD model - results can be output for concentration, total deposition flux, dry deposition flux, and/or wet deposition flux.
- The model contains algorithms for modeling the effects of settling and removal (through dry deposition) of large particulates and for modeling the effects of precipitation scavenging for gases or particulates.
- Two types of files of intermediate results for debugging purposes can be requested. One containing information related to the model results and the other containing gridded profiles of meteorological variables.
- AERMOD does not make any distinction between elevated terrain below release height (simple terrain) and terrain above release height (complex terrain).
- The Polar Plant Boundary receptor type is not available in AERMOD. A new type of receptor was included, the discrete Cartesian receptors that allows for grouping of receptors, e.g., along arcs. This receptor option was designed to be used with the EVALFILE option which is described below.
- Two additional output file options were included in AERMOD. One type of file lists concentrations by rank (RANKFILE). The other type of output file (EVALFILE) provides arc maxima results along with detailed information about the plume characteristics associated with the arc maximum.

The ISCST3 Model

The ISCST3 (**I**ndustrial **S**ource **C**omplex - **S**hort **T**erm version **3**) dispersion model is a steady-state Gaussian plume model which can be used to assess pollutant concentrations and/or deposition fluxes from a wide variety of sources associated with an industrial source complex. The ISCST3 dispersion model from the U. S. Environmental Protection Agency, was designed to support the EPA's regulatory modeling options, as specified in the Guidelines on Air Quality Models (Revised). Some of the ISCST3 modeling capabilities are:

- ISCST3 model may be used to model primary pollutants and continuous releases of toxic and hazardous waste pollutants.
- ISCST3 model can handle multiple sources, including point, volume, area, and open pit source types. Line sources may also be modeled as a string of volume sources or as elongated area sources.
- Source emission rates can be treated as constant or may be varied by month, season, hour-of-day, or other optional periods of variation. These variable emission rate factors may be specified for a single source or for a group of sources.
- The model can account for the effects of aerodynamic downwash due to nearby buildings on point source emissions.
- The model contains algorithms for modeling the effects of settling and removal (through dry deposition) of large particulates and for modeling the effects of precipitation scavenging for gases or particulates.
- Receptor locations can be specified as gridded and/or discrete receptors in a Cartesian or polar coordinate system.
- ISCST3 incorporates the COMPLEX1 screening model dispersion algorithms for receptors in complex terrain.
- ISCST3 model uses real-time meteorological data to account for the atmospheric conditions that affect the distribution of air pollution impacts on the modeling area.
- Results can be output for concentration, total deposition flux, dry deposition flux, and/or wet deposition flux.

The ISC-PRIME Model

The **P**lume **Ri**se **M**odel **E**nhancements (PRIME) model was designed to incorporate the two fundamental features associated with building downwash:

- Enhanced plume dispersion coefficients due to the turbulent wake

- Reduced plume rise caused by a combination of the descending streamlines in the lee of the building and the increased entrainment in the wake.

The PRIME algorithms have been integrated into the ISCST model and called the ISC-PRIME model which contains the same basic options as the ISCST3 model but with enhanced building downwash analysis. ISC-PRIME uses the standard ISCST3 input file with some modifications. These modifications allow the specification of three new inputs used to describe the building/stack configuration. These new inputs are as follows:

- **BUILDLEN**: Projected length of the building along the flow

- **XBADJ**: Along-flow distance from the stack to the center of the upwind face of the projected building.

- **YBADJ**: Across-flow distance from the stack to the center of the upwind face of the projected building.

To be able to run the ISC-PRIME model, you must first run the BPIP-PRIME model. The BPIP-PRIME building downwash output results are then used by the ISC-PRIME model.

All the remaining options for the ISC-PRIME model are the same as for the ISCST3 model. However, some enhancements of the ISCST3 model are currently not being supported in ISC-PRIME such as:

- Post-97 PM10 Processing

- Memory Allocation

- INCLUDED Option

- TOXICS Option

- Sampled Chronological Input Model (SCIM) Option

- Optimized Area Source and Dry Depletion Algorithms

- Gas Dry Deposition Algorithm

- Season by Hour-of-Day Output Option (SEASONHR)

See a detailed description on these options on the ADDENDUM for the User's Guide for the Industrial Source Complex (ISC3) Dispersion Models - Volume 1 – User Instructions (EPA, 2002).

The AERMAP Model

The U.S. EPA AERMOD model was designed to handle all types of terrain, from flat to complex. To model complex terrain, AERMOD requires additional information about the surrounding terrain. This information includes a height scale and a base elevation for each receptor. To obtain a height scale and the base elevation for a receptor, you need to run the U.S. EPA terrain preprocessor AERMAP (**A**MS/**E**PA **R**egulatory **M**odel Terrain **P**re-processor).

> AERMAP requires the DEM data for the area you are modeling to be in either the 7.5-Minute DEM or 1-Degree DEM format. However, the Terrain Processor Wizard allows you to specify any of the above files types and will perform a background conversion of the terrain file into the 7.5-Minute DEM format.

When AERMAP finishes running, a message is displayed asking if you want to check the Summary file (*.ast). The Summary file contains the input file used to run AERMAP followed by fatal error messages, warning messages, and/or informational messages. It is advisable to always check the Summary file for possible errors. AERMAP produces two main output files:

- **Receptor Output File (*.rou):** This file contains the calculated terrain elevations and scale height for each receptor. You have the option of using this file as an INCLUDED file in the input runstream file (AERMOD Input File) for the Receptor Pathway or not in the Terrain Options screen of the Control Pathway. If the ROU File is not used as an INCLUDED parameter, then you will be able to modify the terrain elevations and hill heights within the interface and these values will be copied to the input file for the run.

- **Source Output File (*.sou):** This file contains the calculated base elevations for all your sources.

Additional files are created by AERMAP during the run. These include:

- DOMDETAIL.OUT
- MAPDETAIL.OUT
- MAPPARAMS.OUT
- RECDETAIL.OUT
- RECELV.OUT
- RECNDEM.OUT
- SRCDETAIL.OUT
- SRCNDEM.OUT

AERMOD View also allows you to select the AERMAP API version of the executable, which lets you take advantage of the processing power of multiple cores.

How to Select and Run AERMAP MPI (AERMAP Parallel)

1. Make sure you have installed MPICH2 that can be downloaded from the AERMOD View Update site.

> MPICH2 installation is available in the AERMOD View Update Site for users in current maintenance.

2. Open the Preferences dialog by clicking **File | Preferences** from the menu.

3. In the **Preferences** dialog, select **EPA Models/Limits | AERMAP**. The Preferences - EPA Models/ Limits - AERMAP dialog displays.

4. Make sure the **Default** option is selected, and select **AERMAP_MPI_LAKES.EXE** from the drop-down list.

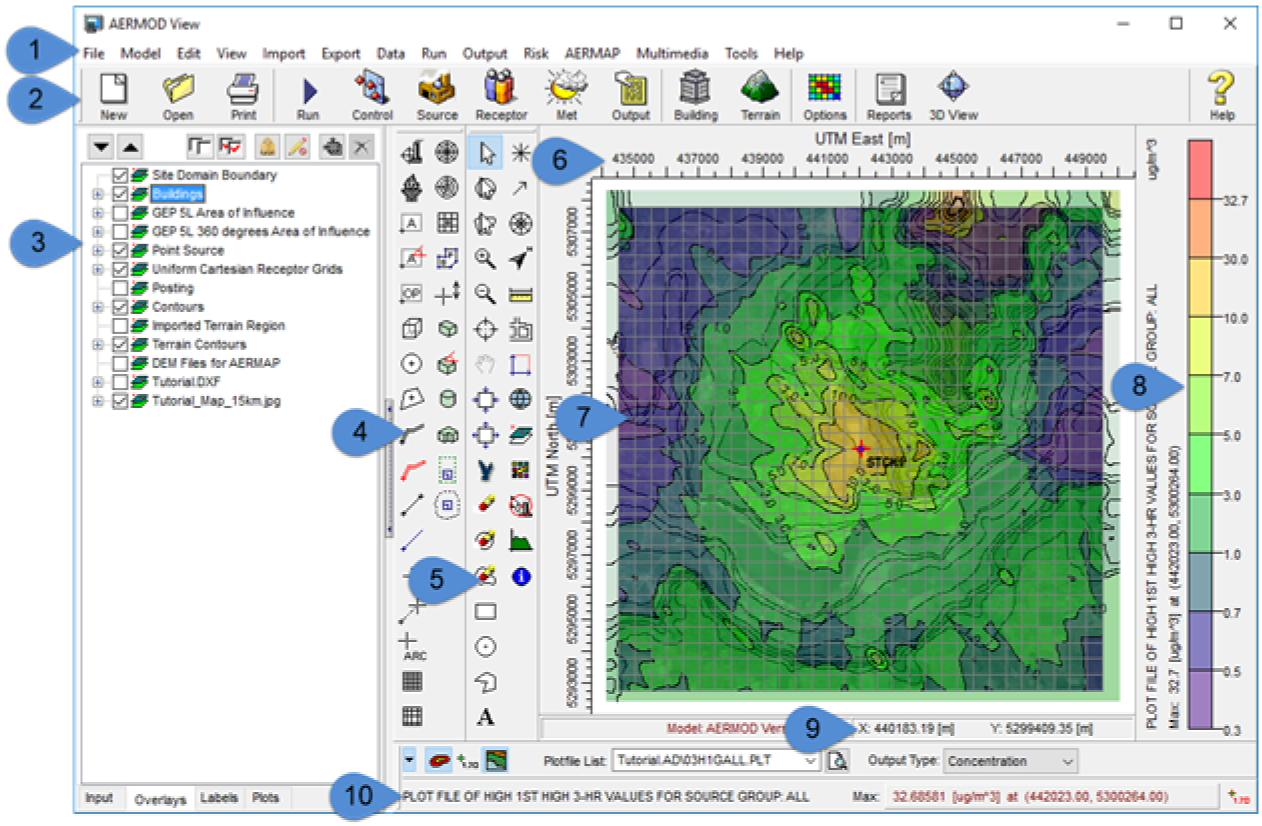
5. From the **# MPI Processes** drop-down list box, select the number of **MPI processes (cores)** you want to use. The number of cores in your computer will be automatically detected in the list up to maximum of 8).

6. Click the **OK** button to save the changes. AERMOD View will automatically recognize this as a parallel version of the AERMAP executable and use it accordingly.

AERMOD View Interface

AERMOD View features a friendly, intuitive interface which provides easy access to all modeling tools and simultaneous visualization of your project.

The components of the AERMOD View interface are briefly described below:



1. **Menu Bar**: Displays menu names. To open a menu, move the mouse over the menu name and then click the left mouse button. A drop-down menu appears displaying a list of related commands.

2. **Toolbar Buttons**: These are a series of buttons that provide a fast method of selecting some of the menu commands.

3. **Tree View**: This display option can be turned on or off at any time. The Tree View has four tabs.

   - Input tab: displays the AERMOD input options specified for the current project.

   - Labels tab: allows you to turn the layer labels on or off.

   - Overlays tab: enables you to specify which layers are visible in the drawing area.

- Plots tab: displays the available output plot files for the current project run. This tab is displayed only after you have successfully run your project.

- Plume Animation tab: displays the plume animation for the current project. This tab is displayed only after you have enabled the plume animation feature in Output Pathway.

4. **Application Toolbar**: The tools available in the Application Toolbar allow you to graphically define the location of your sources, buildings, grids, and other objects. The Application Toolbar can be docked or floating.

5. **Annotation Toolbar**: The tools available in the Annotation Toolbar allow you to manage the contents of the drawing area and enhance the presentation of your modeling project. With these tools you can select objects, delete, zoom in and out of a section of the drawing area, control the display of overlays and import base maps. The Annotation Toolbar can be docked or floating.

6. **Drawing Area**: This is the large white area of the main interface in which your project objects such as sources, buildings and grids are displayed graphically. Here, you can visualize maps, graphically specify objects and view model results.

7. **Axis Labels**: X and Y-axis labels are placed on the top and left side of the drawing area. These labels display the real coordinate values for the modeling area. The Axis can be modified in the Ruler Options page.

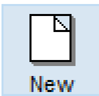
8. **Color Ramp**: The Color Ramp displays the colors you selected for your contours from the Graphical Options dialog. You have the option of displaying or hiding the color ramp and choosing its location in the main window

9. **Coordinates Panel**: This area displays the X and Y coordinates of the location where the mouse cursor is pointing.

10. **Graphical Output Toolbar**: The tools available on the Graphical Output Toolbar allow you to easily turn on and off contouring, posting of results, and select the plotfile to be displayed in the drawing area.

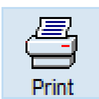
Menu Toolbar Buttons

The **Menu Toolbar Buttons** are shortcuts to some of the menu commands. The function of each one of these buttons is explained below and the equivalent menu bar command is indicated.



New: Lets you create a new AERMOD View project.

Open: Lets you open an existing AERMOD View project.



Print: Displays the Print Preview dialog, where you can preview what will be sent to the printer.

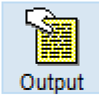
Run: Displays the floating menu in which you can select whether you wish to run an AERMOD, BPIP, or an EVENT model. You may also select to load Batcher to set up multiple runs or use the Multi-Chemical Run utility.

Control: Displays the Control Options tab located in the Options dialog. In this tab, you can specify general control options and parameters for your AERMOD View project.

Source: Displays the Source Pathway dialog where you can specify the sources, and their parameters, for your project.

Receptor: Displays the Receptor Pathway dialog where you can specify the location of grids and receptors.

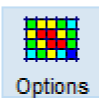
Met: Displays the Meteorology Pathway where you can select the meteorological data to be used in your project.



Output Pathway: Displays the Output Pathway dialog.

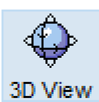
Building: Displays the Building Inputs dialog which allows you to define the properties of the buildings in your project.

Terrain: Displays the Terrain Processor dialog where you can specify terrain options for your project.

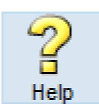


Options: Displays the Graphical Options dialog where you can customize the appearance of your project.

Reports: Displays the Reports dialog from where you can view and print reports of all the input data for the current project.



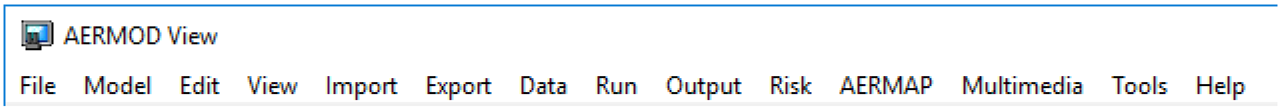
3D View: Click this button to either display 3D View, where you can have a complete three dimensional visualization of your project or open and display your project in Google Earth™.



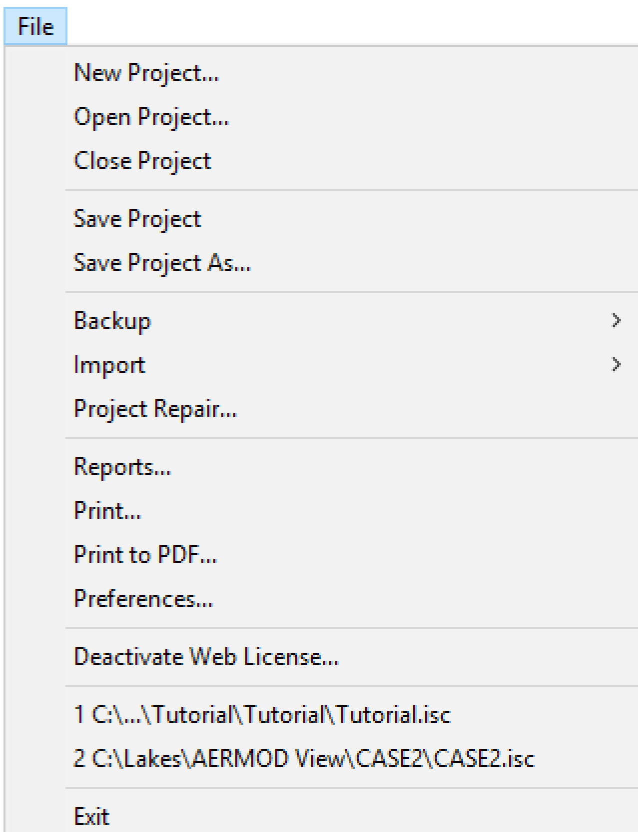
Help: Displays the AERMOD View Help Contents, from which you can select specific topics.

Menu Options

The following menu options are available in AERMOD View:



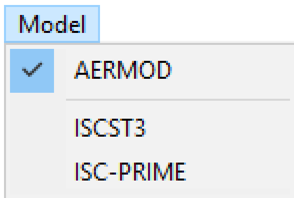
File



- **New Project...:** Displays the New Project Wizard, which will guide you through the process of creating a new project.
- **Open Project...:** Displays the **Open Project** dialog, where you can specify an existing project file (*.isc) to open.
- **Close Project:** Closes the currently open project currently.

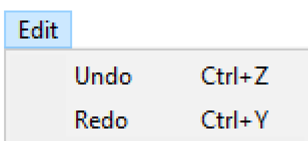
- **Save Project:** Saves the current project file (*.isc) to your AERMOD View project file.
- **Save Project As...:** Displays the Save Project As dialog, allowing you to save the current project with a different name and/or location.
- **Backup:** Displays the following submenu options:
    - **Save to ZIP:** Allows you to backup your entire project adding all files from your project to an archive (ZIP file). The Backup Options dialog is displayed allowing you to add any extra files you need to the zip file.
    - **Extract from ZIP:** Extracts your project files from the archive file (ZIP file).
- **Import:** Displays the following submenu options:
    - **BPIP Input File...:** Imports a BPIP Input File.
    - **AERMOD Input File...:** Imports an AERMOD Input File.
    - **ISCST3 Input File...:** Imports an ISCST3 Input File.
    - **ISC-PRIME Input File...:** Imports an ISC-PRIME Input File.
- **Repair Project...:** Displays the Project Repair dialog allowing you to repair database files for a project that may be corrupted.
- **Reports...:** Displays the Reports dialog from where you can preview and print reports of all the input options for the current project.
- **Print...:** Displays the Print Preview dialog where you can preview and print the contents of the drawing area.
- **Print to PDF...:** Displays the Print to PDF File dialog.
- **Preferences...:** Displays the Preferences dialog, where you can specify preferences, such as company name, modeler & model default settings.
- **Deactivate Web License...:** Deactivates web license on current machine, so it can be activated on a different one.
- **List of Files:** Displays a list of up to four of the most recently used project files
- **Exit:** Closes the program.

Model



- **AERMOD:** Select this menu option if you want to change to the AERMOD model. A check indicates which model is being used.
- **ISCST3:** Select this menu option if you want to change to the ISCST3 model. A check indicates which model is being used.
- **ISC-PRIME:** Select this menu option if you want to change to the ISC-PRIME model. A check indicates which model is being used.

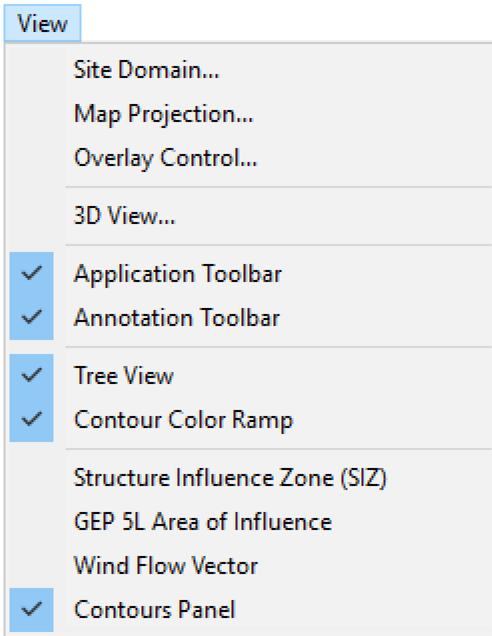
Edit



- **Undo:** Undo a change to an object - Move, Resize, or Rotate.
- **Redo:** Return the annotation to the state before the Undo function was performed.

> The **Undo** and **Redo** features do not work on deleted items. If you accidentally delete an item, you will need to re-create it.

View

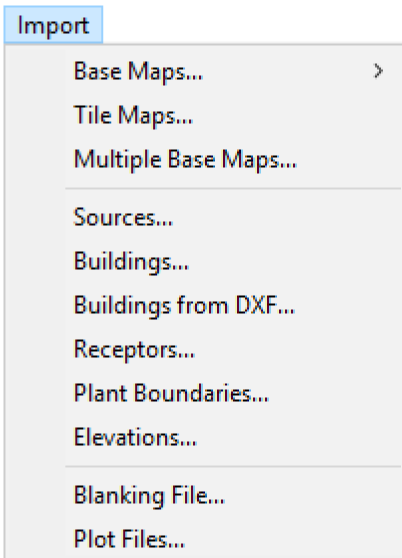


- **Site Domain...:** Displays the Site Domain dialog, where you can define the extents of your modeling area.
- **Map Projection...:** Displays the Map Projection dialog.
- **Overlay Control:** Displays the Overlay Control dialog, where you can control which base maps and objects are being currently displayed in the drawing area. It also controls the order in which maps and objects are displayed.
- **3D View...:** Launches 3D View where you can visualize your project in 3D.
- **Application Toolbar:** This turns on and off the Application Toolbar. A check indicates that the toolbar is currently active.
- **Annotation Toolbar:** Displays or hides the Annotation Toolbar. A check indicates that the toolbar is currently active.
- **Tree View...:** This turns on and off the display of the Tree View. A check indicates that the Tree View is currently displayed.
- **Contour Color Ramp:** This turns on and off the display of the Contour Color Ramp.
- **Structure Influence Zone (SIZ):** Show/Hides the Structure Influence Zone. A check indicates the Structure Influence Zone is currently displayed.
- **GEP 5L Area of Influence:** Show/Hides the GEP 5L Area of Influence. A check indicates the GEP 5L Area of Influence is currently displayed.

- **Wind Flow Vector:** Displays the Wind Flow Vector dialog. A check indicates the Wind Flow Vector dialog is currently displayed.

- **Contours Panel:** This turns on and off the display of the **Contours Panel**. A check indicates that the panel is currently displayed.

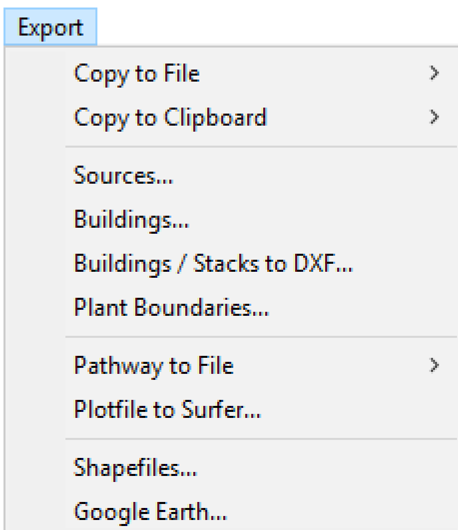
Import



- **Base Maps:** Displays the following submenu options -
  - **DLG...:** Displays the Importing DLG Base Map dialog.
  - **DXF...:** Displays the Importing DXF Base Map dialog.
  - **LULC...:** Displays the Importing LULC Base Map dialog.
  - **MrSID...:** Displays the Importing MrSID Base Map dialog.
  - **Raster Images...:** Displays Importing Raster Image dialog.
  - **Shapefile...:** Displays the Importing ArcView Shapefile dialog.
- **Tile Maps...:** Displays Import Tile Maps dialog.
- **Multiple Base Maps...:** Displays the dialog to Import Multiple Base Maps.
- **Sources...:** Allows you to Import Sources into your project.
- **Buildings...:** Allows you to import buildings from either an Excel spreadsheet with Lakes Format (*.xls, *xlsx) or a BPIP Input File (*.bpi, *.pip, *.inp, *.prm) into your project.

- **Buildings from DXF...:** Displays the Import Buildings from DXF dialog where you can import building locations and dimensions from an AutoCAD DXF file.
- **Receptors...:** Allows you to Import Receptors into your project.
- **Plant Boundaries...:** Allows you to import a plant boundary from a .CSV file or an Excel spreadsheet with Lakes Format into your project.
- **Elevations...:** Allows you to import terrain elevations.
- **Blanking File:** This option allows you to Import a Blanking File (*.rpb) that was created previously.
- **Plot Files...:** Allows you to import contour plot files into your project.

Export



- **Copy to File:** Displays the following sub-menu options -
  - **Bitmap.....:** Displays the Export As Bitmap dialog, allowing you to save the contents of the drawing area as a Windows bitmap.
  - **Metafile...:** Displays the Save Enhanced Metafile dialog, allowing you to save the contents of the drawing area as a Windows metafile.
- **Copy to Clipboard:** Displays the following sub-menu options
  - **Bitmap.....:** Copies the contents of the drawing area to the clipboard, allowing you to paste these contents into another windows application as a Windows bitmap.

- ○ **Metafile...:** Copies the contents of the drawing area to the clipboard, allowing you to paste these contents into another windows application as a Windows metafile.

- **Sources...:** Allows you to export sources as .XLS or .XLSX file.

- **Buildings...:** Allows you to export buildings as an Excel spreadsheet with Lakes Format (*.xls) or a BPIP Input File (*.bpi).

- **Buildings/Stacks to DXF...:** Opens the Export DXF Base Map dialog allowing you to export the buildings and stacks in the drawing area to an AutoCAD DXF.

- **Plant Boundaries...:** Allows you to export plant boundaries as .XLS file.

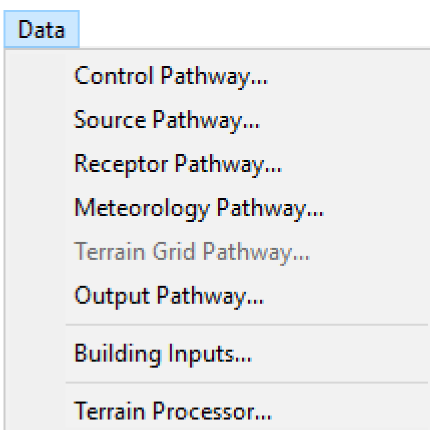
- **Pathway to File:** Allows you to save the inputs written for a specific pathway (e.g. Control, Source, Met, etc.) to a separate file.

- **Plotfile to Surfer...:** Allows you to save your plotfiles as a *.gdr file, which can be imported into Surfer.

- **Shapefiles...:** Displays the Export Layer to Shapefile dialog allowing you to select a layers to export as a shapefiles.

- **Google Earth...:** Allows you to display your project in Google Earth™.

Data



- **Control Pathway...:** Displays the Control Pathway dialog where you can specify the overall job control options.

- **Source Pathway...:** Displays the Source Pathway dialog where you can specify the source input parameters and source group information.

- **Receptor Pathway...:** Displays the Receptor Pathway dialog where you can specify receptor information for your project.

- **Meteorology Pathway...:** Displays the Meteorology Pathway dialog where you can specify the input meteorological data for your project.

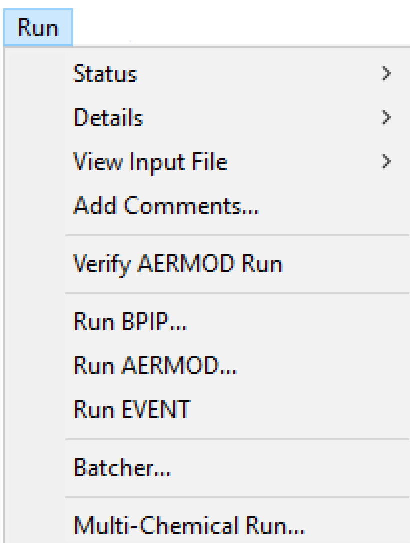
- **Terrain Grid Pathway...:** Displays the Terrain Grid Options screen where you may define the input terrain grid data used in calculating dry depletion in elevated or complex terrain. This option is available only for the ISCST3 and ISC-PRIME models.

- **Output Pathway...:** Displays the Output Pathway dialog where you can specify the output options for a particular run.

- **Building Inputs...:** Displays the Building Inputs dialog where you specify your building parameters.

- **Terrain Processor...:** Displays the Terrain Processor dialog where you can import and process digital terrain elevation data into your project.

Run



- **Status...:** Displays the following sub-menu options -
  - **BPIP...:** Displays the Project Status - BPIP dialog, which displays a summary of all the BPIP options selected for the current project.
  - **Model...:** Displays the Project Status dialog, which displays a summary of all the model options selected for the current project.
- **Details...:** Displays the following sub-menu options -
  - **BPIP...:** Displays the Details dialog which lists any missing data for the current project that is required to run the BPIP model.

  - **Model...:** Displays the Details dialog, which lists any missing data for the current project that is required to run the selected U.S. EPA model.

- **View Input File:** Displays the following sub-menu options -

  - **BPIP...:** Displays the BPIP Input File (*.bpi) for the current project in WordPad.

  - **Model...:** Displays the Input File for the current model using Windows WordPad.

  - **EVENT...:** Displays the EVENT Input File for the current model using Windows WordPad.

- **Add Comments:** Displays the Add Comments to Input File dialog allowing you to specify comments for each pathway section of the input file.

- **Verify Model Run:** This option allows you to process the U.S. EPA model to verify the completeness of your project, checking for any fatal errors or warning messages before running the model.

- **Run BPIP...:** Displays the Project Status - BPIP dialog from where you can run the BPIP model.

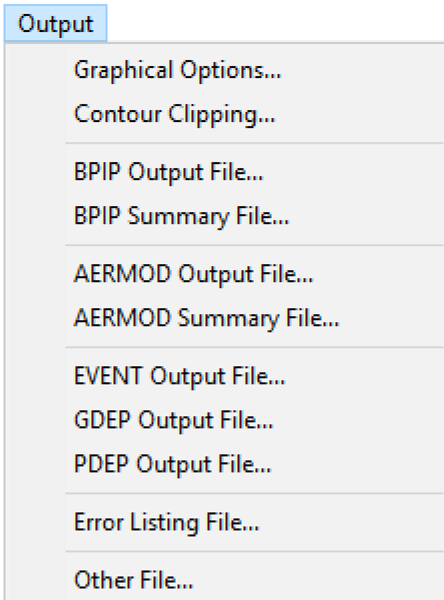
- **Run Model...:** Displays the Project Status dialog from where you can run the U.S. EPA model.

- **Run EVENT...:** Runs the EVENT model.

- **Batcher...:** Launches the AERMOD Batcher utility, which allows you to run any one of the models outside the AERMOD View interface. Ideal for running multiple projects in an unattended mode.

- **Multi-Chemical Run...:** Launches the Multi-Chemical Run utility which can be used to model sources emitting multiple pollutants.

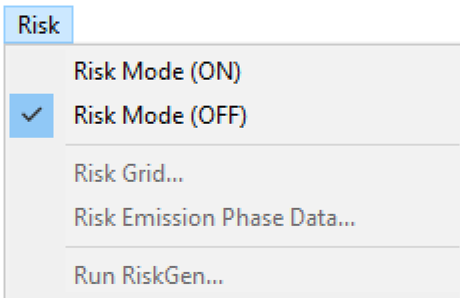
Output



- **Graphical Options...:** This opens the Graphical Options dialog where you can customize the appearance of your project.
- **Contour Clipping...:** Displays the Contour Clipping dialog which allows you the option of using contour clipping.
- **BPIP Output File...:** Displays the BPIP Primary Output File (*.pro) for the current project in WordPad.
- **BPIP Summary File...:** Displays the BPIP Summary File (*.sup) for the current project in WordPad.
- **Model Output File...:** Displays the Output File for the current model using Windows WordPad. The output file is created when you run your project.
- **Model Summary File...:** Displays the set up summary for the model and the message summary of the model execution.
- **EVENT Output File...:** Displays the EVENT Output File for the current model using Windows WordPad. This output file is created only if you have run the EVENT model.
- **GDEP Output File...:** Displays the GDEP.DAT File produced during an AERMOD run. This output file is created only when you run AERMOD.
- **PDEP Output File...:** Displays the PDEP.DAT File produced during an AERMOD run. This output file is created only when you run AERMOD.
- **Error Listing File...:** Displays the Error Listing File for the current model using Windows WordPad. This file is created only if you have selected the Error Listing File option in the Control Pathway and have run the model.

- **Other File...:** Displays the Open Text File dialog from where you can specify any text file to view using Windows WordPad.

Risk



- **Risk Mode (ON):** Select this option to turn ON the Risk Mode. This allows you to specify options for preparing input files according to the U.S. EPA Human Health and Ecological Risk Assessment Protocols.

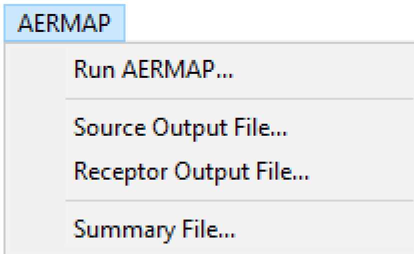
- **Risk Mode (OFF):** Select this option to turn OFF the Risk Mode.

- **Risk Grid...:** Select this option if you want to define a Risk Grid according to the U.S. EPA Human Health and Ecological Risk Assessment Protocols.

- **Risk Emission Phase Data...:** Displays the Risk Mode - Gas & Particle Data screen where you can specify source emission information for all the three phases (vapor, particle, and particle-bound phases) required under the U.S. EPA OSW Human Health and Screening Level Ecological Risk Assessment Protocols (HHRAP & SLERAP). The information specified in this dialog will not be placed in the input file created by AERMOD View. This information, however, is used by Lakes RiskGen, when creating all the necessary files for your risk assessment project.

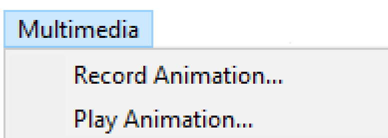
- **Run RiskGen...:** This option displays the RiskGen utility. RiskGen sets up all the required input files needed for your risk project following the requirements of the U.S. EPA - OSW Human Health Risk Assessment Protocol and Screening Level Ecological Risk Assessment Protocol (HHRAP & SLERAP).

AERMAP



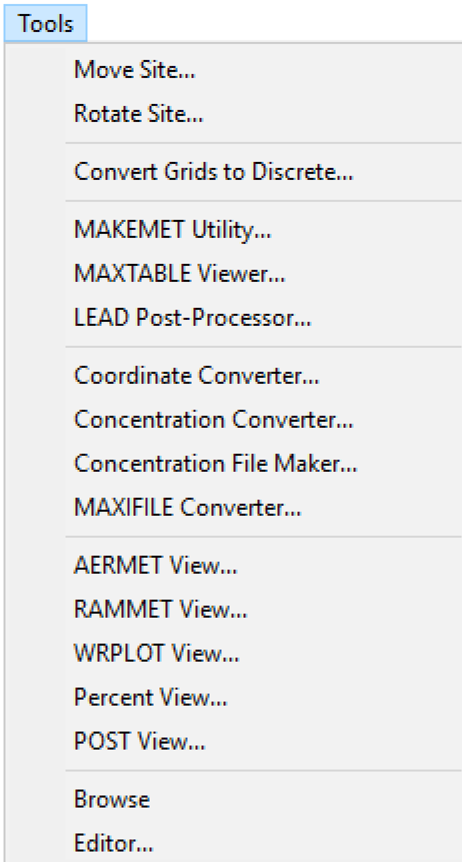
- **Run AERMAP...:** Displays the Terrain Processor, where you can define the DEM files to be used by AERMAP.
- **Source Output File...:** Displays the source output file (*.sou) generated by AERMAP that contains the source data, including source elevations.
- **Receptor Output File...:** Displays the receptor output file (*.rou) generated by AERMAP that contains the receptor data, including receptor elevations and height scales.
- **Summary File...:** Displays the output message file containing an echo of the input file and a listing of any warning or error messages generated by AERMAP.

Multimedia



- **Record Animation...:** Displays the Multimedia Options dialog from where you can record animations.
- **Play Animation...:** This option allows you to select the name and location of the animation file to play.

Tools



- **Move Site...:** This option displays the Move Site By... dialog which allows you to shift the coordinates of your sources, receptors, and buildings by a specified amount.

- **Rotate Site...:** This option displays the Rotate Site By... dialog which allows you to rotate your building from plant to true north coordinates. This tool allows you to import drawings from plant north.

- **Convert Grids to Discrete...:** Displays the Convert Receptor Grids to Discrete Receptors dialog where you can easily convert receptor grids into discrete receptors.

- **MAKEMET Utility...:** Launches the dialog that you can use to specify the parameters to run the MAKEMET utility.

- **MAXTABLE Viewer...:** Opens the MAXTABLE Viewer where you can view and export all the Maximum Value Tables within your output file.

- **LEAD Post-Processor...:** Opens the LEAD Post-Processor Utility (LEADPOST) where you can calculate rolling 3-month averages by receptor group and overall maximum concentration.

- **Coordinate Converter...:** Displays the Coordinate Converter utility where you can convert between geographic and projected cartesian coordinates.

- **Concentration Converter...:** Displays the Concentration Converter utility, which allows you to modify existing plotfiles to generate new plotfiles without having to re-run the EPA model.

- **Concentration File Maker...:** Displays the Concentration File Maker utility which allows you to create your own ozone files (*.dat).

- **MAXIFILE Converter...:** Opens the MAXIFILE Converter where you can convert a MAXIFILE into a plotfile.

- **AERMET View...:** Launches AERMET View which preprocesses your met data for use with the AERMOD model.

- **RAMMET View...:** Launches RAMMET View which preprocesses your met data for use with the ISCST3 and ISC-PRIME models.

- **WRPLOT View...:** Launches WRPLOT View which generates wind rose statistics and plots of your meteorological data.

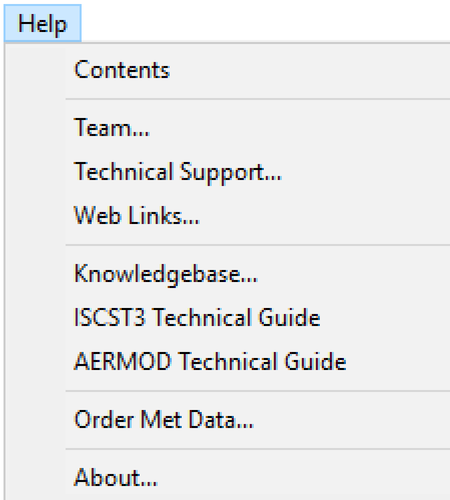
- **Percent View...:** Launches Percent View which generates percentile plots of a given averaging period contained within a Post-Processing File (POSTFILE), and allows you to perform rolling averages.

- **POST View...:** Launches POST View where you can view and customize multiple contour plots.

- **Browse...:** Opens a Microsoft® Windows® Explorer dialog allowing you to browse through files.

- **Editor...:** Opens Microsoft WordPad for easy access to a text editor.

Help

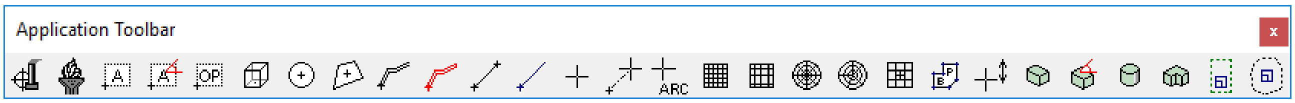


- **Contents:** Displays Help Contents, from which you can select topics.
- **Team...:** Displays information on the program development team.
- **Technical Support...:** Displays a dialog containing available Technical Support options for the program.
- **Web Links...:** Displays a dialog containing links to sites useful to the user.
- **Knowledgebase...:** Displays the Knowledgebase screen on our support site using your default browser.
- **ISCST3 Technical Guide:** Displays the ISCST3 Tech Guide using your default browser.
- **AERMOD Technical Guide:** Displays the AERMOD Tech Guide using your default browser.
- **Order Met Data...:** Loads the website through which you can order worldwide meteorological data.
- **About...:** Displays the copyright notice and version number for the interface.

Application Tools

The tools available in the **Application Toolbar** allow you to graphically define the location of your sources, building locations, and project reference point.

You can turn on and off the display of the **Application Toolbar** in the View menu. See the sections below for a description of each one the tools available in the Application Toolbar:



- Point Source Tool
- Flare Source Tool
- Area Source Tool
- Angled Area Source Tool
- Open Pit Source Tool
- Volume Source Tool
- Circle Area Source Tool
- Polygonal Area Source Tool
- Area Line Source Tool
- Volume Line Source Tool
- AERMOD Line Source
- Buoyant Line Source Tool
- Discrete Cartesian Receptor Tool
- Discrete Polar Receptor Tool
- Discrete Cartesian Receptor (ARC) Tool
- Uniform Cartesian Grid Tool
- Non-Uniform Cartesian Grid Tool
- Uniform Polar Grid Tool
- Non-Uniform Polar Grid Tool
- Nested Grid Receptors Tool

- Cartesian Plant Boundary Tool
- Elevations/Flagpole Heights for Discrete Receptors Tool
- Rectangular Building Tool
- Angled Rectangular Building Tool
- Circular Building Tool
- Polygonal Building Tool
- Show/Hide Structure Influence Zone (SIZ) Tool
- Show/Hide GEP 5L Area of Influence Tool

> There is an option in the Preferences - General page to allow toolbar buttons to be grouped together based on common functionality. This allows for simplification of the toolbar.

Point Source Tool

The **Point Source** tool allows you to graphically define the location of a point source on the drawing area.

How to Graphically Define a Point Source:

1. Press the **Point Source** tool ( ) located on the Application Toolbar.

2. Left-click, with the mouse pointer on the drawing area, the desired location of the point source. Note that as you move the mouse the current coordinates are displayed on the status bar.

3. The Source Inputs dialog is displayed to allow you to adjust the coordinates, if necessary, and define additional point source information.

4. When you finish entering all the required information, press the **OK** button. A marker ( ) will be placed at the selected location for the point source.

> If you want to review or modify any information for a particular source, press the Select tool ( ) on the Annotation Toolbar, select the desired source, click the

right mouse button and select **Edit** from the floating menu. The Source Inputs dialog will be displayed.

Flare Source Tool

The **Flare Source** tool allows you to graphically define the location of a flare source on the drawing area.

How to Graphically Define a Flare Source:

1. Press the **Flare Source** tool ( ) located on the Application Toolbar.

2. Left-click, with the mouse pointer on the drawing area, the desired location of the flare source. Note that as you move the mouse the current coordinates are displayed on the status bar.

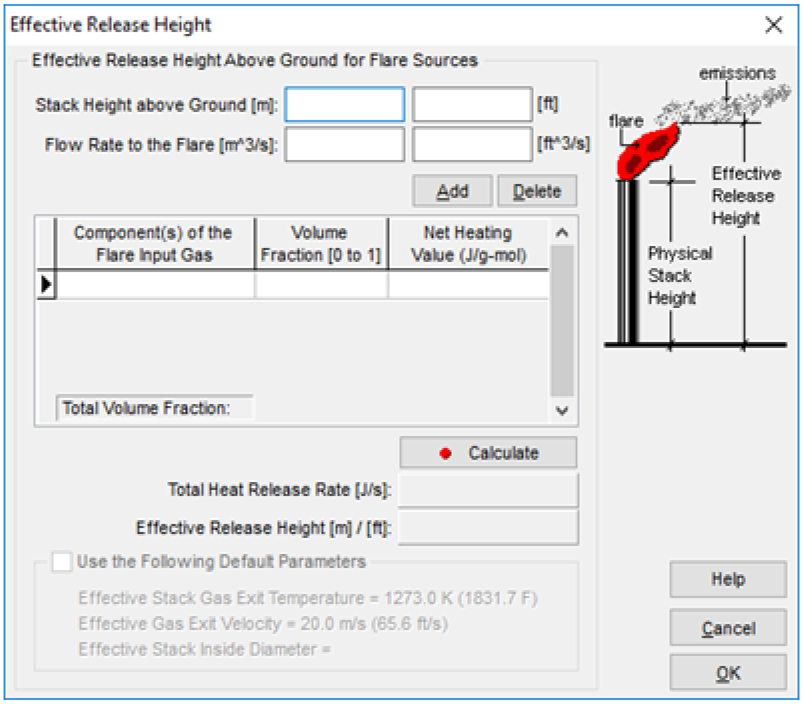
3. The Source Inputs dialog is displayed to allow you to adjust the coordinates, if necessary, and define additional flare source information.

4. When you finish entering all the required information, press the Close button. A marker ( ) will be placed at the selected location for the flare source.

If you want to review or modify any information for a particular source, press the Select tool ( ) on the Annotation Toolbar, select the desired source, click the right mouse button and select **Edit** from the floating menu. The Source Inputs dialog will be displayed.

Effective Release Height

The **Effective Release Height** dialog allows you to calculate the effective flare release height above ground for flare sources. You have access to this dialog by pressing the Tip... button located in the Source Inputs dialog, only for flare sources. AERMOD View follows the procedures presented in the "Workbook of Screening Techniques for Assessing Impacts of Toxic Air Pollutants" (US EPA, 1992b) for calculating the effective release height above ground for flare sources.



Effective Release Height dialog

The following parameters are requested:

- **Stack Height above Ground:** Specify the height of the stack above ground in meters [m] or in feet [ft].
- **Flow Rate to the Flare:** Enter the volumetric feed gas flow rate in cubic meters per second (m^3/s) or in cubic feet per second (ft^3/s).
- **Component(s) of the Flare Input Gas:** Here you can enter the components of the gas stream for the flare source. You can click on the Add button to add a row in order to specify a new component to the gas.
- **Volume Fraction:** Enter here the volume fraction for each component of the gas stream. As an example; if the gas stream is made up of 50% methane, 40% carbon dioxide, and 10% ethane, then the volume fractions will be 0.5, 0.4, 0.1 respectively. The total of the volume fractions must add up to 1.
- **Net Heating Value:** Enter here the heat of combustion for each component of the gas stream in J/g-mol. If a component is not combustible (does not affect flame heat), then enter 0 for the net heating value.

Once you have entered all the above parameters, click on the Calculate button to get the calculated values for the following parameters:

- **Total Heat Release Rate [J/s]:** The total heat release rate is calculated by AERMOD View by using the Lahey & Davis equation as presented in US EPA, 1992. Approximately 45% of the total heat release is assumed to be radiated as sensible heat.

- **Effective Release Height [m]/[ft]:** The effective release height above ground is calculated by adding the flare height to the stack height using Beychok equation as presented in US EPA, 1992. This parameter is displayed in meters [m] or in feet [ft].

> **Release Height** value must be less than 3000 m (9842 ft).

If you wish, you can also use default values for the effective stack parameters. Check the **Use the Following Default Parameters** box to use the following default parameters:

- **Effective Stack Gas Exit Temperature:** This parameter is assumed to be 1,273K or i831.7 F.

- **Effective Stack Gas Exit Velocity:** This parameter is assumed to be 20 m/s or 65.6 ft/s.

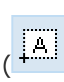
- **Effective Stack Diameter:** This parameter is calculated based on the heat release.

> The above effective stack parameters are somewhat arbitrary, but the resulting buoyancy flux estimate is expected to give reasonable final plume rise estimates for flares. However, since building downwash estimates depend on transitional buoyant plume rise calculations, the selection of effective stack parameters could influence the estimates. Therefore, building downwash estimates should be used with extra caution for flare sources (US EPA 1995b). If more realistic stack parameters can be determined for the above parameters, then you should not check the Use the Following Default Parameters box and instead specify your own values in the Source Inputs dialog.

Area-Rectangular Source Tool

The **Area-Rectangular Source** tool allows you to graphically define the location of an area source on the drawing area.

How to Graphically Define an Area Source:

1. Press the **Area-Rectangular Source** tool () located on the Application Toolbar.

2. Position the mouse pointer where one corner of the area source is located and left click. Drag the mouse at any angle you wish to the next corner of the area source and left click. You can then left drag up or down and left click when you reach the desired size of your area source. Note that as you move the mouse the current angle and coordinates are displayed on the status bar.

3. The Source Inputs dialog is displayed to allow you to adjust the coordinates, if necessary, and define additional area source information.

4. When you finish entering all the required information, press the **OK** button. A representation of the area source ( ) will be displayed on the drawing area at the specified location.

> If you want to review or modify any information for a particular source, press the Select tool ( ) on the Annotation Toolbar, select the desired source, click the right mouse button and select **Edit** from the floating menu. The Source Inputs dialog will be displayed.

Angled Area Source

The **Angled Area Source** tool allows you to graphically define the location of an area source on the drawing area that is at an angle.

How to Graphically Define a Flare Source:

1. Press the **Angled Area Source** tool ( ) located on the Application Toolbar.

2. Click at the location where the first corner of the area source is located. Then click at the second corner. Drag up or down on a diagonal, until you reach the desired size and shape, then click to finalize the shape. Note that as you move the mouse the current coordinates are displayed on the status bar.

3. The Source Inputs dialog is displayed to allow you to adjust the coordinates, if necessary, and define additional area source parameters.

4. When you finish entering all the required information, press the **Close** button. A representation of the area source ( ) will be displayed on the drawing area with the specified dimension and location.

> If you want to review or modify any information for a particular source, press the Select tool ( ) on the Annotation Toolbar, select the desired source, click the right mouse button and select **Edit** from the floating menu. The Source Inputs dialog will be displayed.

Open Pit Source Tool

The **Open Pit Source** tool allows you to graphically define the location of an open pit source on the drawing area.

How to Graphically Define an Open Pit Source:

1. Press the **Open Pit Source** tool ( ) located on the Application Toolbar.

2. Position the mouse pointer where one corner of the open pit source is located. Holding down the mouse left button, drag up or down on a diagonal, until you reach the desired size and shape. Note that as you move the mouse the current coordinates are displayed on the status bar.

3. The Source Inputs dialog is displayed to allow you to adjust the coordinates, if necessary, and define additional open pit source information.

4. When you finish entering all the required information, press the **Close** button. A representation of the open pit source ( ) will be displayed on the drawing area with the specified dimension and location.

> If you want to review or modify any information for a particular source, press the Select tool ( ) on the Annotation Toolbar, select the desired source, click the right mouse button and select **Edit** from the floating menu. The Source Inputs dialog will be displayed.

Volume Source Tool

The **Volume Source** tool allows you to graphically define the location of a volume source on the drawing area.

How to Graphically Define a Volume Source:

1. Press the **Volume** Source tool ( ) located on the Application Toolbar.

2. Position the mouse pointer where one corner of the volume source is located and left click. Drag the mouse at any angle you wish to the next corner of the volume source and left click. You can then left drag up or down and left click when you reach the desired size of your volume source. Note that as you move the mouse the current angle and coordinates are displayed on the status bar.

3. The Source Inputs dialog is displayed to allow you to adjust the coordinates, if necessary, and define additional volume source information.

4. When you finish entering all the required information, press the **OK** button. A representation of the volume source ( ) will be displayed on the drawing area with the specified dimension and location.

> If you want to review or modify any information for a particular source, press the Select tool ( ) on the Annotation Toolbar, select the desired source, click the right mouse button and select **Edit** from the floating menu. The Source Inputs dialog will be displayed.

Area-Circular Source Tool

The **Area-Circular Source** tool allows you to graphically define the location of an circular area source on the drawing area.

How to Graphically Define a Circular Area Source:

1. Press the **Area-Circular Source** tool ( ) located on the Application Toolbar.

2. Position the mouse pointer on the drawing area, where the center of the circular area source is located. Holding down the left mouse button, drag until you reach the desired size. As you drag with the mouse, the current radius of the circle being drawn and the coordinates of the center of the circular area source are displayed on the status bar. When you have reached the desired size release the left mouse button.

3. The Source Inputs dialog is displayed to allow you to adjust the coordinates, if necessary, and define additional circular area source information.

4. When you finish entering all the required information, press the **Close** button. A representation of the circular area source ( ) will be displayed on the drawing area with the specified dimension and location.

> If you want to review or modify any information for a particular source, press the Select tool ( ) on the Annotation Toolbar, select the desired source, click the right mouse button and select **Edit** from the floating menu. The Source Inputs dialog will be displayed.

Area-Polygonal Source Tool

The **Area-Polygonal Source** tool allows you to graphically define the location of an polygon area source on the drawing area.

How to Graphically Define an Polygon Area Source:

1. Press the **Area-Polygonal Source** tool ( ) located on the Application Toolbar.

2. Left-click, with the mouse pointer on the drawing area, the location for one of the corners of the polygon area source. Release the left mouse button. Drag the mouse pointer to the location of the next (adjacent) corner and click the left mouse button. Follow this procedure until you digitize all corners. To close the polygon, left-click with the mouse pointer inside the small box that marks the starting corner, or click the right mouse button.

3. The Source Inputs dialog is displayed to allow you to adjust the coordinates, if necessary, and define additional polygon area source information.

4. When you finish entering all the required information, press the **Close** button. A representation of the polygon area source ( ) will be displayed on the drawing area at the specified location.

> If you want to review or modify any information for a particular source, press the Select tool ( ) on the Annotation Toolbar, select the desired source, click the right mouse button and select **Edit** from the floating menu. The Source Inputs dialog will be displayed.

Line-Volume Source Tool

The **Line-Volume Source** tool allows you to graphically define the location of a line source on the drawing area.

How to Graphically Define a Volume Line Source:

1. Press the **Line-Volume Source** tool ( ) located on the Application Toolbar.

2. Left-click, with the mouse pointer on the drawing area, at the location for one end of the line source. Continue to left-click at each desired line segment. When complete, right-click the mouse to finalize the line.

3. The Source Inputs dialog is displayed, allowing you to adjust the coordinates, if necessary, and define additional source information.

4. When you finish entering all the required information, press the **OK** button. A representation of the line source will be displayed on the drawing area at the specified location.

> If you want to review or modify any information for a particular source, press the Select tool ( ) on the Annotation Toolbar, select the desired source, click the right mouse button and select **Edit** from the floating menu. The Source Inputs dialog will be displayed.

Line-Area Source Tool

The **Line-Area Source** tool allows you to graphically define the location of a line source on the drawing area.

How to Graphically Define an Area Line Source:

1. Press the **Line-Area Source** tool ( ) located on the Application Toolbar.

2. Left-click, with the mouse pointer on the drawing area, at the location for one end of the line source. Continue to left-click at each desired line segment. When complete, right-click the mouse to finalize the line.

3. The Source Inputs dialog is displayed, allowing you to adjust the coordinates, if necessary, and define additional source information.

4. When you finish entering all the required information, press the **OK** button. A representation of the line source will be displayed on the drawing area at the specified location.

> If you want to review or modify any information for a particular source, press the Select tool ( ) on the Annotation Toolbar, select the desired source, click the right mouse button and select **Edit** from the floating menu. The Source Inputs dialog will be displayed.

AERMOD Line Source Tool

The **AERMOD Line Source** tool allows you to define a one segment line source with start/end points and width.

How to Graphically Define a Line Source:

1. Click the **AERMOD Line Source** tool ( ) button located on the Application Toolbar.

2. Left-click and drag the mouse pointer from start point to end point of your line source.

3. At the end point of the source let go of the mouse button.

4. The Source Inputs dialog is displayed, allowing you to adjust the coordinates, if necessary, and define additional source information.

5. When you finish entering all the required information, press the **OK** button. A representation of the line source will be displayed on the drawing area at the specified location.

> If you want to review or modify any information for a particular source, press the Select tool ( ) on the Annotation Toolbar, select the desired source, click the right mouse button and select **Edit** from the floating menu. The Source Inputs dialog will be displayed.

Buoyant Line Source Tool

The **Buoyant Line Source** tool allows you to graphically define the location of a line source on the drawing area.

How to Graphically Define a Line Source:

1. Press the **Buoyant Line Source** tool ( ) located on the Application Toolbar.

2. Left-click, with the mouse pointer on the drawing area, at the location for one end of the line source. Drag the mouse pointer to the location of the end of the line and release the left mouse button.

3. The Source Inputs dialog is displayed, allowing you to adjust the coordinates, if necessary, and define additional source information.

4. When you finish entering all the required information, press the **OK** button. A representation of the line source ( ) will be displayed on the drawing area at the specified location.

If you want to review or modify any information for a particular source, press the Select tool ( ) on the Annotation Toolbar, select the desired source, click the right mouse button and select **Edit** from the floating menu. The Source Inputs dialog will be displayed.

This tool is NOT suitable for defining roads. To define roads please use Area Line Source Tool or Volume Line Source Tool.

Discrete Receptor Tool

The **Discrete Receptor** tool allows you to graphically define the location of discrete receptors.

How to Graphically Define a Discrete Receptor:

1. Select the **Discrete Receptor** tool ( ) located on the Application Toolbar.

2. Left-click, with the mouse pointer on the drawing area, the desired location for the discrete receptor.

3. The Discrete Receptors dialog is displayed, allowing you to adjust the coordinates, if necessary, and to define additional receptor information.

4. When you finish entering all the required information, click the **OK** button. A cross marker (+) will be placed at the selected location for the discrete receptor.

> If you want to review or modify any information for a particular receptor, click the Select tool ( ) located on the Annotation Toolbar, select the desired receptor, click the right mouse button and select **Edit** from the floating menu. The Discrete Receptors dialog will be displayed.

Discrete Polar Receptor Tool

The **Discrete Polar Receptor** tool allows you to graphically define the location of discrete polar receptors.

How to Graphically Define a Discrete Polar Receptor:

1. Select the **Discrete Polar Receptor** tool ( ) located on the Application Toolbar.

2. Left-click, with the mouse pointer on the drawing area, the desired location for the discrete polar receptor. To graphically define multiple discrete polar receptors at once, keep the Shift key

pressed down while you click with the left mouse button. When defining the last receptor, release the Shift key.

3. The **Select Reference Source for Polar Receptor** dialog is displayed where you must select the reference source for the polar receptor. Select the desired source from the drop-down list. Click the **OK** button.

4. The Discrete Polar Receptors dialog is displayed allowing you to adjust the coordinates if necessary, and to define additional receptor information.

5. When you have finished entering all the necessary information press the **Close** button. A cross marker (+ - — - —) will be placed at the selected location for the receptor with a dashed line attaching the receptor to the reference source.

> If you want to review or modify any information for a particular receptor press the Select tool ( ) on the Annotation Toolbar, select the desired receptor, click the right mouse button and select **Edit** from the floating menu. The Discrete Polar Receptors dialog will be displayed.

Discrete Cartesian Receptor (ARC) Tool

AERMOD Only

The **Discrete Cartesian Receptor (ARC)** tool allows you to graphically define the location of discrete Cartesian ARC receptors.

How to Graphically Define a Discrete Cartesian Receptor (ARC):

1. Select the **Discrete Cartesian Receptor (ARC)** tool (ARC) located on the Application Toolbar.

2. Left-click with the mouse pointer on the drawing area, the desired location for the receptor. To graphically define multiple receptors at once, keep the Shift key pressed down while you click with the left mouse button. When defining the last receptor, release the Shift key.

3. The Discrete ARC Receptors dialog is displayed allowing you to adjust the coordinates if necessary, and to define additional receptor information.

4. When you finish entering all the required data press the Close button. A cross marker (+) will be placed at the selected location for the receptor.

> If you want to review or modify any information for a particular receptor press the Select tool ( ) on the Annotation Toolbar, select the desired receptor, click the right mouse button and select **Edit** from the floating menu. The Discrete ARC Receptors dialog will be displayed.

Uniform Cartesian Grid Tool

The **Uniform Cartesian Grid** tool allows you to graphically define the location of a uniform Cartesian receptor grid on the drawing area.

How to Graphically Define a Uniform Cartesian Receptor Grid:

1. Select the **Uniform Cartesian Grid** tool ( ) located on the Application Toolbar.

2. Left-click, with the mouse pointer on the drawing area, the desired location for one of the grid corners. Holding down the left mouse button drag the mouse pointer diagonally until you reach the desired grid size. Release the left mouse button.

3. The Uniform Cartesian Grid dialog is displayed to allow you to adjust if necessary the origin coordinates, number of X and Y points, and spacing.

4. When you finish entering all the required information, press the **Close** button. A representation of the uniform Cartesian grid ( ) will be displayed on the drawing area at the specified location.

If you want to review or modify any information for a particular grid press the Select tool ( ) on the Annotation Toolbar, select the desired grid, click the right mouse button and select **Edit** from the floating menu. The Uniform Cartesian Grid dialog will be displayed.

Non-Uniform Cartesian Grid Tool

The **Non-Uniform Cartesian Grid** tool allows you to graphically define the location of a non-uniform Cartesian receptor grid on the drawing area.

How to Graphically Define a Non-Uniform Cartesian Grid:

1. Select the **Non-Uniform Cartesian Grid** tool ( ) located on the Application Toolbar.

2. Left-click, with the mouse pointer on the drawing area, the desired location for one of the grid corners. Holding down the left mouse button drag the mouse pointer diagonally until you reach the desired grid size. Release the left mouse button.

3. The Non-Uniform Cartesian Grid dialog is displayed to allow you to adjust if necessary the origin coordinates, number of X and Y points, and spacing.

4. When you finish entering all the required information, press the **Close** button. A representation of the non-uniform Cartesian receptor grid ( ) will be displayed on the drawing area at the specified location.

If you want to review or modify any information for a particular grid press the Select tool ( ) on the Annotation Toolbar, select the desired grid, click the right mouse button and select **Edit** from the floating menu. The Non-Uniform Cartesian Grid dialog will be displayed.

Uniform Polar Grid Tool

The **Uniform Polar Grid** tool allows you to graphically define the location of a uniform polar grid on the drawing area.

How to Graphically Define a Uniform Polar Grid:

1. Select the **Uniform Polar Grid** tool ( ) located on the Application Toolbar.

2. Position the mouse pointer on the drawing area, where the center of the receptor grid should be. Holding down the mouse left button, drag until you reach the desired size. As you drag with the mouse, the current radius of the uniform polar grid being drawn and the coordinates are displayed on the status bar. When you have reached the desired release the left mouse button.

3. The Uniform Polar Grid dialog is displayed to allow you to adjust if necessary the origin coordinates, number of X and Y points, and spacing.

4. When you finish entering all the required information, press the **Close** button. A representation of the uniform polar receptor grid ( ) will be displayed on the drawing area at the specified location.

> If you want to review or modify any information for a particular grid press the Select tool ( ) on the Annotation Toolbar, select the desired grid, click the right mouse button and select **Edit** from the floating menu. The Uniform Polar Grid dialog will be displayed.

Non-Uniform Polar Grid Tool

The **Non-Uniform Polar Grid** tool allows you to graphically define the location of a non-uniform polar receptor grid on the drawing area.

How to Graphically Define a Non-Uniform Polar Grid:

1. Select the **Non-Uniform Polar Grid** tool ( ) located on the Application Toolbar.

2. Position the mouse pointer on the drawing area, where the center of the receptor grid should be. Holding down the mouse left button, drag until you reach the desired size. As you drag with the mouse, the current radius of the non-uniform polar grid being drawn and the coordinates are displayed on the status bar. When you have reached the desired release the left mouse button.

3. The Non-Uniform Polar Grid dialog is displayed to allow you to adjust if necessary the origin coordinates, number of X and Y points, and spacing.

4. When you finish entering all the required information, press the **Close** button. A representation of the non-uniform polar receptor grid ( ) will be displayed on the drawing area at the specified location.

> If you want to review or modify any information for a particular grid press the Select tool ( ) on the Annotation Toolbar, select the desired grid, click the right mouse button and select **Edit** from the floating menu. The Non-Uniform Polar Grid dialog will be displayed.

Nested Grid Receptors Tool

The **Nested Grid Receptors** tool allows you to graphically define the location of a nested receptor grid on the drawing area.

How to Graphically Define a Nested Grid:

1. Select the **Nested Grid Receptors** tool ( ) located on the Application Toolbar.

2. Left-click with the mouse pointer on the drawing area, and holding down the left mouse button, drag the mouse pointer diagonally until you graphically define a box around the sources in your project. Release the left mouse button.

3. The Nested Grid Receptors dialog is displayed to allow you to adjust if necessary the origin coordinates, size, and receptor spacing.

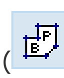
4. When you finish entering or reviewing the required information, click the **Close** button. A representation of the Nested Grid Receptors will be displayed on the drawing area at the specified location.

> If you want to review or modify any information for a particular grid click the Select tool () on the Annotation Toolbar, select the desired grid, click the right mouse button and select **Edit** from the floating menu. The Nested Grid Receptors dialog will be displayed.

Cartesian Plant Boundary Tool

The **Cartesian Plant Boundary** tool allows you to graphically define the location of a plant boundary or fence line on the drawing area.

How to Graphically Define a Plant Boundary:

1. Select the **Cartesian Plant Boundary** tool () located on the Application Toolbar.

2. Left-click, with the mouse pointer on the drawing area, the location for one of the corners of the plant boundary. Release the left mouse button. Drag the mouse pointer to the location of the next (adjacent) corner and click the left mouse button. Follow this procedure until you digitize all corners. To close the polygon, left-click with the mouse pointer inside the small box that marks the starting corner, or click the right mouse button.

3. The Cartesian Plant Boundary dialog is displayed to allow you to adjust if necessary the coordinates for the corners of the plant boundary polygon.

4. When you finish entering all the required information, click the **Close** button. A representation of the plant boundary ( ) will be displayed on the drawing area at the specified location.

> If you want to review or modify any information for a plant boundary click the Select tool ( ) on the Annotation Toolbar, select the plant boundary, click the right mouse button and select **Edit** from the floating menu. The Cartesian Plant Boundary dialog will be displayed.

Elevations/Flagpole Heights for Discrete Receptors Tool

The **Elevations/Flagpole Heights** tool allows you to graphically define terrain elevations and/or flagpole heights for discrete receptors. The Elevations/Flagpole Heights tool does not apply to the following types of receptors:

- Uniform and Non-Uniform Cartesian Grids
- Uniform and Non-Uniform Polar Grids
- Primary Receptors for the Cartesian Plant Boundary
- Polar Plant Boundary Receptors

> In order to use the Elevations/Flagpole Heights tool for the Cartesian and Polar grid receptors, you must first convert them to discrete receptors.

How to Graphically Define Elevations/Flagpole Heights:

1. Select the **Elevations/Flagpole Heights** tool ( ) located on the Application Toolbar.

2. Left-click, with the mouse pointer on the drawing area, the location for one of the corners of the polygon. Release the left mouse button. Drag the mouse pointer to the location of the next (adjacent) corner and click the left mouse button. Follow this procedure until you digitize all corners of the polygon. To close the polygon, left-click with the mouse pointer inside the small box that marks the starting corner, or click the right mouse button.

3. The Terrain Elevations/Flagpole Heights dialog is displayed to allowing you to specify the terrain elevation and/or flagpole height to be applied to each discrete receptor that falls within the defined polygon area.

4. When you finish entering all the required information, press the **Close** button.

Rectangular Building Tool

The **Rectangular Building** tool allows you to draw buildings that are rectangular or square in shape.

How to Graphically Define a Rectangular Building:

1. Select the **Rectangular Building** tool ( ) located on the Application Toolbar.

2. Position the mouse pointer where one corner of the building is located. Holding down the mouse left button, drag up or down on a diagonal, until you reach the desired size and shape. Note that as you move the mouse the current coordinates are displayed on the status bar.

3. The Building Inputs dialog is displayed. Complete the information and if necessary, adjust the coordinates and dimensions of the building.

4. When you finish entering all the required information, press the **OK** button. A representation of the rectangular building ( ) will be displayed on the drawing area for the selected dimensions and location.

> If you want to review or modify any information for a particular building, press the Select tool ( ) on the Annotation Toolbar, select the desired building, click the right mouse button and select **Edit** from the floating menu. The Building Inputs dialog will be displayed.

Angled Rectangular Building Tool

The **Angled Rectangular Building** tool allows you to draw buildings that are rectangular or square in shape at an angle.

How to Graphically Define an Angled Rectangular Building:

1. Select the **Angled Rectangular Building** tool ( ) located on the Application Toolbar.

2. Position the mouse pointer where one corner of the building is located and left click. Drag the mouse at any angle you wish to the next corner of the building and left click. You can then left drag up or down and left click when you reach the desired size of your building. Note that as you move the mouse the current angle and coordinates are displayed on the status bar.

3. The Building Inputs dialog is displayed. Complete the information and if necessary, adjust the coordinates and dimensions of the building.

4. When you finish entering all the required information, press the **OK** button. A representation of the angled rectangular building ( ) will be displayed on the drawing area for the selected dimensions and location.

> If you want to review or modify any information for a particular building, press the Select tool ( ) on the Annotation Toolbar, select the desired building, click the right mouse button and select **Edit** from the floating menu. The Building Inputs dialog will be displayed.

Circular Building Tool

The **Circular Building** tool allows you to draw buildings with a circular shape such as a tank.

How to Graphically Define a Circular Building:

1. Select the **Circular Building** tool ( ) located on the Application Toolbar.

2. Position the mouse pointer on the drawing area, where the center of the building is located. Holding down the left mouse button, drag until you reach the desired size. As you drag with the mouse, the current radius of the circle being drawn and the coordinates of the center of the building are displayed on the status bar. When you have reached the desired size release the left mouse button.

3. The Building Inputs dialog is displayed. Complete the information and if necessary, adjust the diameter and coordinates for the building.

4. When you finish entering all the required information, press the **OK** button. A representation of the circular building ( ) will be displayed on the drawing area at the selected location and at the specified diameter.

> If you want to review or modify any information for a particular building, press the Select tool ( ) on the Annotation Toolbar, select the desired building, click the right mouse button and select **Edit** from the floating menu. The Building Inputs dialog will be displayed.

Polygonal Building Tool

The **Polygonal Building** tool allows you to draw buildings of any shape, except rounded buildings.

How to Graphically Define a Polygonal Building:

1. Select the **Polygonal Building** tool ( ) located on the Application Toolbar.

2. Left-click, with the mouse pointer on the drawing area, the location for one of the corners of the polygonal building. Release the left mouse button. Drag the mouse pointer to the location of the next (adjacent) corner and click the left mouse button. Follow this procedure until you digitize all corners. To close the polygon, left-click with the mouse pointer inside the small box that marks the starting corner, or click the right mouse button.

3. The Building Inputs dialog is displayed. Complete the information and, if necessary, adjust the coordinates and dimensions of the building.

4. When you finish entering all the required information, press the **OK** button. A representation of the polygonal building ( ) will be displayed on the drawing area at the selected location and dimensions.

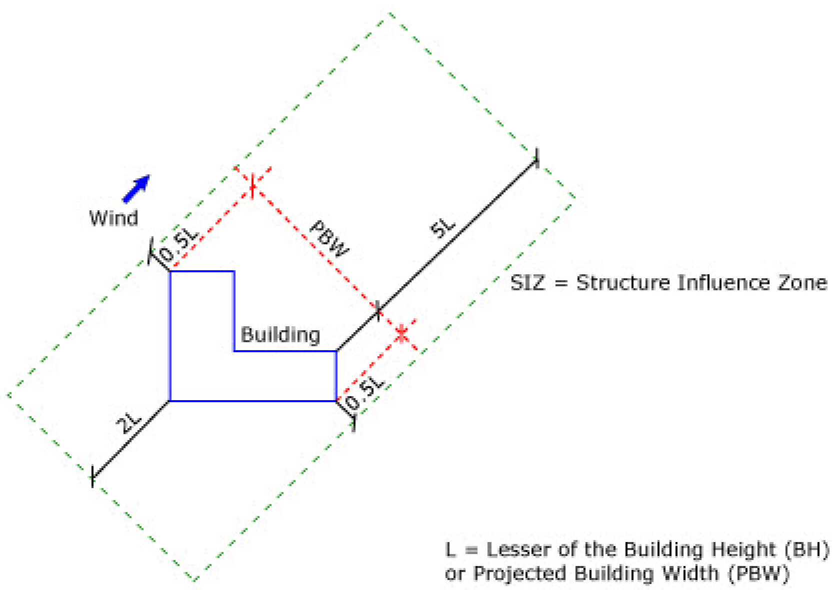
> If you want to review or modify any information for a particular building, press the Select tool ( ) on the Annotation Toolbar, select the desired building, click the right mouse button and select **Edit** from the floating menu. The Building Inputs dialog will be displayed.

Show/Hide Structure Influence Zone (SIZ) Tool

This tool allows you to view the **Structure Influence Zone (SIZ)** for a selected wind direction. Click on the **Show/Hide Structure Influence Zone (SIZ)** tool ( ) located on the Application Toolbar.

For building downwash analyses with direction-specific building dimensions, wake effects are assumed to occur if the stack is within a rectangle composed of two lines perpendicular to the wind direction, one at 5L downwind of the building and the other at 2L upwind of the building, and by two lines parallel to the wind direction, each at 0.5L away from each side of the building, as shown below. L is the lesser of the height and projected width of the building for the particular direction sector. This rectangular area has been termed a **Structure Influence Zone (SIZ)**.

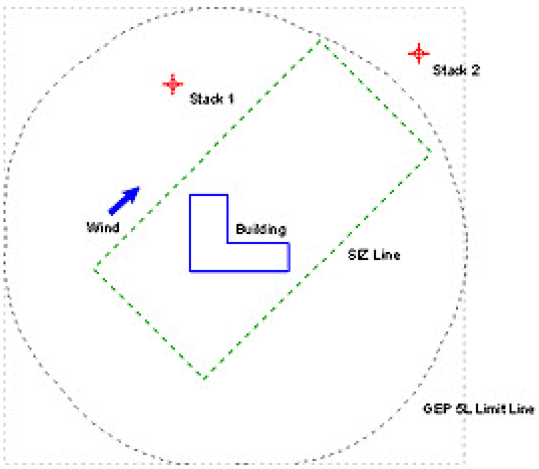
The **Structure Influence Zone** is displayed for any buildings in the drawing area and the **Wind Direction** dialog is opened. You can click on the up/down arrows to change the orientation of the wind flow or you can simply click on any of the markers around the arrow to select the desired wind flow vector. This changes the orientation of the SIZ.

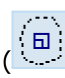


Show/Hide GEP 5L Area of Influence Tool

As the wind rotates full circle, each direction-specific Structure Influence Zone (SIZ) changes and is integrated into one overall area of influence termed the **GEP 5L Area of Influence**.

Each structure produces an area of wake effect influence that extends out to a distance of five times **L** directly downwind from the trailing edge of the structure, where L is the lesser of the BH or PBW. As the wind rotates full circle, each direction-specific area of influence changes and is integrated into one overall area of influence termed the **GEP 5L** Area of Influence. Any stack that is on or within the limit line is affected by GEP wake effects for some wind direction or range of wind direction.



Click on the **Show/Hide GEP 5L Area of Influence** tool () located on the Application Toolbar to display or hide the **GEP 5L Area of Influence** on the drawing area.

Annotation Tools

The **Annotation Toolbar** contains tools that allow you import base maps, manipulate what is visible on the drawing area, zoom in on locations of interest and make annotations that enhance the appearance of your project. The **Annotation Toolbar** can be moved anywhere on your screen by left-clicking your mouse and dragging the toolbar to where you would like it. See the following sections for a description of each of these tools:

Select Tool

Circular Select Tool

Polygonal Select Tool

Zoom In Tool

Zoom Out Tool

Zoom to Location

Pan Tool

View Min. Extents Tool

View Max. Extents Tool

Eagle Watch View Tool

Point/Rectangular Delete Tool

Circular Delete Tool

Polygonal Delete Tool

Rectangle Annotation Tool

Circle Annotation Tool

Polygon Annotation Tool

Text Annotation Tool

Marker Annotation Tool

Arrow Annotation Tool

Web Annotation Tool

North Arrow Annotation Tool

Measure Distances and Areas Tool

Map Import Tool

Site Domain Tool

Map Projection Dialog

Overlay Control Tool

Graphical Options

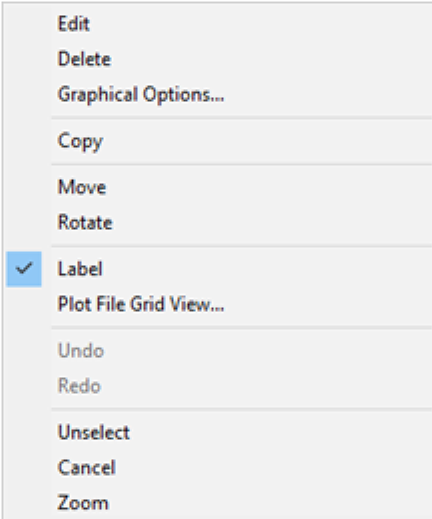
Impact Tool

Cross Section Tool

Identify Tool

Select Tool

With the **Select** tool ( ), you can select an object from the drawing area and right-click on it to display a floating menu from which you can perform several tasks such as move, resize, edit, delete, and label the object. The available options in the floating menu will be dependent on the type of object being selected.



Select Tool Menu

- **Edit:** Select this option if you would like to edit your object. Usually opens the object specific input dialog.

- **Delete:** Deletes the selected object. You can also delete an object by selecting it with the **Select** tool and pressing the **Delete** key.

- **Graphical Options:** Directly opens the **Graphical Options** dialog.

- **Move:** Select this option if you would like to move the object to another location.

- **Resize:** This option allows you to resize the selected object.

- **Rotate:** If you have selected a polygonal object, this option allows you to rotate the object 360º.

- **Edit/Add Points:** If you have selected a polygonal object such as a facility or area sub process, this option allows you to graphically edit or add one or more nodes of the object you have defined.

- **Delete Points:** If you have selected a polygonal object such as a facility or area sub process, you can graphically delete one or more nodes of the object you have defined.

- **Label:** Displays or hides the label for the object. A check indicates the label is currently displayed.

- **Undo:** Undo a change to the selected element, such as Move, Resize, Rotate.

> Does not undo **Delete**.

- **Redo:** Cancel the latest **Undo** action.

- **Plot File Grid Viewer:** Select this option to open the Plot File Grid Viewer where you can view the plot file results for the selected receptors.

- **Unselect:** Use this option to unselect an object.

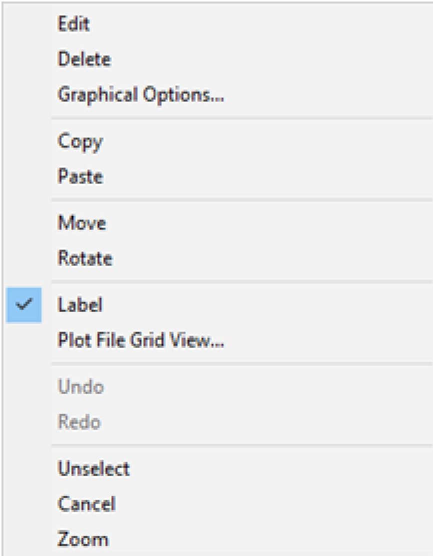
- **Cancel:** Select this option to cancel the menu.

- **Zoom:** Zooms in/out to the extents of the selected object.

> Some options may or may not be available depending on the selected object

Circular Select Tool

With the **Circular Select** tool (), you can select object within a circular radius. Click the left mouse button on the drawing area in the location where you want to place the circle. Holding down the left mouse button, draw a circle on the drawing area by dragging the mouse. Release the left mouse button when you have reached the desired circle size. All objects within the circle will be selected. Right click with the mouse to display the following menu:



- **Edit:** Select this option if you would like to edit your objects. Usually opens the object specific input dialog.

- **Delete:** Deletes the selected objects.

- **Graphical Options:** Opens the Graphical Options dialog where you can modify the appearance of the selected objects.

- **Copy:** You can use this option to copy a source from the graphical display.

- **Move:** Select this option if you would like to move the objects to another location.

- **Rotate:** This option allows you to rotate the selected objects.

- **Edit/Add Points:** If you have selected a polygonal object such as a facility or area sub process, this option allows you to graphically edit or add one or more nodes of the object you have defined.

- **Delete Points:** If you have selected a polygonal object such as a facility or area sub process, you can graphically delete one or more nodes of the object you have defined.

- **Label:** Displays or hides the label for the object. A check indicates the label is currently displayed.

- **Undo:** Undo a change to the selected element, such as Move, Resize, Rotate.

> Does not undo **Delete**.

- **Redo:** Cancel the latest **Undo** action.

- **Plot File Grid Viewer:** Select this option to open the Plot File Grid Viewer where you can view the plot file results for the selected receptors.

- **Unselect:** Use this option to unselect the objects.

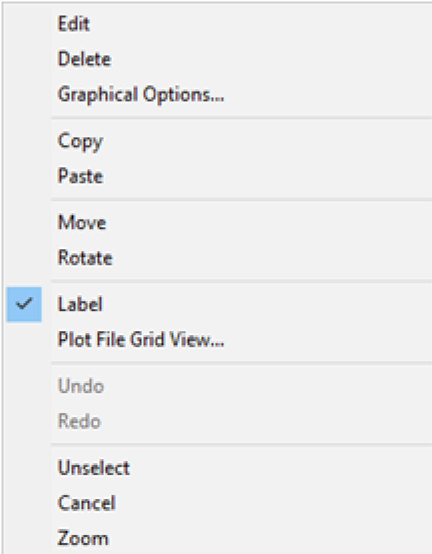
- **Cancel:** Select this option to cancel the menu.

- **Zoom:** Zoom to selection.

> Some options may or may not be available depending on the selected object

Polygonal Select Tool

With the **Polygonal Select** tool (), you can select object within a polygonal area. Left-click, with the mouse pointer on the drawing area, the location for one of the corners of the polygon. Release the left mouse button. Drag the mouse pointer to the location of the next (adjacent) corner and click the left mouse button. Follow this procedure until you digitize all corners. To close the polygon, left-click with the mouse pointer inside the small box that marks the starting corner, or click the right mouse button. All objects within the defined polygon will be selected. Right click with the mouse to display the following menu:



- **Edit:** Select this option if you would like to edit your objects. Usually opens the object specific input dialog.

- **Delete:** Deletes the selected objects.

- **Graphical Options:** Opens the Graphical Options dialog where you can modify the appearance of the selected objects.

- **Copy:** You can use this option to copy a source from the graphical display.

- **Move:** Select this option if you would like to move the objects to another location.

- **Rotate:** This option allows you to rotate the selected objects.

- **Edit/Add Points:** If you have selected a polygonal object such as a facility or area sub process, this option allows you to graphically edit or add one or more nodes of the object you have defined.

- **Delete Points:** If you have selected a polygonal object such as a facility or area sub process, you can graphically delete one or more nodes of the object you have defined.

- **Label:** Displays or hides the label for the object. A check indicates the label is currently displayed.

- **Undo:** Undo a change to the selected element, such as Move, Resize, Rotate.

> Does not undo **Delete**.

- **Redo:** Cancel the latest **Undo** action.

- **Plot File Grid Viewer:** Select this option to open the Plot File Grid Viewer where you can view the plot file results for the selected receptors.

- **Unselect:** Use this option to unselect the objects.

- **Cancel:** Select this option to cancel the menu.

- **Zoom:** Zoom to selection.

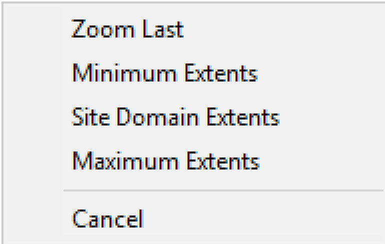
> Some options may or may not be available depending on the selected object

Zoom In Tool

The **Zoom In** tool () allows you to magnify a portion of the drawing area. To zoom into a portion of the drawing area, click on the Zoom In tool located on the Annotation Toolbar, move the cursor into the drawing area, holding down the mouse left button, drag up or down on a diagonal until you cover the area you want to magnify, and release the mouse left button.

You can also zoom in on an area of interest by clicking the left mouse button on the drawing area. The drawing area will be magnified and re-centered on the point where the mouse cursor was clicked.

Clicking the right mouse button over the drawing area will generate a pop-up menu providing further zoom control. The following menu options appear on this pop-up menu:



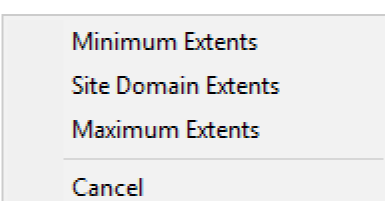
Zoom In Menu

- **Zoom Last:** Returns the drawing area to your previous view.
- **Minimum Extents:** Zooms the drawing area to the minimum project extents as defined by model objects.
- **Site Domain Extents:** Zooms the drawing area out to your entire site modeling area.
- **Maximum Extents:** Zooms the drawing area to the maximum project extents as defined by model objects.
- **Cancel:** Closes the pop-up menu allowing you to return to the drawing area.

Zoom Out Tool

The **Zoom Out** tool () allows you to zoom out of a portion of the drawing area. To zoom out, click on the Zoom Out tool located on the Annotation Toolbar, move the cursor into the drawing area over the area of interest and click the left mouse button.

Clicking the right mouse button over the drawing area will generate a pop-up menu providing further zoom control. The following menu options appear in this pop-up menu:



Zoom Out Menu

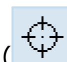
- **Minimum Extents:** Zooms the drawing area to the minimum project extents as defined by model objects.

- **Site Domain Extents:** Zooms the drawing area out to your entire site modeling area.

- **Maximum Extents:** Zooms the drawing area to the maximum project extents as defined by model objects such as your sources and buildings.

- **Cancel:** Closes the pop-up menu allowing you to return to the drawing area.

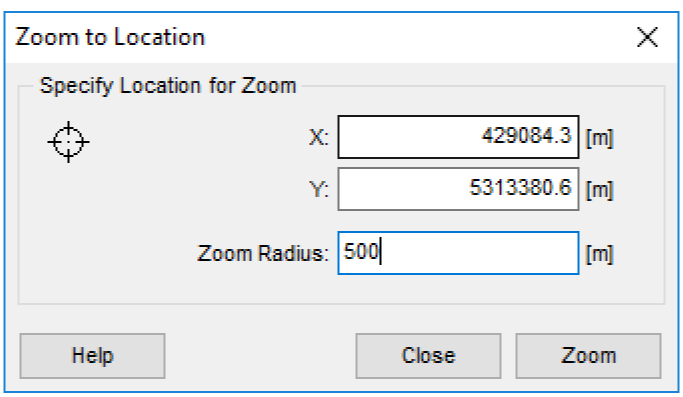
Zoom to Location

The **Zoom to Location** tool () allows you to zoom to a specified location.

How to Zoom to a Specific Location on the Drawing Area:

1. Press the **Zoom to Location** tool located on the Annotation Toolbar.

2. Click the left mouse button on the drawing area where you wish to zoom into. The **Zoom to Location** dialog is displayed.



Zoom to Location dialog

3. The **X** and **Y** coordinates where you clicked on the drawing area will be displayed. You can modify these coordinates if you wish. You will also need to specify the **Zoom Radius**.

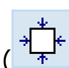
4. Press the **Zoom** button. You will be zoomed into the specified location.

Pan Tool

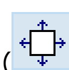
The **Pan** tool () lets you pan the view of your project easily by dragging the display in any direction with the mouse. This is useful if you zoom in and can no longer see the entire drawing area. The **Pan** tool becomes active only after you have used the **Zoom In** tool.

To pan, click the **Pan** tool located on the Annotation Toolbar and the hand cursor appears. Move the cursor anywhere over the drawing area, press and hold down the left mouse button and drag in the desired direction. Release the mouse button when you reach the desired display.

View Min. Extents Tool

Clicking on this tool () sets your view to the minimum extents which encompasses only the objects in you modeling area such as your point and area sources.

View Max. Extents Tool

Clicking on this tool () sets your view to the maximum extents as defined by model objects (i.e. point and area sources) and all other layers in your modeling project (i.e. base maps).

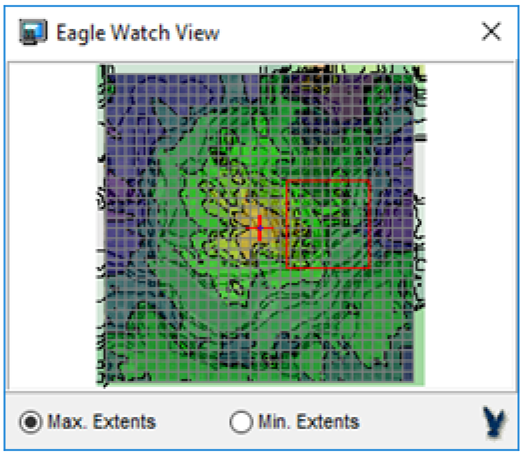
Eagle Watch View Tool

The **Eagle Watch View** tool () displays a small dialog showing the full extents of your modeling domain with a red rectangle marking the area that is currently being showed in the main graphical area. This tool can be especially useful as it allows you to zoom into an area in context with the entire modeling domain.

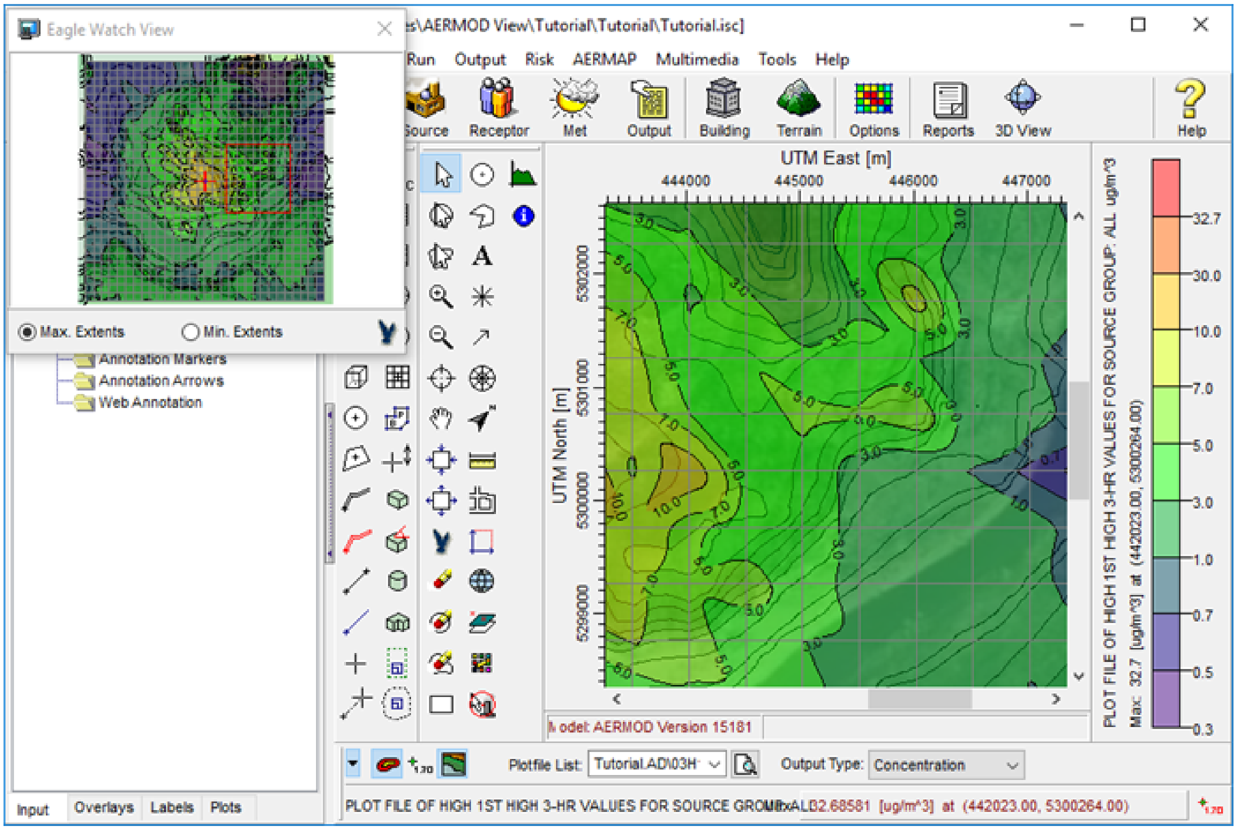
How to Use the Eagle Watch Tool:

1. Press the **Eagle Watch View** tool located on the Annotation Toolbar to open the **Eagle Watch View** dialog which displays a preview of your modeling site.

2. Using the mouse, left-click to draw a red box around the area you wish to preview. Notice that when you release the mouse button, the selected area will be displayed in the drawing area of the main interface.



Eagle Watch View Dialog



Eagle Watch View - Example

3. When the desired area is displayed in the drawing area, click the red close button on the top right corner of the **Eagle Watch View dialog**.

Point/Rectangular Delete Tool

The **Point/Rectangular Delete** tool () can be used to delete an object from the drawing area.

How to Use the Point/Rectangular Delete Tool:

1. Click the **Point/Rectangular Delete** tool located on the Annotation Toolbar and then click the left mouse button on the drawing area in the location where you want to place the starting corner of the rectangle.

2. Holding down the left mouse button, draw a rectangle on the drawing area over top of the target object by dragging the mouse.

3. Release the left mouse button. The Delete Objects dialog is displayed.

4. The **Delete Objects** dialog lists all the objects that have been selected for deletion. A box can be found beside each object allowing you to check the box for any objects in the list that you want to delete.

> You can also select the **Point/Rectangular Delete Tool** and just click on the object you would like to delete without having to draw a rectangle as described in the above steps.

Circular Delete Tool

The **Circular Delete** tool () can be used to delete an object from the drawing area.

How to Use the Circular Delete Tool:

1. Click the **Circular Delete** tool located on the Annotation Toolbar and then click the left mouse button on the drawing area in the location where you want to place the center of the circle.

2. Holding down the left mouse button, draw a circle on the drawing area by dragging the mouse.

3. Release the left mouse button. The Delete Objects dialog is displayed.

4. The **Delete Objects** dialog lists all the objects that have been selected for deletion. A box can be found beside each object allowing you to check the box for any objects in the list that you want to delete.

Polygonal Delete Tool

The **Polygonal Delete** tool () can be used to delete an object from the drawing area.

How to Use the Polygonal Delete Tool:

1. Click the **Polygonal Delete** tool located on the Annotation Toolbar and then left-click, with the mouse pointer on the drawing area, the location for one of the corners of the polygon. Release the left mouse button.

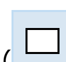
2. Drag the mouse pointer to the location of the next (adjacent) corner and click the left mouse button.

3. Follow this procedure until you digitize all corners of the polygon.

4. To close the polygon, left-click with the mouse pointer inside the small box that marks the starting corner, or click the right mouse button. The Delete Objects dialog is displayed.

5. The **Delete Objects** dialog lists all the objects that have been selected for deletion. A box can be found beside each object allowing you to check the box for any objects in the list that you want to delete.

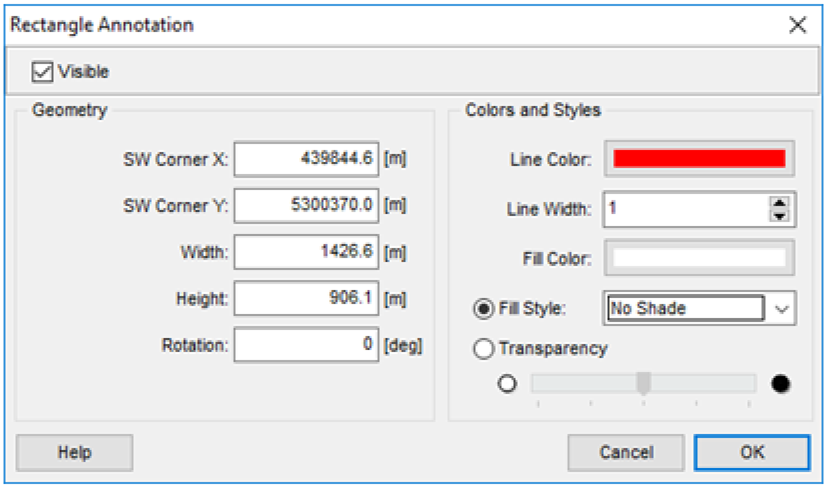
Rectangle Annotation Tool

The **Rectangle Annotation** tool () is used to draw rectangles on the drawing area.

How to Draw a Rectangle on the Drawing Area:

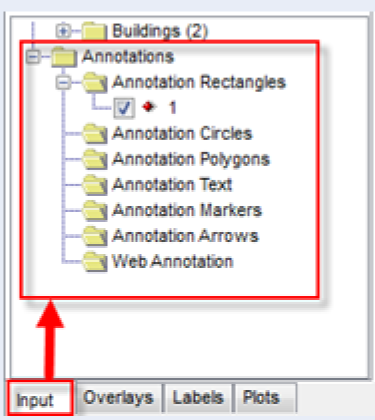
1. Press the **Rectangle Annotation** tool located on the Annotation Toolbar.

2. Click the left mouse button on the drawing area in the location where you want to place one of the corners of the rectangle. Holding down the left mouse button, draw a rectangle on the drawing area by dragging the mouse up or down on a diagonal. Release the left mouse button. The **Rectangle Annotation** dialog is displayed.



Rectangle Annotation dialog

The **Visible** option allows you to enable or disable the display of annotation on the map interface. After an annotation is created, you can click on the **Input** tab located in the bottom left of the interface to access the corresponding **Annotations** folder (e.g., Annotation Rectangles) with a check box that allows you to either hide or show the annotation on the map interface.

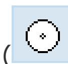


3. The X and Y coordinates of the SW corner of the rectangle, as well as the width, height and the rotation angle of the rectangle is displayed in the **Geometry** panel. You can change these values at any time if you desire.

4. In the **Colors and Styles** panel, choose the color and width for the line, the fill color and style, and the transparency level of the rectangle and press the **OK** button.

5. The rectangle is placed on the drawing area in the location and style you have previously selected.

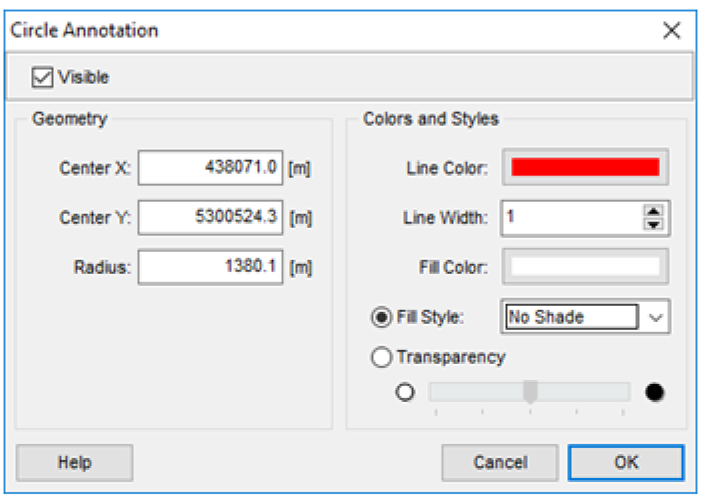
Circle Annotation

The **Circle Annotation** tool () is used to draw circles on the drawing area.

How to Draw a Circle on the Drawing Area:

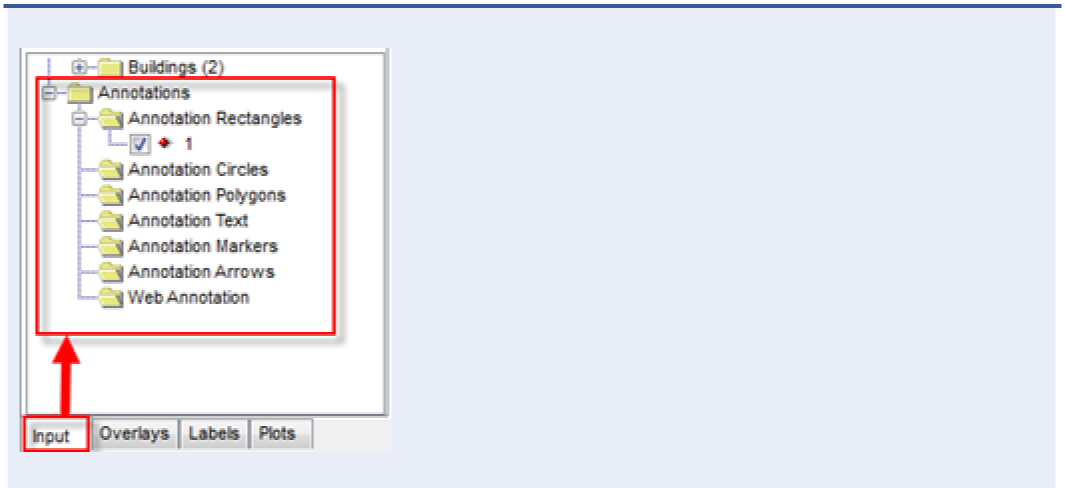
1. Press the **Circle Annotation** tool located on the Annotation Toolbar.

2. Click the left mouse button on the drawing area in the location where you want to place the circle. Holding down the left mouse button, draw a circle on the drawing area by dragging the mouse. Release the left mouse button when you have reached the desired circle size. The **Circle Annotation** dialog is displayed.



Circle Annotation dialog

> The **Visible** option allows you to enable or disable the display of annotation on the map interface. After an annotation is created, you can click on the **Input** tab located in the bottom left of the interface to access the corresponding **Annotations** folder (e.g., Annotation Rectangles) with a check box that allows you to either hide or show the annotation on the map interface.



3. The X and Y coordinates of the SW corner of the rectangle, as well as the width, height and the rotation angle of the rectangle is displayed in the **Geometry** panel. You can change these values at any time if you desire.

4. Choose the color for the line, the line width, the fill color and style of the circle, and the transparency level and press the **OK** button.

5. The circle is placed on the drawing area in the location and style you have selected.

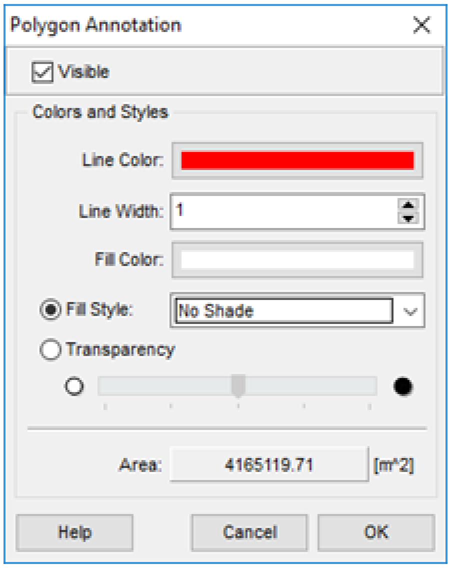
Polygon Annotation Tool

The **Polygon Annotation** tool () is used to draw rectangles on the drawing area.

How to Draw a Polygon on the Drawing Area:

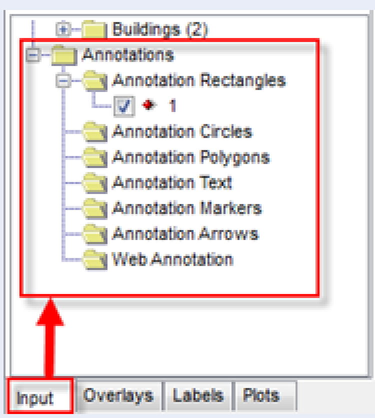
1. Press the **Polygon Annotation** tool located on the Annotation Toolbar.

2. Left-click, with the mouse pointer on the drawing area, the location for one of the corners of the polygon. Release the left mouse button. Drag the mouse pointer to the location of the next (adjacent) corner and click the left mouse button. Follow this procedure until you digitize all corners. To close the polygon, left-click with the mouse pointer inside the small box that marks the starting corner, or click the right mouse button. The **Polygon Annotation** dialog is displayed.



Polygon Annotation dialog

The **Visible** option allows you to enable or disable the display of annotation on the map interface. After an annotation is created, you can click on the **Input** tab located in the bottom left of the interface to access the corresponding **Annotations** folder (e.g., Annotation Rectangles) with a check box that allows you to either hide or show the annotation on the map interface.



3. The X and Y coordinates of the SW corner of the rectangle, as well as the width, height and the rotation angle of the rectangle is displayed in the **Geometry** panel. You can change these values at any time if you desire.

4. The polygon is placed on the drawing area in the location and style you have previously selected.

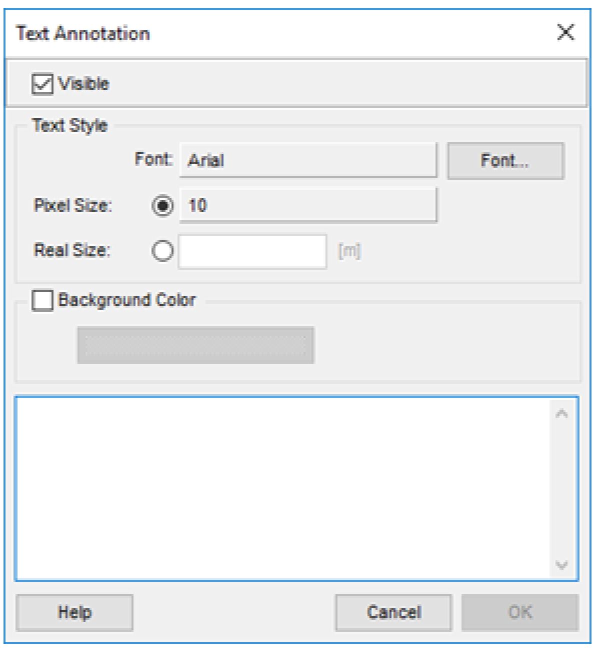
Text Annotation Tool

The **Text Annotation** tool () is used to write text on the drawing area. This allows you to add more information to your printouts.

How to Write Text on the Drawing Area:

1. Press the **Text Annotation** tool located on the Annotation Toolbar.

2. Click the left mouse button on the drawing area where you want the beginning of the text to be placed. The **Text Annotation** dialog is displayed.



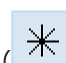
Text Annotation dialog

The **Visible** option allows you to enable or disable the display of annotation on the map interface. After an annotation is created, you can click on the **Input** tab located in the bottom left of the interface to access the corresponding **Annotations** folder (e.g., Annotation Rectangles) with a check box that allows you to either hide or show the annotation on the map interface.

3. Type the text in the dialog and press the **Font** button if you want to select different font options such as type, style, color and size. Specifically, you can adjust the size of the text annotation by either the pixel size or the real size as shown in the map interface. If desired, check the **Background Color** box to change the background color of the text. You can click on the color bar to open the Color dialog where you can select a color of your choice.

4. Press the **OK** button. The text is placed on the drawing area in the location you have previously selected.

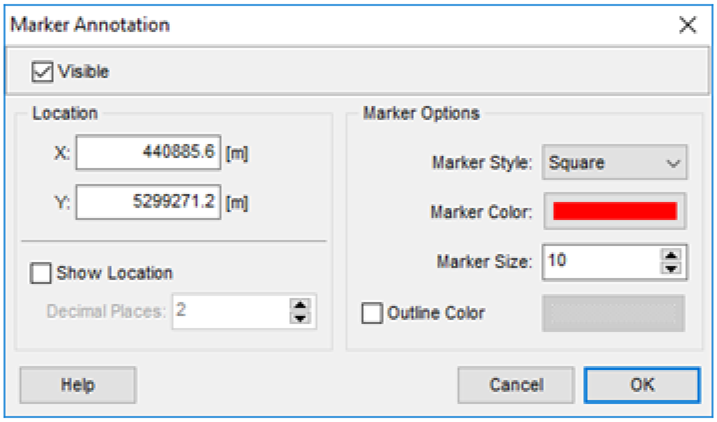
Marker Annotation Tool

The **Marker Annotation** tool () is used to draw markers on the drawing area.

How to Draw Markers on the Drawing Area:

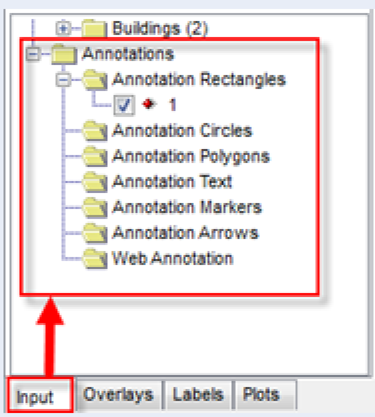
1. Press the **Marker Annotation** tool on the Annotation Toolbar.

2. Click the left mouse button on the drawing area, where you want the marker to be placed. The **Marker Annotation** dialog is displayed.



Marker Annotation dialog

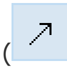
The **Visible** option allows you to enable or disable the display of annotation on the map interface. After an annotation is created, you can click on the **Input** tab located in the bottom left of the interface to access the corresponding **Annotations** folder (e.g., Annotation Rectangles) with a check box that allows you to either hide or show the annotation on the map interface.



3. The X and Y coordinates of the marker are displayed in the Location panel. You can change these values at any time if you desire. Select the marker style, color and size, and the outline color if desired and press the **OK** button.

4. The marker is placed on the drawing area in the location you have previously selected.

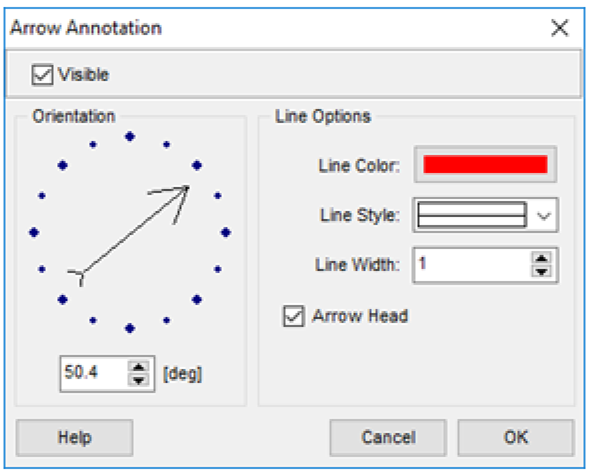
Arrow Annotation

The **Arrow Annotation** tool () is used to draw arrows or lines on the drawing area.

How to Draw Arrows and Lines on the Drawing Area:

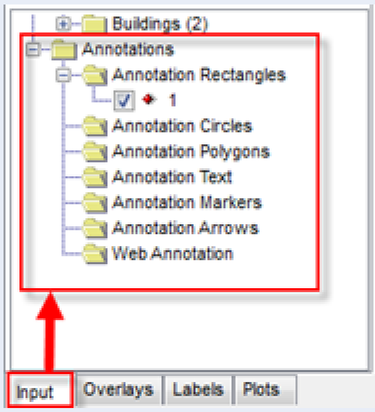
1. Press the **Arrow Annotation** tool located on the Annotation Toolbar.

2. Click the left mouse button on the drawing area in the location where the arrow head should start. Holding down the left mouse button, draw a line on the drawing area. Release the left mouse button. The **Arrow Annotation** dialog is displayed.



Arrow Annotation dialog

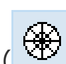
> The **Visible** option allows you to enable or disable the display of annotation on the map interface. After an annotation is created, you can click on the **Input** tab located in the bottom left of the interface to access the corresponding **Annotations** folder (e.g., Annotation Rectangles) with a check box that allows you to either hide or show the annotation on the map interface.



3. The X and Y coordinates of the SW corner of the rectangle, as well as the width, height and the rotation angle of the rectangle is displayed in the **Geometry** panel. You can change these values at any time if you desire.

4. The arrow or line is placed on the drawing area in the location and direction you have previously selected.

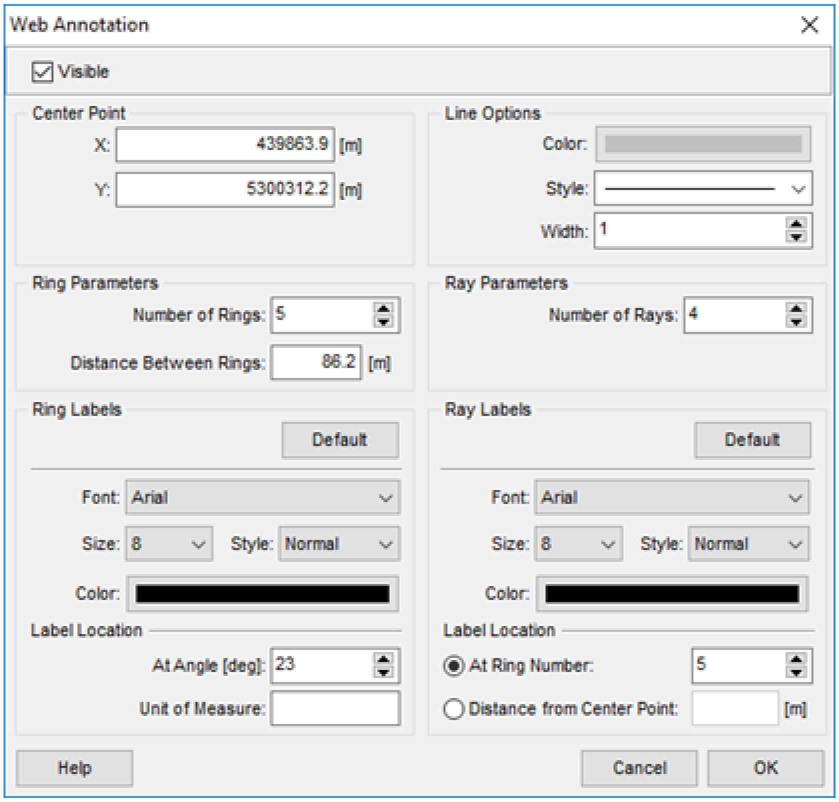
Web Annotation Tool

The **Web Annotation** tool () allows you to draw a polar grid with the option to specify rings, rays, and labels at specific distances for annotation purposes only. This annotation tool can be especially useful when used as a scaling tool for graphical visualization of the modeling area and its dimensions.

How to Draw a Web Annotation on the Drawing Area:

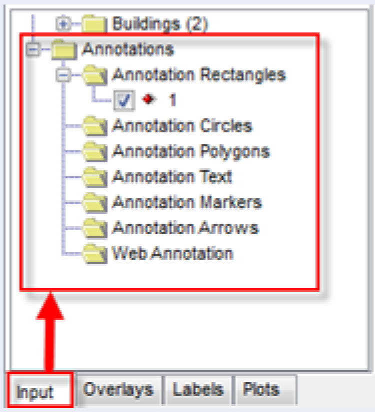
1. Click on the **Web Annotation** tool located on the Annotation Toolbar.

2. Click the left mouse button on the drawing area in the location where you want the center point of the web annotation to originate from. Holding down the left mouse button, draw a circle on the drawing area by dragging the mouse. Release the left mouse button when you have reached the desired web annotation size. The **Web Annotation** dialog is displayed.



Web Annotation dialog

The **Visible** option allows you to enable or disable the display of annotation on the map interface. After an annotation is created, you can click on the **Input** tab located in the bottom left of the interface to access the corresponding **Annotations** folder (e.g., Annotation Rectangles) with a check box that allows you to either hide or show the annotation on the map interface.



3. The following web annotation parameters can be changed in the **Web Annotation** dialog:

Center Point

- **Coordinate X:** Here you can specify the X Coordinate for the center point of your web annotation.
- **Coordinate Y:** Here you can specify the Y Coordinate for the center point of your web annotation.

Ring Parameters

- **Number of Rings:** Specify the number of rings you would like displayed in the web annotation.
- **Distance Between Rings:** Specify the distance between each ring displayed in the web annotation.

Ring Labels

- **Default:** Click on this button to return to all of the default **Ring Label** settings.
- **Font:** Select a font from the drop-down list.

- **Size:** Select a font size from the drop-down list.
- **Style:** Select a font style from the drop-down list.
- **Color:** Click on the color bar to display the Color dialog from where you can select a color for the labels.
- **Label Location**
  - **At Angle [deg]:** Specify the angle location where you want the ring label displayed.
  - **Units of Measure:** Specify the measurement unit for the ring label. The unit of measure must be the same as the one specified in the drawing area (e.g., meters for UTM coordinates, degrees for geographic coordinates).

Line Options

- **Color:** Click on the color bar to display the Color dialog where you can select the color for the lines in the web annotation.
- **Style:** Select a line style from the drop-down list.
- **Width:** Specify the thickness of the lines in the web annotation.

Ray Parameters

- **Number of Rays:** Specify the number of rays you would like displayed in the web annotation.

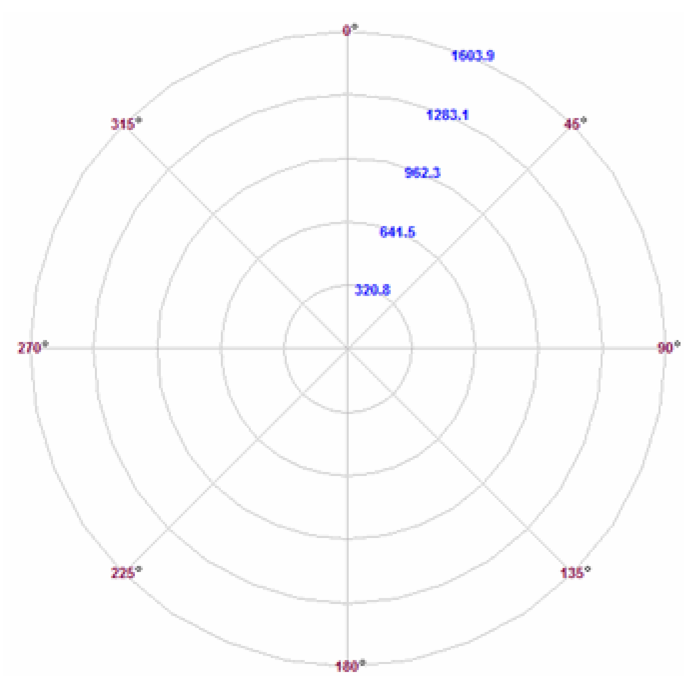
If you specify 4 rays, your web annotation will be divided into 8 sectors.

Ray Labels

- **Default:** Click on this button to return to all of the default **Ray Label** settings.

- **Font:** Select a font from the drop-down list.
- **Size:** Select a font size from the drop-down list.
- **Style:** Select a font style from the drop-down list.
- **Color:** Click on the color bar to display the Color dialog from where you can select a color for the labels.
- **Label Location**
    - **At Ring Number:** Select this option if you want the label to be placed on a specific ring.
    - **Distance from Center Point:** Select this option if you want to specify the distance from the center point of the ray where you want the ray label displayed.

5. Once you have finished defining the parameters of your web annotation, click **OK**. The following is an example of how your drawing area will appear now that you have generated a web annotation.



Web Annotation Preview

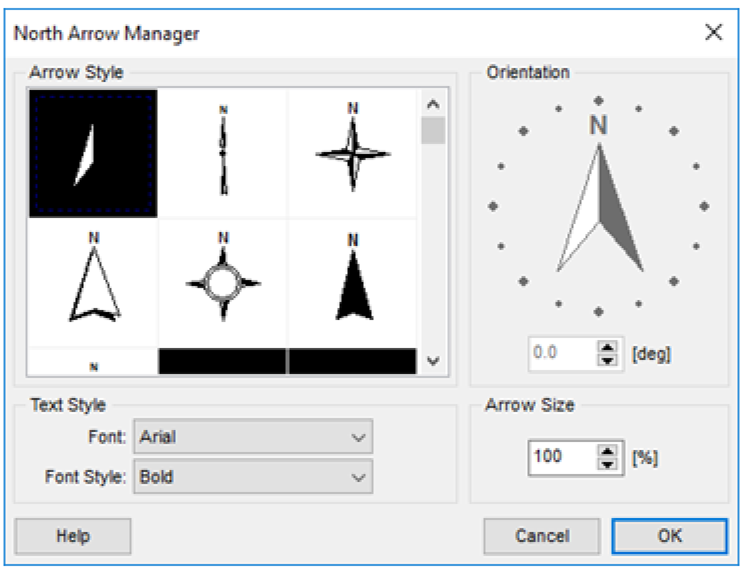
North Arrow Annotation Tool

The **North Arrow Annotation** tool () allows you to indicate the orientation of your modeling domain.

How to Draw a North Arrow on the Drawing Area:

1. Press the **North Arrow Annotation** tool located on the Annotation Toolbar.

2. Click the left mouse button on the drawing area in the location where you want to place the north arrow annotation. The **North Arrow Manager** dialog is displayed:

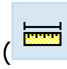


North Arrow Manager dialog

3. From this dialog you have the option to select the arrow style, orientation, text style and size of the north arrow.

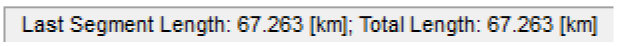
4. Press the **OK** button. The north arrow is placed on the drawing area in the location you have previously selected.

Measure Distances and Areas Tool

The **Measure Distances** tool () allows you to measure both distances and areas.

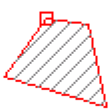
How to Measure Distances:

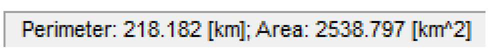
To measure distances or the length of objects, simply click on the start point where you wish to measure from and drag the cursor to the end point. A line will be formed showing what you are measuring and a length readout will appear on the status bar in the lower left corner of the window. The units of measurement are displayed under the drawing area, beside the measured value. Right click or press the Esc button to cancel or terminate the operation.



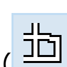
How to Measure Areas:

To measure areas, left-click with the mouse pointer on the drawing area, the location for one of the corners of the polygon area. Release the left mouse button. Drag the mouse pointer to the location of the next (adjacent) corner and click the left mouse button. Follow this procedure until you digitize all corners. To close the polygon, left-click with the mouse pointer inside the small box that marks the starting corner. A perimeter and area readout will appear on the status bar. The units of measurement are displayed under the drawing area, beside the measured value. Right click or press the Esc button to cancel or terminate the operation.





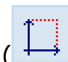
Map Import Tool

Use the **Map Import** tool () located on the Annotation Toolbar to import one or more base maps to be displayed in the drawing area.

- Importing DLG Base Maps
- Importing DXF Base Maps
- Importing LULC Base Maps

- [Importing MrSID Base Maps]
- [Importing Raster Image Base Maps]
- [Importing ArcView Shapefiles]
- [Importing Tile Maps]

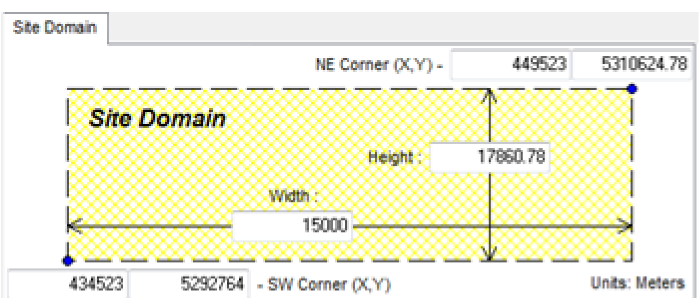
Site Domain Tool

The **Site Domain** tool () dialog allows you to define the domain extents of your modeling area. Initially, your domain extents are automatically setup but you can change the size of the initial domain at any time.

You have access to the **Site Domain** dialog by selecting **View | Site Domain...** from the menu or by clicking the **Site Domain** tool located on the Annotation Toolbar. In the **Site Domain** dialog, you can redefine the domain extents of your project area.

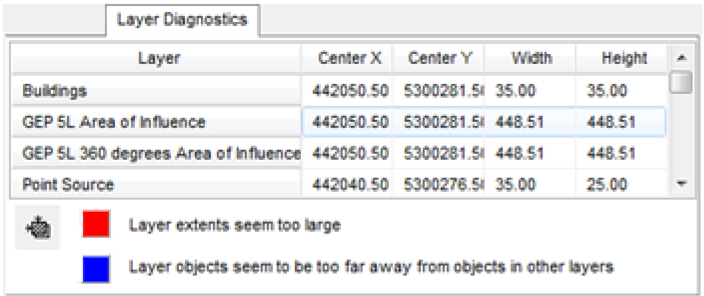
The **Site Domain** dialog contains the following options:

- **Site Domain tab:** Specify the X and Y coordinates for the southwest (SW) corner and the northeast (NE) corner of your domain and/or enter the width and height of your domain.

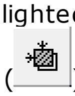


Site Domain tab

- **Layer Diagnostics tab:** This tab is available to help you identify potential problems with the locations of your modeling objects. All objects that are used in the modeling are assigned a layer in the domain. For example, a project containing sites and emission release points will contain two layers in the **Layer Diagnostics** tab.



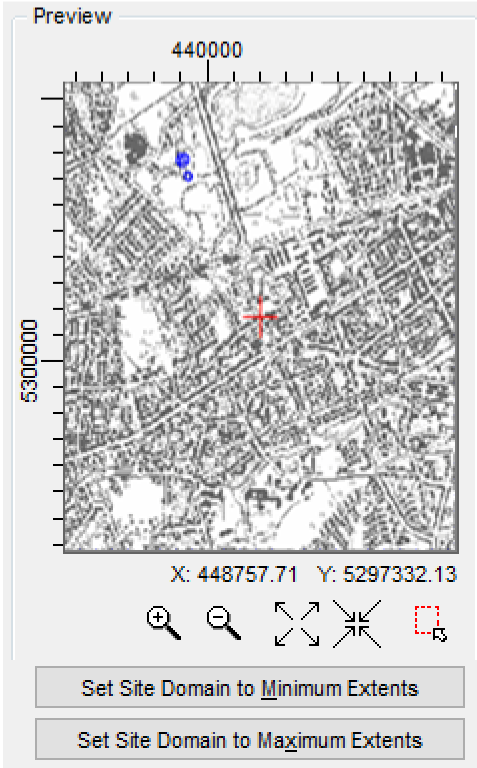
Layer Diagnostics tab

If one of the layers contains objects that are located far away from the rest of the modeling objects, it will be highlighted in blue in the table. This could result from one set of objects being entered in UTM coordinates while another set is entered in Cartesian coordinates. If the extents of a particular layer seems too large it will be highlighted in red. You can zoom to a layer by selecting a layer and then clicking on the **Zoom to Layer** () button.

- **Import Base Maps:** You can import base maps in various file formats. To import a base map, click the **Open** button to display a floating menu from which you can select the type of base map to import. The imported files are displayed in a list box. To display a base map, select the desired base map and check the box. The base map will be displayed in the Preview panel. At any time you may uncheck this box to turn off the display of the base map.

Import Base Maps panel

- **Preview Panel:** The **Preview** panel provides you with a preview of what your drawing area will look like.



Preview panel

If you have imported a base map and it is displayed in the **Preview** panel you can do one of the following:

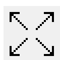
1. Use the outer extents of the imported base maps as your domain extents. To select this option, press the **Set Site Domain to Maximum Extents** button.

2. Use the **Choose Extents** tool located below the **Preview** area to graphically select your domain extents. Click the **Choose Extents** tool and, holding down the left mouse button, draw a rectangle on the preview area to define your domain area. Release the left mouse button. Note that the selected domain area is represented by a grey dotted box. The coordinates for the selected domain area are displayed in the **Site Domain** tab.

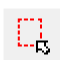
An explanation of all the tools found in the Preview panel is available below:

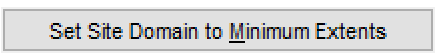
Zoom In tool: Allows you to zoom in on a selected region in the preview area.


Zoom Out tool: Allows you to zoom out in the preview area. Right-clicking displays a pop-up menu with additional zoom options.

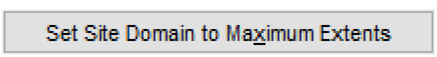

Expand tool: Increases the preview area.


Shrink tool: Decreases the preview area.


Choose Extents tool: Gives you the ability to graphically define your view. Holding down the left mouse button, draw a rectangle on the preview area to define your domain area extents. Release the left mouse button.

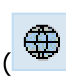


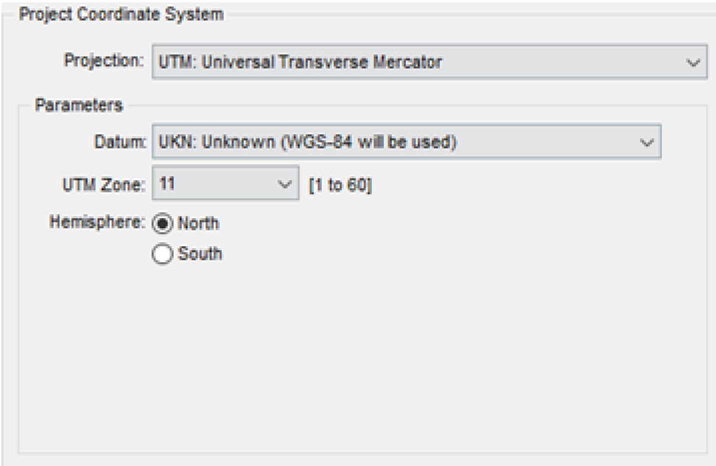
Set Site Domain to Minimum Extents: This button sets the site domain extents to encompass only the modeling objects (i.e. point sources, release points) in the open project.



Set Site Domain to Maximum Extents: This button sets the site domain extents to encompass all the modeling objects in the project as well as any base maps that are available.

Map Projection Tool

The **Project Coordinate System** tool () allows you to select the coordinate system you want to use for your project.



New Project Wizard - Project Coordinate System panel

How To Specify the Project Coordinate System:

1. Select the *Projection* for the project from the drop down list. You have the following options:

- **Unknown**
- **UTM: Universal Transverse Mercator**
- **Other: Local Cartesian Coordinate System**
- **ITM: Israeli Transverse Mercator (New Israeli Grid)**
- **OSNG: Ordnance Survey National Grid (Great Britain Grid)**

2. Complete the fields in the *Parameters* section, which displays the appropriate options for the Projection option that you have selected. The table below explains the options available:

Projection	Parameters Fields	Interface
Unknown	• N/A	N/A

UTM	• Datum • UTM Zone: possible values are from 1-60 • Hemisphere: N (North) or S (South)	View

Other	• Reference Point in Local coordinates and its corresponding coordinates in either UTM or Latitude/Longitude • Datum • UTM Zone and Hemisphere (if reference point given in UTM coordinates)	View

ITM	• Datum • NID: New Israeli Datum (GRS80 ellipsoid)	View

OSNG	• Datum • OSGB36: Ordnance Survey Great Britain 1936	View

ACEA	• Datum • ACEA: Albers Conical Equal Area (conical)	View

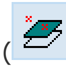
Click the **OK** button.

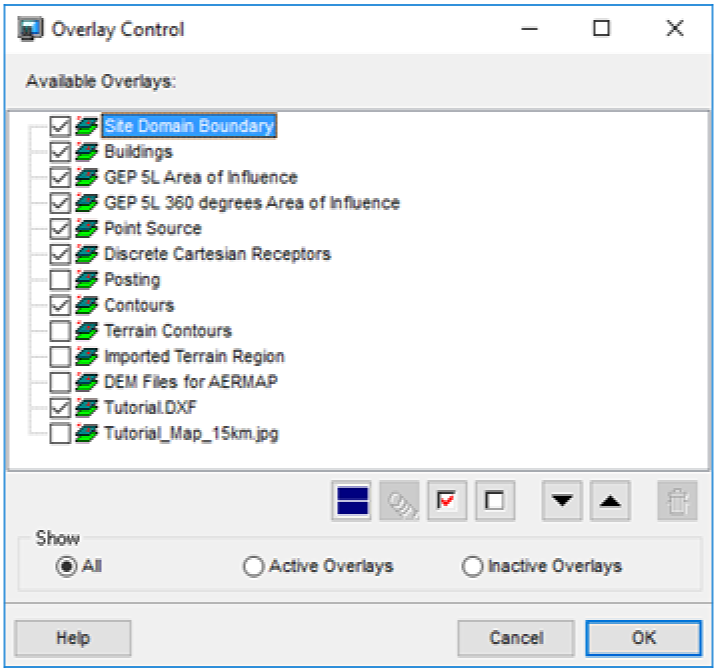
Overlay Control Tool

From time to time, during your project, you may wish to remove some objects/overlays from the drawing area. For example, you can view or print only the desired objects in your drawing area. From the **Overlay Control** dialog, you can specify which objects should be displayed or not on the drawing area.

You also have access to the options available in the **Overlay Control** dialog through the **Tree View - Overlays** tab.

How to Turn On or Off the Display of Overlays:

1. Press the **Overlay Control** tool () located on the [Annotation Toolbar] or select **View | Overlay Control** from the menu. The **Overlay Control** dialog is displayed.



Overlay Control dialog

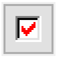
2. All the objects defined in your project are listed in this dialog. The checked/unchecked boxes indicates if the object is being displayed or not in the drawing area. You can easily turn on or off the display of these objects using the buttons located at the bottom of the dialog.

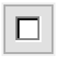
Button Guide

You can easily check or uncheck items from the list using the buttons located at the bottom of the dialog.

Select All: Use this button to select all of the layers listed.

Unselect All: Use this button to unselect all of the layers listed.

 **Check Selected:** Use this button to make all selected layers active.

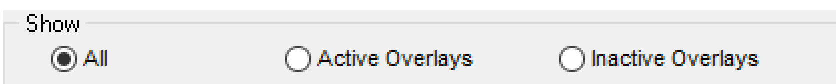
 **Uncheck Selected:** Use this button to make all selected layers inactive.

 **Move Down:** Use this button to move the selected layer down a level.

 **Move Up:** Use this button to move the selected layer up a level.

 **Delete Selected:** Use this button to delete the selected layers. You can only delete the layers containing base maps.

You can display the list of layers in different ways. You can show only the active overlays by selecting **Active Overlays**, or show only the inactive overlays by selecting **Inactive Overlays**.

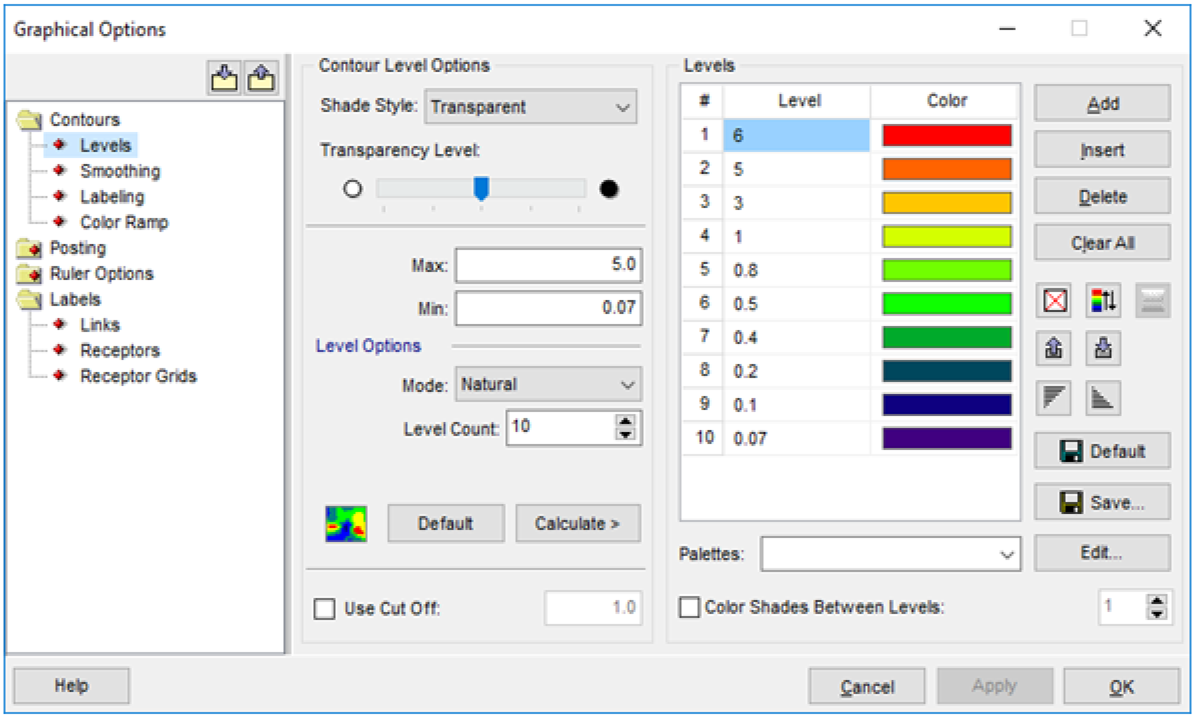


Graphical Options Tool

The **Graphical Options** tool () dialog contains options to enhance the appearance of your project. Within the Graphical Options dialog you can define options such as contour levels, shading, contour line thickness and color, and labels.

You have access to the **Graphical Options** dialog by select **Output | Graphical Options...** from the menu or by clicking on the **Output** button located on the Annotation Toolbar.

The **Graphical Options** dialog uses a two-pane view. The tree, located on the left side of the dialog, is used for navigation and item selection. Select an item (marked as) in the tree to display the available options on the right panel.



Graphical Options dialog

Impact Tool

The **Impact** tool () allows you to draw a radius of impact, outside of which there are no concentration/deposition values that are greater than a specified threshold value. This tool would be useful from a regulatory standpoint, as a "radius of impact" could be drawn on the contour plots for areas of concern such as a school.

How to Use the Impact Tool:

1. Press the **Impact** tool () located on the Annotation Toolbar.

2. Click the left mouse button on the drawing area in the location where you want the center of the radius to be located. The **Impact Tool Options** dialog is displayed.

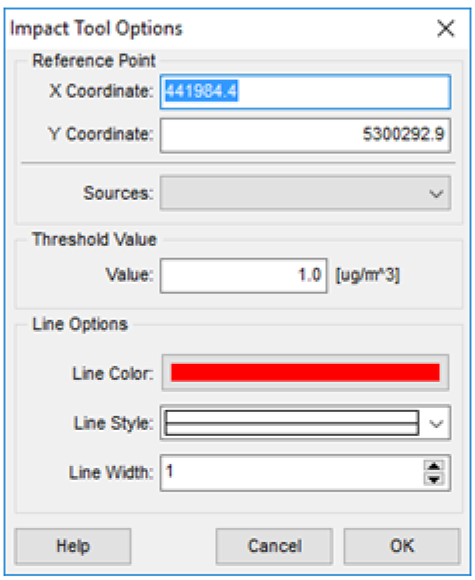
3. The X and Y coordinates of the reference point is displayed. You can change these values at any time if you desire. Alternatively, you may select from the **Sources** drop-down list a source to be

considered as the reference point. This drop-down list contains all the sources defined in your current project and also provides an **All Sources** option, which is the centroid all sources.

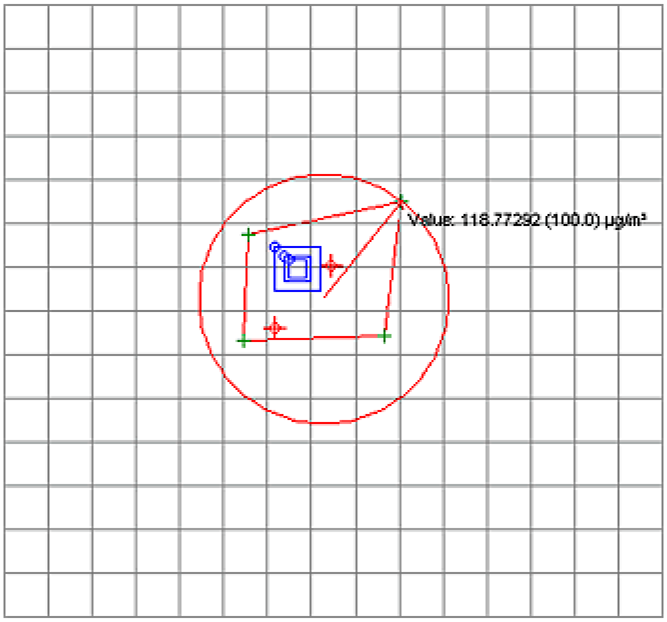
4. In the **Threshold Value** panel, specify a threshold value.

5. Choose the color for the line, the line style and width and press the **OK** button.

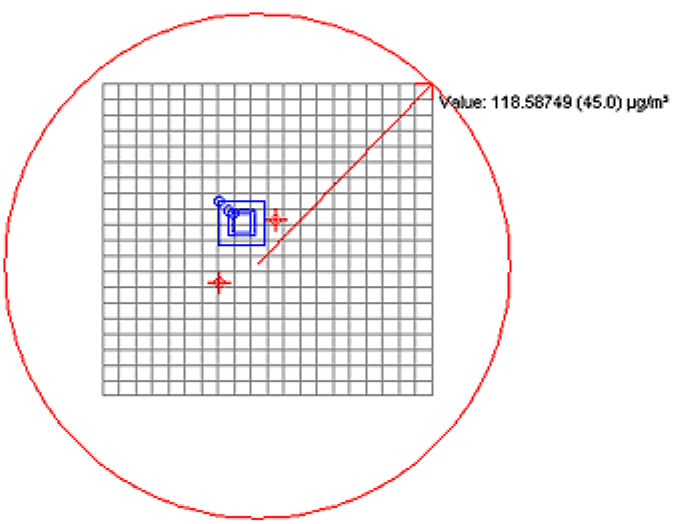
6. The impact radius will be placed on the drawing area for the location, threshold value and style you have selected.



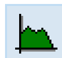
Impact Tool Options dialog



In certain cases the threshold value provided will result in a radius that extends to the farthest discrete receptor in your project. This does not necessarily mean in this case that any concentration/deposition values outside radius of impact is less than a specified threshold value as there are no receptors beyond this radius to detect values. We recommend you add receptors or extend your receptor grid, re-run your project and then use the impact tool to determine whether the radius of impact will change or remain the same.



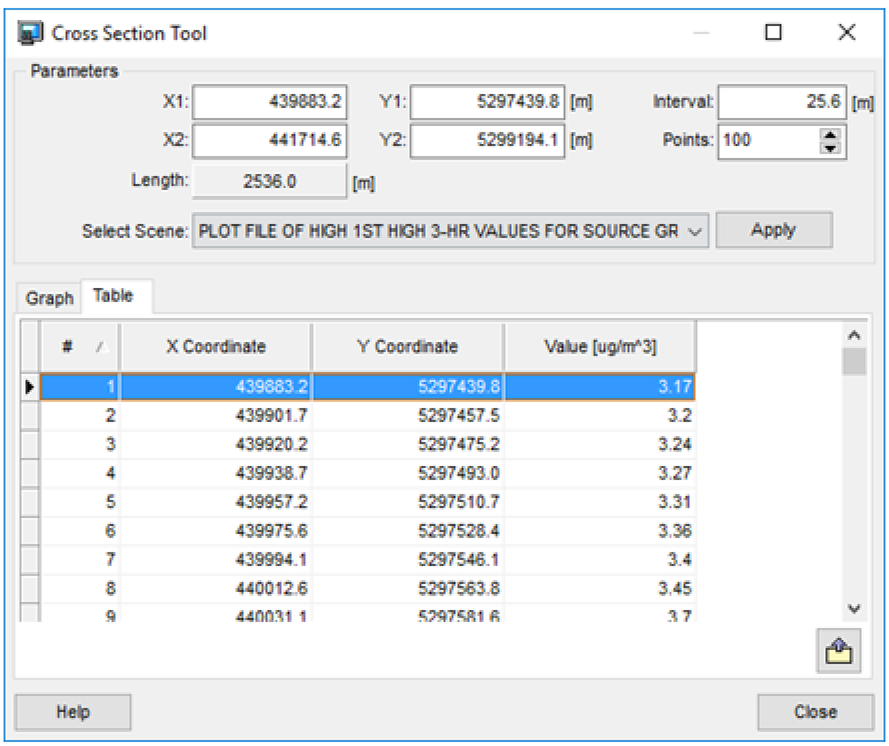
Cross Section Tool

The **Cross Section** tool () is used to graphically define a line segment within the modeling domain, in order to view available data for the points on that line. Available data includes terrain elevations and concentration results, and can be displayed in either a table or graph format.

How to Define a Line Segment:

1. Click the **Cross Section** tool located on the Annotation Toolbar.

2. Click the left mouse button on the drawing area at the location for one end of the cross section line. Drag the mouse pointer to the end point of the cross section line and click the left mouse button. The **Cross Section Tool** dialog is displayed.



Cross Section Tool dialog

3. From the **Select Scene** drop-down list, select the scene you want (e.g. model results, terrain elevations) and click the **Apply** button. The results will be displayed below as a table (**Table** tab) or graph (**Graph** tab).

4. In order to see different views of the data, in the **Parameters** section you can change the following parameters:

   - **X1, X2, Y1, Y2**: Change the length/location of the cross section by editing the X1, X2, Y1, and Y2 fields

   - **Interval (m)**: Change the interval of data points along the cross section.

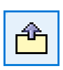
   - **Points**: Change the number of points along the cross section.

5. After each change, click the **Apply** button. The table or graph will display the updated data.

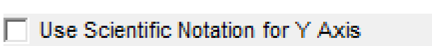
Button Guide

The following controls are available on the bottom of the dialog:

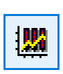
- If you are viewing the results as a **table**:

   **Export:** Allows you to export the displayed data to a comma-delimited file (.csv).

- If you are viewing the results as a **graph**:

 Use Scientific Notation for Y Axis: Check this box if you want the labels on the graph's Y axis to be written in scientific notation.

   **Options:** Opens the Graph Options dialog, which allows you to select various display options for your graph. You can also enhance the appearance of your graph by clicking the right mouse button anywhere within the graph and choosing any one of the available Graph Menu Options.

Print: Opens the Print dialog which allows you to select options to print your graph.

Export: Allows you to export the displayed data to a comma-delimited file (.csv).

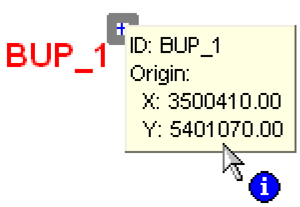
Identify Tool

The **Identify** tool ( ) allows you to quickly view the main properties for objects in the graphical display.

How to Identify Object Properties:

1. Select the **Identify** tool located on the Annotation Toolbar. The cursor changes to indicate that the tool has been selected.

2. Click any object in the graphical display. The main properties are displayed, as shown in the image below.



Accessory Information

Delete Objects Dialog

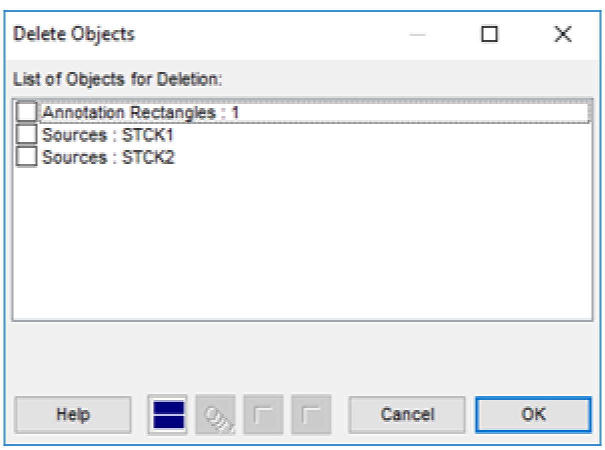
The **Delete Objects** dialog lists all the objects that have been selected for deletion. The selection of objects for deletion can be done using the following graphical tools:

 Point/Rectangular Delete Tool

Under the **Delete Objects** dialog, a list of objects selected for deletion will be displayed. A box can be found beside each object allowing you to check the box for any objects in the list that you want to delete.

The following options are also available:

- **Delete Objects Inside:** Select this option if you want the list to display all objects inside the selected area.
- **Delete Objects Outside:** Select this option if you want the list to identify all objects outside the selected area.



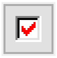
Delete Objects dialog

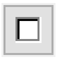
Button Guide

You can easily check or uncheck items from the list using the buttons located at the bottom of the dialog.

 **Select All:** Press this button to select all of the objects from the list.

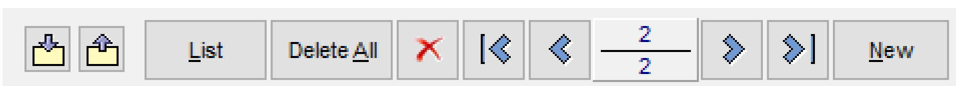
 **Unselect All:** Press this button to unselect all the objects from the list.

 **Check Selected:** Press this button to make all selected objects checked.

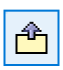
 **Uncheck Selected:** Press this button to make all selected objects unchecked.

Record Navigator

The **Record Navigator** is used in the windows where you can specify more than one record of the same type. The buttons contained in the **Record Navigator** will vary from window to window. The **Record Navigator** displayed here is an example of one configuration.



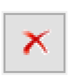
Buttons that can be found in the **Record Navigator** are:

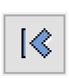
 **Export:** Click this button to be able to export the data displayed in the current window to a file.

 **Import:** Click this button to be able to import data to the current window from a file.

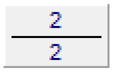
List: This opens a list of all the records already created for the type of information defined on the current window.

Delete All: Deletes all the records already defined in the current window.

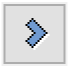
 **Delete:** Deletes the currently selected record.

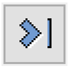
 **First Record:** Click this button to display the information for the first record.

 **Previous Record:** Click this button to display the information for the previous record.



Record Panel: The top panel will display the entry number for the current record. The bottom panel displays the total number of records already defined.


Next Record: Click this button to display the information for the next record.


Last Record: Click this button to display the information for the last record.

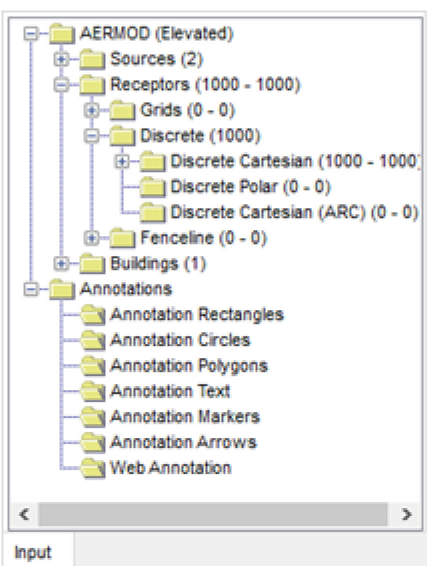
New: Allows you to create a new record.

Tree View

Input tab

The **Tree View - Input tab** contains shortcuts to input options necessary to run the model. By double-clicking on the folder icon of a specific input option, you have access to the dialog where data related to this option is specified.

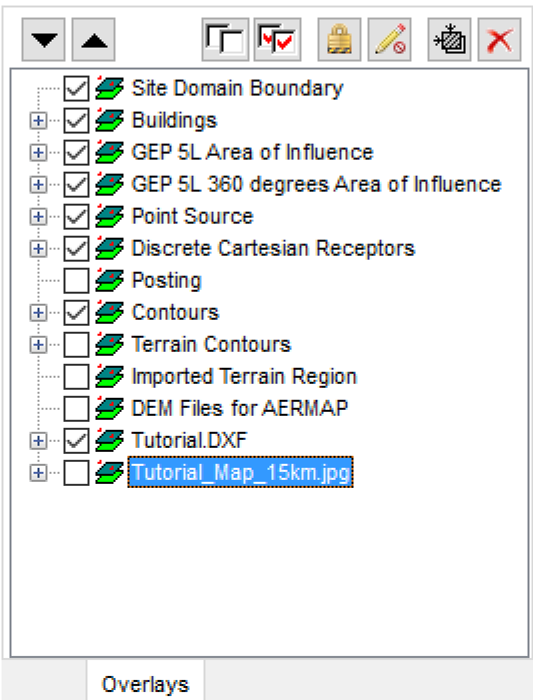
The **Input tab** gives you a general overview of the input options already specified for the current project.



Overlays tab

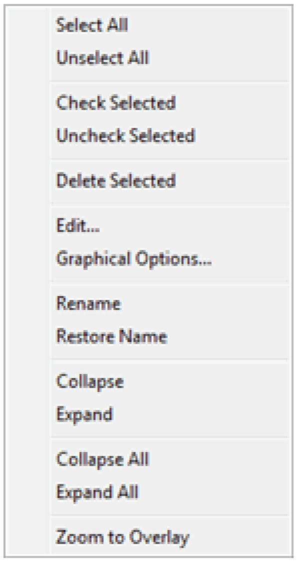
The **Tree View - Overlays tab** allows you to specify which objects should be displayed or not on the drawing area. From time to time, during your project, you may wish to remove some objects/overlays from the drawing area so that you can view or print only the desired objects in your drawing area.

> You also have access to this display option through the Overlay Control dialog.



All the objects defined in your project are listed in the **Tree View - Overlays tab**. The checked/unchecked boxes indicates if the object is being displayed or not in the drawing area. You can easily turn on or off the display of these objects by checking or unchecking these boxes.

You can also select an item from the list and right click with the mouse to display a floating menu as shown below.



- **Select All:** Selects all the objects in the list.
- **Unselect All:** Unselects all the objects in the list.
- **Check Selected:** Checks the selected objects.
- **Uncheck Selected:** Unchecks the selected objects.
- **Delete Selected:** Deletes the selected objects. You can only delete the layers containing base maps and gridded buildings.
- **Edit...:** Displays the dialog where data related to the selected object is specified. For example, selecting the **Edit** option for the point sources object will open the **Source Inputs** dialog. You also have access to these dialogs by double clicking on selected items in the Tree View list.
- **Graphical Options...:** Displays the Graphical Options dialog from where, depending on the objected selected, you can define options such as contour levels, shading, contour line thickness and color, and labels. You can also double click on objects in the **Tree View** list to display this dialog.
- **Rename:** Select this option to create a new name
- **Restore Name:** Select this option to restore to the original name
- **Collapse:** Select this option to collapse the selected layer and hide the marker, color or shading associated with that layer.
- **Expand:** Select this option to expand all layers and display the marker, color or shading associated with that layer.
- **Collapse All:** Select this option to collapse all layers.
- **Expand All:** Select this option to expand all layers.

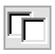
- **Zoom to Overlay:** Select this option to zoom to the selected overlay.

Button Guide

The following buttons are found in this tab:

Move Down: Use this button to move the selected layer down a level.

Move Up: Use this button to move the selected layer up a level.



Uncheck All: Use this button to remove check marks from all overlays. This will display an empty view.

Check All: Use this button to check all overlays. All overlays will become visible.

Disable Selection: Click this button to prevent the objects in this layer from being selected. This feature is used to disallow base maps to be selected when selecting a source, building, etc.

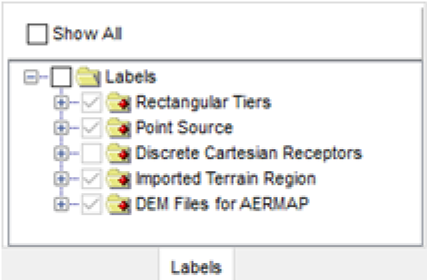
Disable Graphical Editing: Click this button to prevent the objects in that layer layer from being moved, resized or deleted.

Zoom to Overlay: Select this option to zoom to the selected overlay.

Delete Selected: Use this button to delete the selected layers. You can only delete the layers containing base maps.

Labels tab

The **Tree View - Labels tab** allows you to change the labels for the objects displayed in the drawing area. A list of all the sources, buildings, grids, and other objects you have defined for your project, is displayed in the **Tree View - Labels tab**.



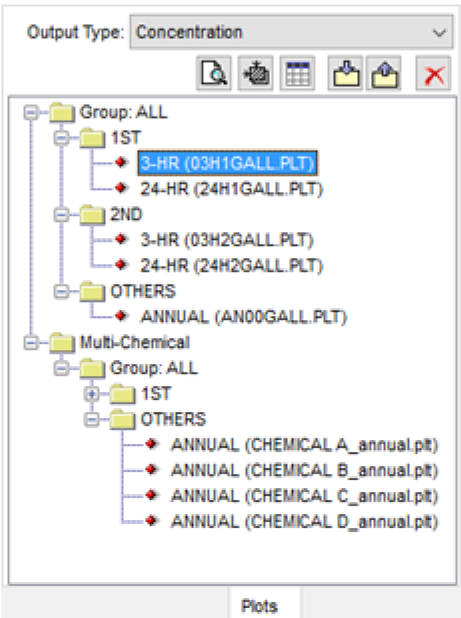
There are many ways to display or change the appearance of the labels:

- **Show All:** Check this box to display all available labels. If this box is NOT checked, the number of labels displayed in this tab will be capped at 1000.
- Check or uncheck the boxes next to object names to display or hide the labels. If, for example, you wish to hide the label for a particular source, simply uncheck the box next to that source. If you wish to hide all the labels for your point sources, uncheck the Point Sources box.
- Double click on any item in the list to open the Graphical Options dialog from where you can make the changes to the label for the selected object.
- Select an item from the list and right click with the mouse to display a floating menu as shown below.

- **Show Labels:** Displays the label for the selected object
- **Hide labels:** Hides the label for the selected object
- **Select All:** Selects all the objects for that object type. For example, if you have selected a building and click on the Select All option in the floating menu, all the buildings in the list will be selected.
- **Allow Labels to Overlap:** Select this option if you want your labels to overlap. A check indicates this option is active.
- **Label Options...:** Displays the Labels page of the **Graphical Options** dialog from where you can make changes to the labels.

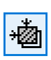
Plots tab

The **Tree View - Plots tab** lists all available plot files generated for the current run. This tab is only available after successfully running the model. The plot files generated are grouped by source group and high value.



Results displayed in the drawing area are for the plotfile currently highlighted in the **Plots tab**. As you select a different output plotfile, the drawing area is refreshed and contour results are displayed for the new selected output option.

 Click on this button to display the selected plotfile in text editor.

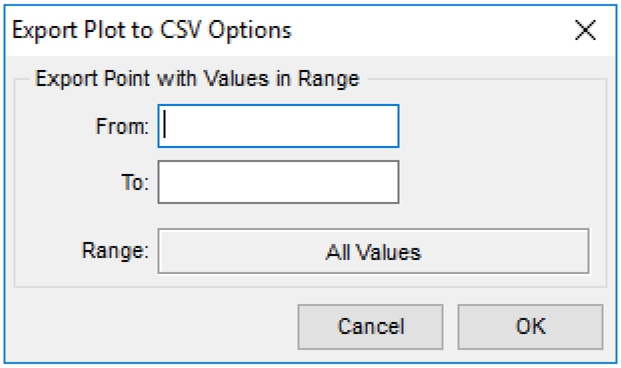
 Click on this button to zoom to the extents of the currently selected plotfile.

 Click on this button to display the selected plotfile in the Plot File Grid Viewer.

 Click on this button to import a plotfile into the project.

 Click to export concentration results. Using the presented menu you can **Export All** or **Export Range** of concentrations to a CSV file.

If you choose to **Export Range**, the following **Export Plot to CSV Options** dialog will allow you to specify the range bounds:



 Click on this button to delete the selected plotfile. You may select multiple plotfiles from the tab by pressing the **Ctrl** key while selecting the files.

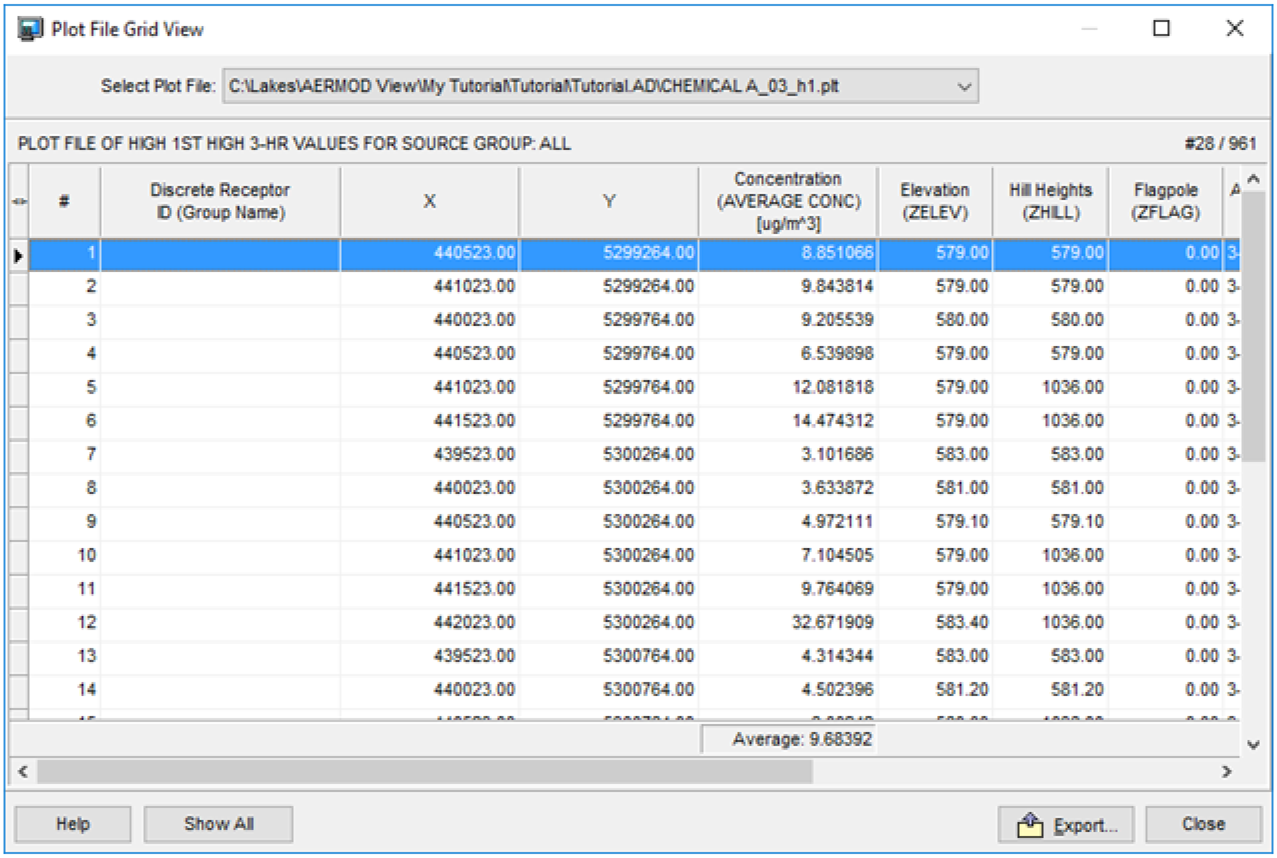
> The selected plotfiles will be permanently deleted from your project, not just from the **Plots** tab unless they are imported plotfile.

You can also right click with the mouse button on any of the plotfile in the tree view to display the following floating menu -

- **Delete Results:** Allows you to delete the selected plot file(s).
- **View Results...:** Displays the selected AERMOD View plotfile for the current contour plot either as text in a text editor or in grid format in the Plot File Grid Viewer.
- **Save As...:** Allows you to save the selected plotfile under a different name and/or location.
- **Select All in Group:** Selects all the plotfile located within the folder.
- **Select All:** Selects all plotfiles.
- **Unselect All:** Unselects all plotfiles.

Plot File Grid View

The **Plot File Grid View** is displayed by clicking on the button in the Tree View - Plots tab. Here you can view your plot file in grid format.

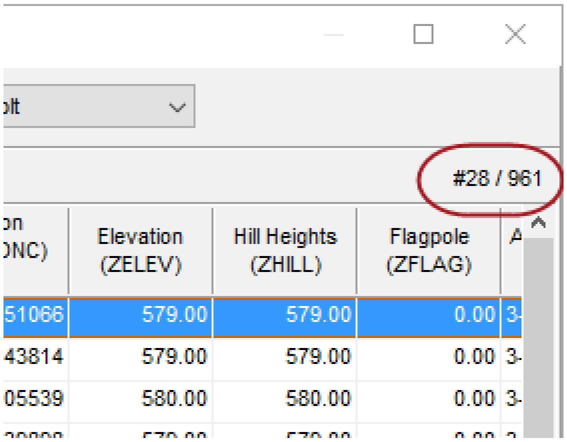


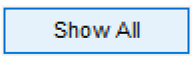
Plot File Grid View

At the top of this dialog, the name and location of the plot file is displayed. From this drop down list you can select which plot file you wish to view in grid format. Below this, a description of the plot file can be seen along with the total number of receptors.

The information from the plot file is displayed in an easy to read grid format. The information can be sorted by column, for example you can sort the data by highest to lowest concentration (or vice versa) by clicking on the column header. You can also drag and drop the columns to change the order in which they are displayed. The average concentration is also displayed on the bottom panel, under the **Concentration** column.

When you open the **Plot File Grid View** through the **Tree View - Plots tab**, results are displayed for all the receptors in that plot file. If you wish to view results for a specific set of receptors only, you may do this by using the any of the **Select** tools (Point/Rectangular Select Tool, Circular Select Tool, Polygonal Select Tool) in the Annotation Toolbar. Simply select the receptors, right click and from the menu select **Post File Grid View**. The dialog will open with only the selected receptor results displayed. Note in the example below, only 88 of the 1402 receptors are displayed:





Click on this button to display results for all receptors in the plot file

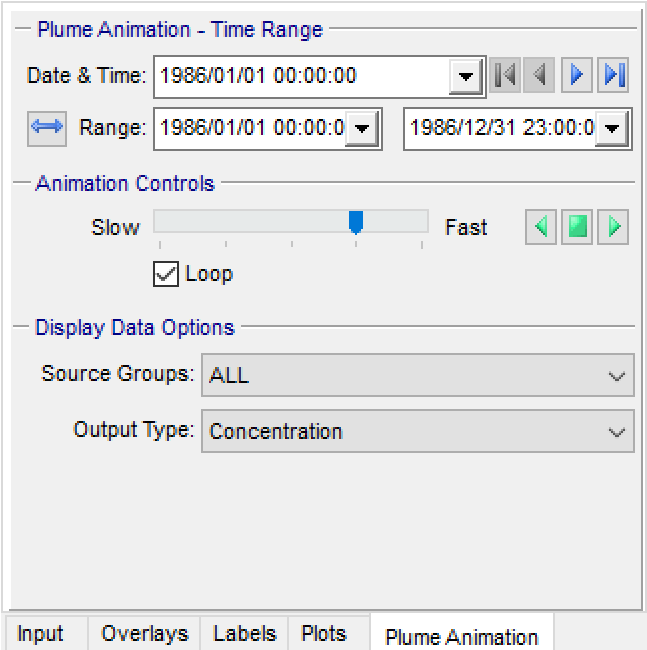
Click on this button to export the plot file data to **csv** format.

Plume Animiation Tab

The **Plume Animation** tab allows you to view the individual plumes for select time periods, species, and sources. You can view the plume for any particular time or play an animation that will show the plume for the selected parameter and source over a period of time.

In order to access this tab and view plume animations for your project, you need to first enable the feature in **Output Pathway**. After AERMOD has been run successfully, the information regarding specific source groups will be extracted from the AERMOD model-generated **POSTFILES** in order to generate the corresponding plume animations in the map interface.

See Also: Output Pathway - Plume Animation



Tree View - Plume Animation tab

Plume Animation - Time Range

The **Date & Time** field indicates the date and hour, for which the concentration/deposition data is currently being displayed in the drawing area. You can select a specific **Date** by clicking on the down arrow in this field and selecting a date from the calendar.

 - Move to the beginning of the time period

 - Move back one hour.

 - Move forward one hour.

 - Move to the end of the time period.

 - Set **Range** to full.

> The averaging period is fixed to 1 hour as the **Plume Animation** feature uses the 1-hour average POSTFILE output to produce hourly contours.

Animation Controls

Use the slide control to set the animation speed.

- Click this button to play the animation in reverse.

- Click this button to stop the animation.

- Click this button to play the animation.

Check the **Loop** box to have the animation automatically restart once it reaches the end of the designated time period.

Project Setup and Management

This section will walk you through the various features of AERMOD View and show you how they are used to set-up, run, and display the results for your model. While this section provides an explanation of the AERMOD View functionality, if you wish to get started with using AERMOD View right away, please refer to the AERMET View tutorial and the AERMOD View tutorial in the subsequent sections of the manual. In this section you can find information on the following topics:

- Project Management
- New Project Wizard
- Working with Base Maps
- Tools & Utilities
- Graphical Options
- Printing
- Preferences

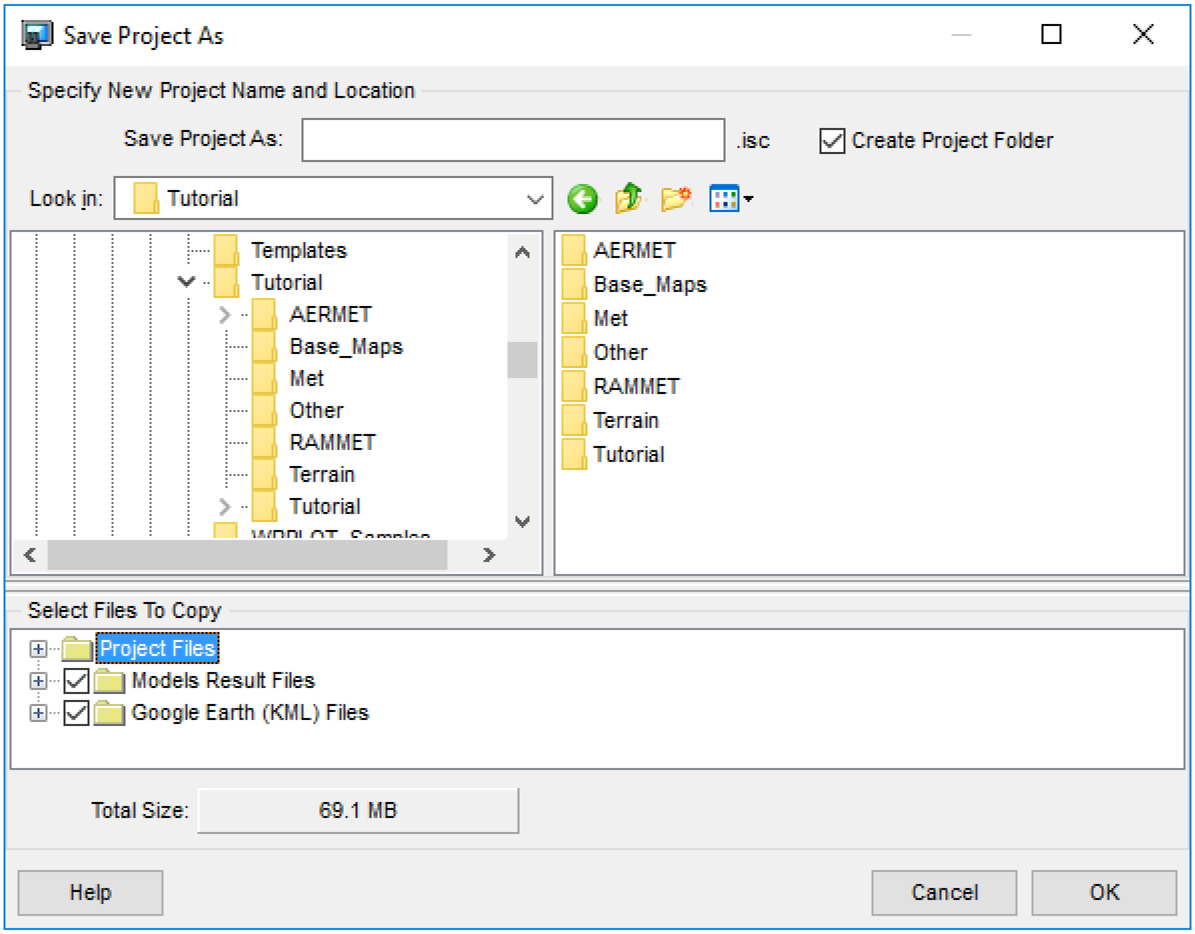
Project Management

The **Project Management** section describes the options available for project backup and repair.

- Save Project As
- Backup Options
- Project Repair

Save Project As

The **Save Project As** dialog is displayed when you select the menu option **File | Save Project As**. The **Save Project As** option allows you to create different scenarios within the same project. Using this option will allow you to save your project files under a new project name so that you can change some of your project parameters rather than starting from the beginning.



Save Project As dialog

How to Use the Save Project As Option:

1. Under the **Save Project As** field, specify the name for your new project and select the location where you want your project to be located. You cannot create more than one project in the same location.

2. Check the **Create Project Folder** box, if you want AERMOD View to automatically create a folder with the same name as you project to save your new project files.

3. Under the **Select Files to Copy** list, make sure to check the boxes for all the files that you wish to be saved with the new project.

4. Click on the **OK** button once you have finished.

5. The new AERMOD View project will be automatically opened and ready for your changes.

Project Files List

The following table lists what files are saved in case they exist in the original project:

Module	File Type	Description
AERMOD		
	[ProjectName].isc	Project File
	[ProjectName].ini	INI File
	[ProjectName].INP	ISCST3 Input File*
	[ProjectName].ADI	AERMOD Input File
	[ProjectName].PIN	ISC-PRIME Input File*
	[ProjectName].dat	Project Database Folder
	[ProjectName].IS	Folder to Save ISCST3 output files*
	[ProjectName].PR	Folder to Save ISC-PRIME output files*
	[ProjectName].AD	Folder to Save AERMOD output files
BPIP		
	[ProjectName].bpi	BPIP Input File
	[ProjectName].pro	BPIP Output File
	[ProjectName].sup	BPIP Summary File
AERMAP		
	[ProjectName].api	AERMAP Input File
	[ProjectName].ast	AERMAP Summary Output File
	[ProjectName].ROU	AERMAP Receptor Output File

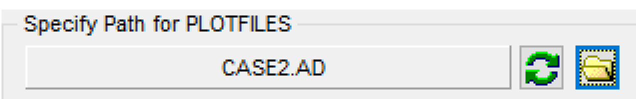
Module	File Type	Description
	[ProjectName].SOU	AERMAP Source Output File

> * If you are saving your project as ISCST3 or ISC-PRIME, the [ProjectName] must not contain spaces.

> In the Output Pathway, any paths specified in the original project will be reverted to the default path in the new 'Save Project As' project.

For example, suppose that in your original project you have chosen to save your output plotfiles in a folder called **Plotfiles_OU_1**. The specified path for these files would be C:\Lakes\AERMODView \plotfiles_OU_1 After using the **Save Project As** function, the path to save your plotfiles will be changed to the default one. That is, if the name of the 'Save Project As' project is **Case2** and you were using the AERMOD model, the default path would be CASE2.AD. For ISCST3 it would be CASE2.ISC and for ISC-PRIME it would be CASE2.PR.

Specify Pathway - Original Project

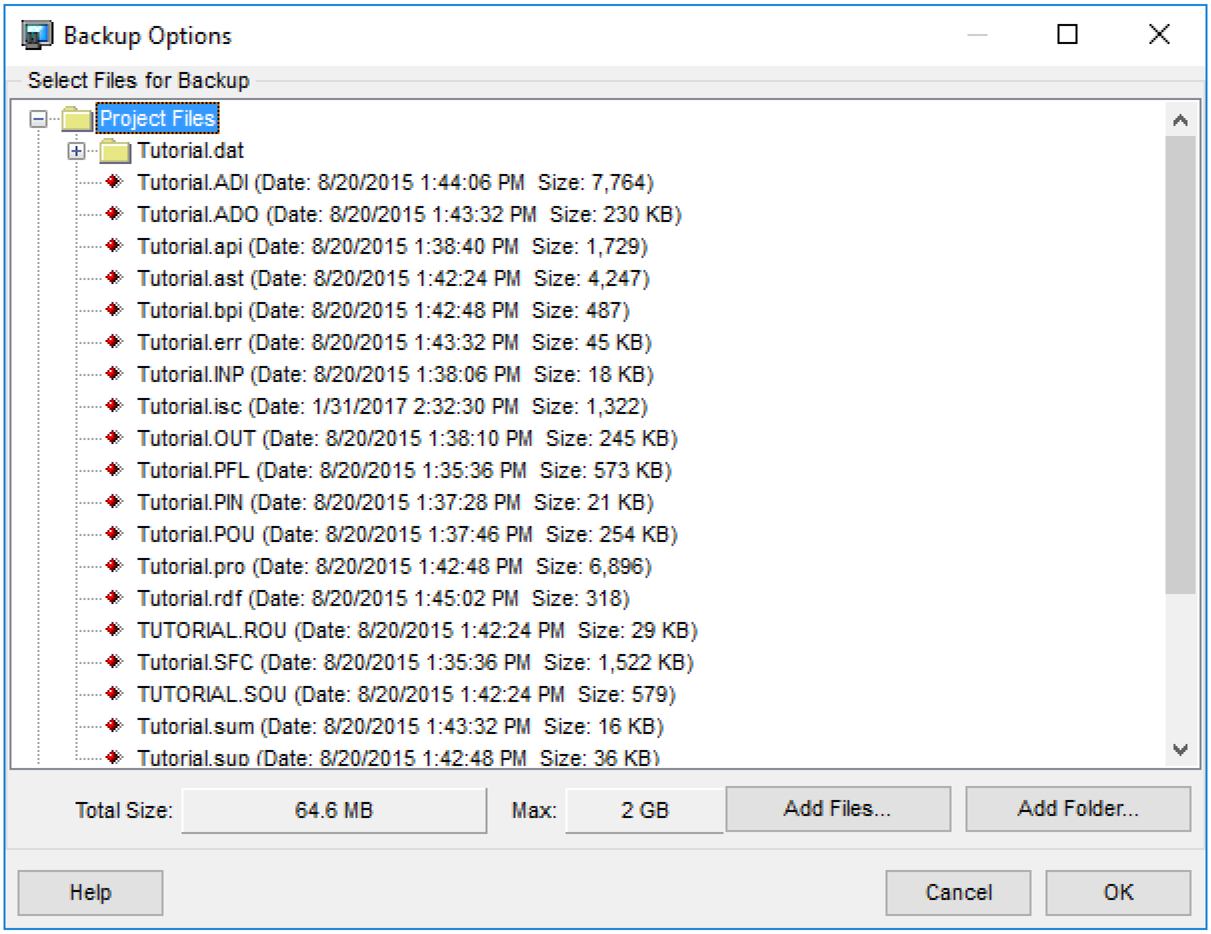


Specify Pathway - Using 'Save Project As' Option

> When using the **Save Project As** function, the output files (e.g. main output file, plotfiles, postfiles, etc.) from your original project will not be saved in the new project folder. Once you have modified the new project, you will need to run the project again.

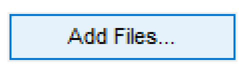
Backup Options

The **Backup Options** dialog is displayed when you are saving your project to a zip file by selecting **File | Backup | Save to Zip...** from the menu.



Backup Options dialog

The **Select Files for Backup** dialog displays the files that are automatically added to your zip file by default. These are the basic files containing the essential project information.

If you wish to add any additional files to the zip file, click on the 

button. The **Choose Additional Files for Backup** dialog opens, displaying your main project folder contents. You may select files from here, or navigate to a different location to specify the files you wish to add to the zip file. Click on the **Open** button.

If you wish to add any additional folders to the zip file, click the 

button. A browsing dialog opens that will allow you to navigate to and select the folder you wish to include.

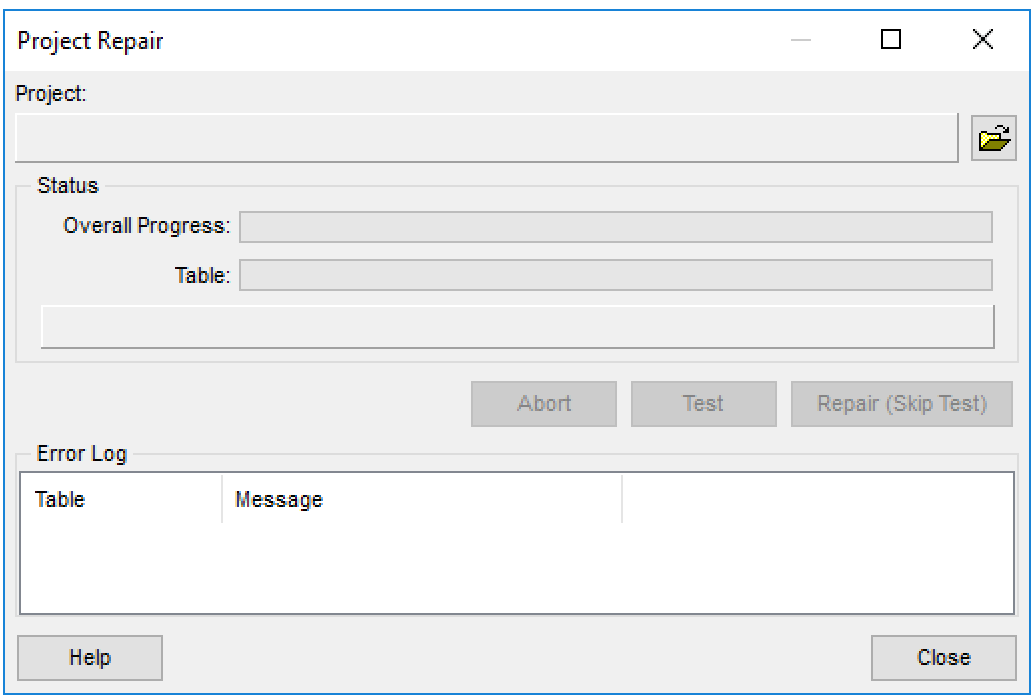
> You cannot add folders to the zip file; however, you can go into any folder and select the files within that folder to add to the zip file.

At the bottom of the **Select Files for Backup** dialog is the **Total Size** panel which displays the size of your zip file in MB. You cannot save more than 2GB in your zip file.

> You cannot save more than 2GB in your zip file.

Project Repair

If your system crashes or the power goes off while you are working on your AERMOD View project, the database files may become corrupted. If you try to open your project and AERMOD View gives you a message saying that it cannot open the project because it may be corrupt, then you should use the project repair option. You have access to the **Project Repair** dialog by selecting **File | Repair Project...** from the menu.



Project Repair dialog

How to Use Project Repair:

1. Use the  button to specify the AERMOD View project that needs to be repaired (*.irp). If the damaged project is open, you must close it before running the Project Repair tool.

2. Once you have specified the file of concern, click the Test button. The **Status** bar will display the overall progress of the scan. When the scan is complete, a message will be displayed indicating whether or not any problems were identified. The **Error Log** will contain a list of any errors detected.

3. If there were errors in the database, click on the Repair (Skip Test) button. In order to prevent data loss, make sure you have a backup copy of the file before you perform this function.

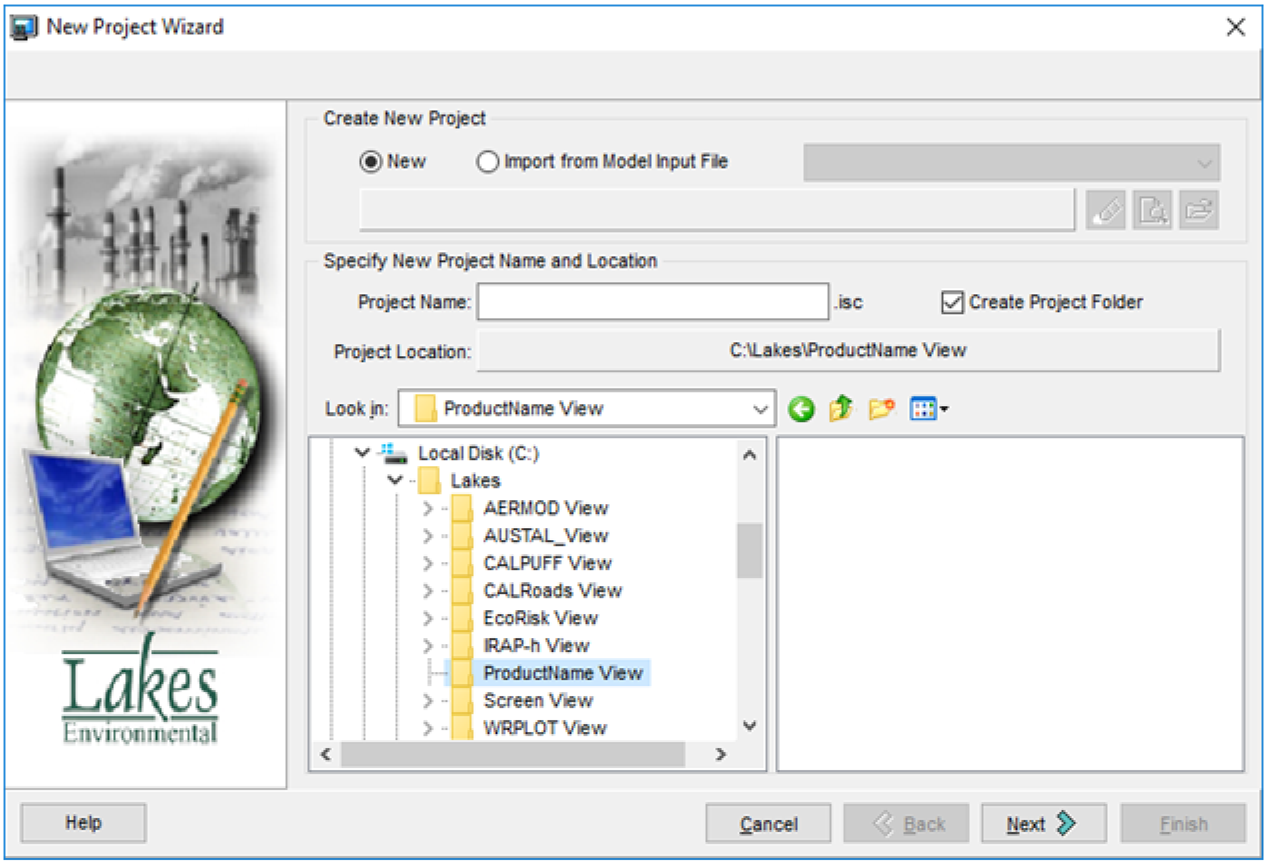
4. Click on the Test button again to ensure that all errors were repaired successfully. When you are finished, click **Close**.

> In order to run the **Project Repair** tool, you must ensure that the project of concern is not currently open.

New Project Wizard

The **New Project Wizard** takes you step-by-step through all the options available for creating a new AERMOD View project. You have access to the **New Project Wizard** by selecting **File | New Project...** from the menu or by clicking on the **New** button on the Menu Toolbar.

This wizard allows you to easily specify information such as your project file name, location and coordinate system for the new AERMOD View project file you want to create.



New Project Wizard

There are several panels in the **New Project Wizard**. See the following sections for a description of each of the options found within these panels:

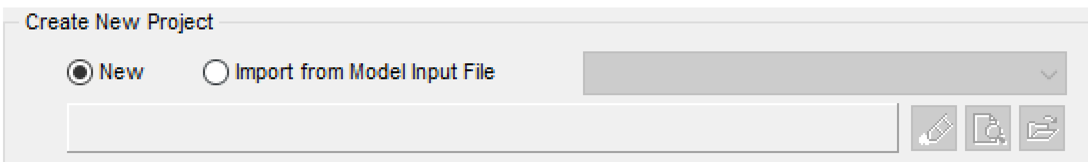
- Create New Project
- Project Coordinate System
- Project Reference Point

New Project Wizard - Create New Project

The **Create New Project** panel allows you to specify whether the new project will be created from an existing model input file or from the settings you provide.

How To Create a New Project:

1. Specify whether the project will be created from new or by importing from an existing model input file.



New Project Wizard - Create New Project panel

- **New:** To create a new project, leave this option selected.
- **Import from Model Input File:** Select this option if you want to create the new project by importing the parameters from an existing model input file. If this option is selected, you will need to select a model input file type from the drop down box.

> If you are creating a new project by importing a model input file, you will be able to configure more options in the AERMOD Input File Options screen later on.

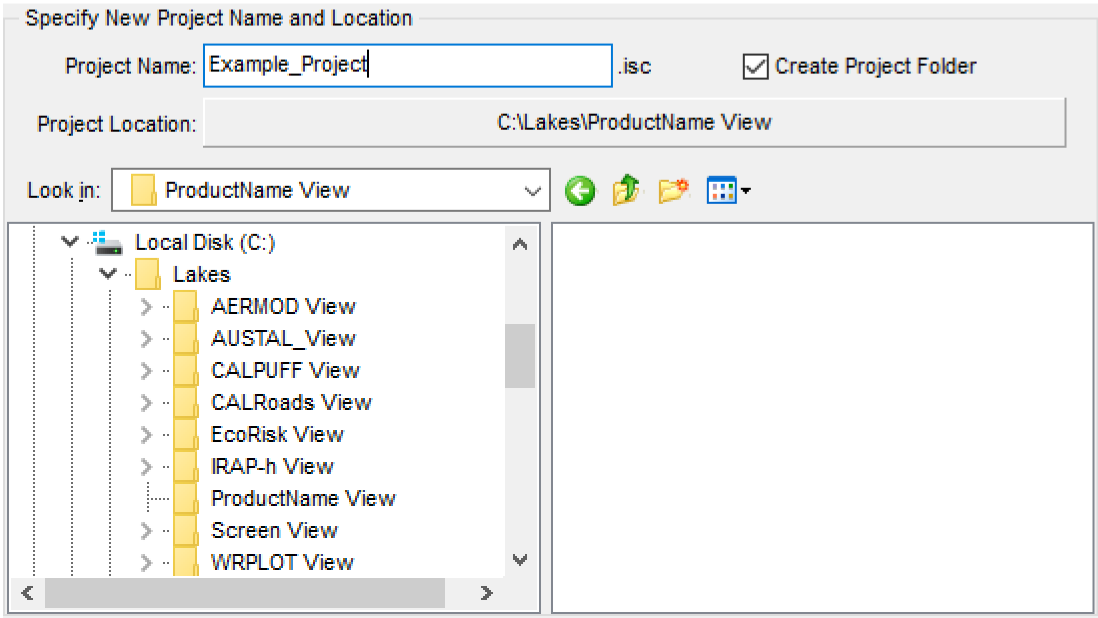
> You can also create a new project by importing ISCST3 or ISC-PRIME input files.

2. After selecting the model input file type click the **Select Folder** button () and specify the location and input file to import.

3. Specify a **Project Name** and select the path to save your project in. You cannot have more than one project in the same location. If you check the **Create Project Folder** box, your new project will be automatically created inside a folder with the same name as the project file name you have specified.

> Every Lakes product has a separate folder under C:\Lakes\. It is advised that you create new projects in the folder for the appropriate product.

> If you will be working with ISCST3 or ISC-PRIME, the project name cannot contain spaces. The project path (e.g. folder name) can contain spaces.

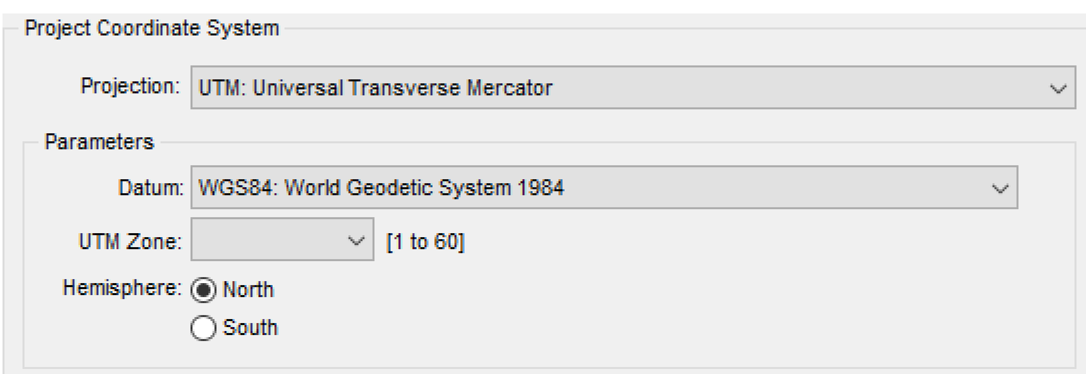


New Project Wizard - Specify New Project Name and Location

4. Click the **Next** button.

New Project Wizard - Project Coordinate System

The **Project Coordinate System** panel allows you to select the coordinate system you want to use for your project.



New Project Wizard - Project Coordinate System panel

How To Specify the Project Coordinate System:

1. Select the *Projection* for the project from the drop down list. You have the following options:
   - **Unknown**
   - **UTM: Universal Transverse Mercator**
   - **Other: Local Cartesian Coordinate System**
   - **ITM: Israeli Transverse Mercator (New Israeli Grid)**
   - **OSNG: Ordnance Survey National Grid (Great Britain Grid)**

2. Complete the fields in the **Parameters** section, which displays the appropriate options for the **Projection** option that you have selected. The table below explains the options available:

Projection	Parameters Fields	Interface
Unknown	• N/A	N/A
UTM	• Datum • UTM Zone: possible values are from 1-60 • Hemisphere: N (North) or S (South)	View
Other	• Reference Point in Local coordinates and its corresponding coordinates in either UTM or Latitude/Longitude • Datum • UTM Zone and Hemisphere (if reference point given in UTM coordinates)	View
ITM	• Datum	View

	• NID: New Israeli Datum (GRS80 ellipsoid)	

OSNG	• Datum • OSGB36: Ordnance Survey Great Britain 1936	View

3. If you are creating an entirely new project , click the **Next** button. If you are creating a new project from an existing modeling input file, click the **Finish** button.

Datum Options

A datum must be defined based on the options available for the map projection you selected for your project. The following options are available:

Projection	Parameter Fields
Unknown	• N/A
UTM	• UKN: Unknown (WGS-84 will be used) • ED50: European 1950 • ETRS89: European Terrestrial Reference System 1989 • GDA94: Geocentric Datum of Australia 1994 • NAD27: North American Datum 1927 • NAD83:North American Datum 1983 • OHD: Old Hawaiian Datum • PRD: Puerto Rico Datum • WGS72: World Geodetic System 1972 • WGS84: World Geodetic System 1984
Other	• ETRS89: European Terrestrial Reference System 1989 • NAD27: North American Datum 1927

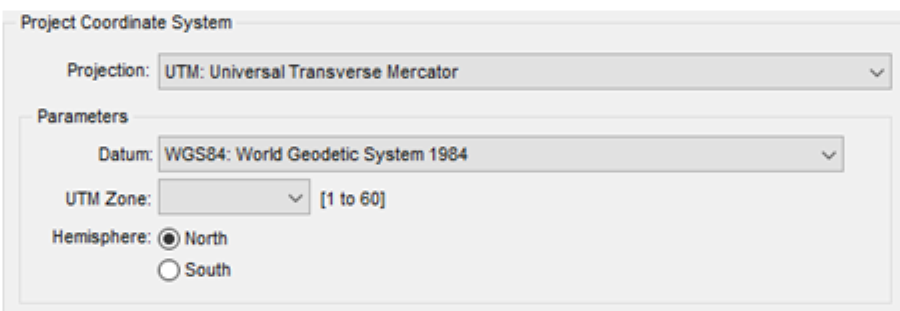
	• NAD83: North American Datum 1983 • OHD: Old Hawaiian Datum • PRD: Puerto Rico Datum • WGS72: World Geodetic System 1972 • WGS84: World Geodetic System 1984 • ED50: European 1950

ITM	• NID: New Israeli Datum (GRS80 ellipsoid)

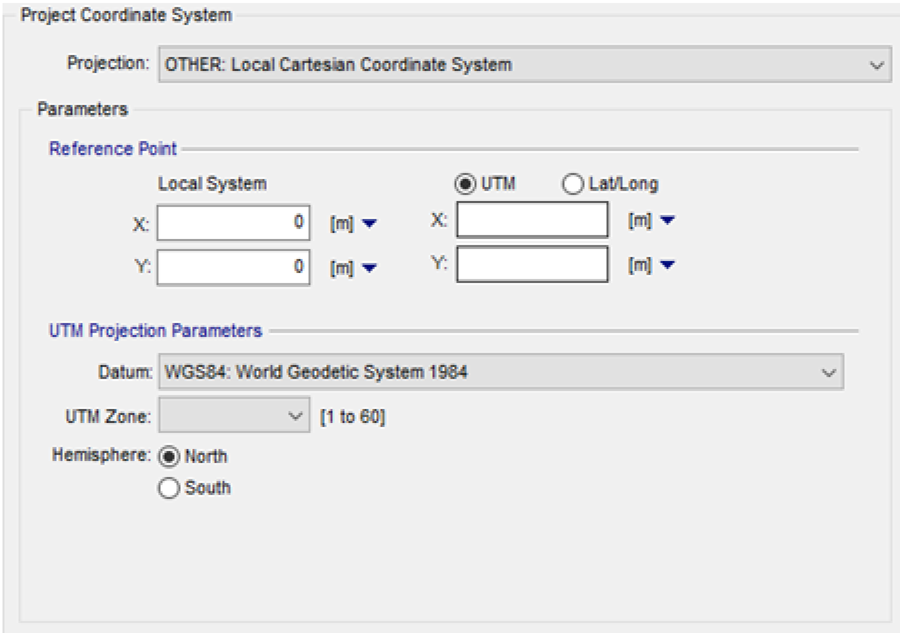
OSNG	• OSGB36: Ordnance Survey Great Britain 1936

Coordinate System Options

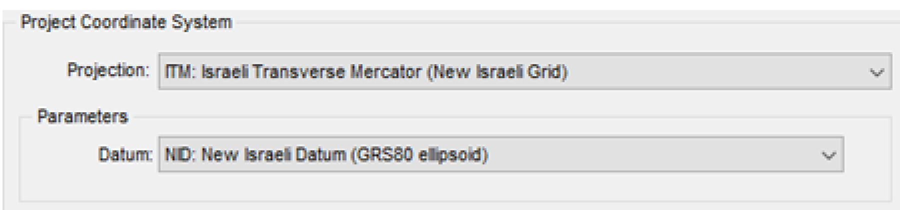
Coordinate System Parameters - Universal Transverse Mercator



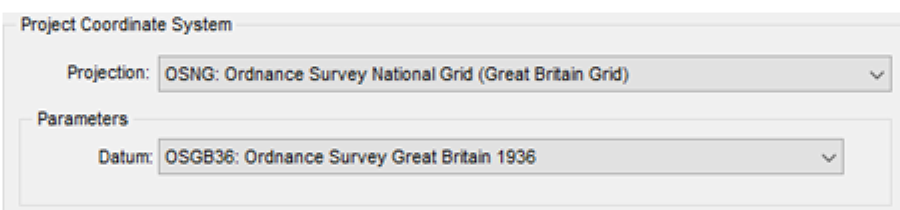
Coordinate System Parameters - Local Cartesian Coordinate System



Coordinate System Parameters - Israeli Transverse Mercator Coordinate System

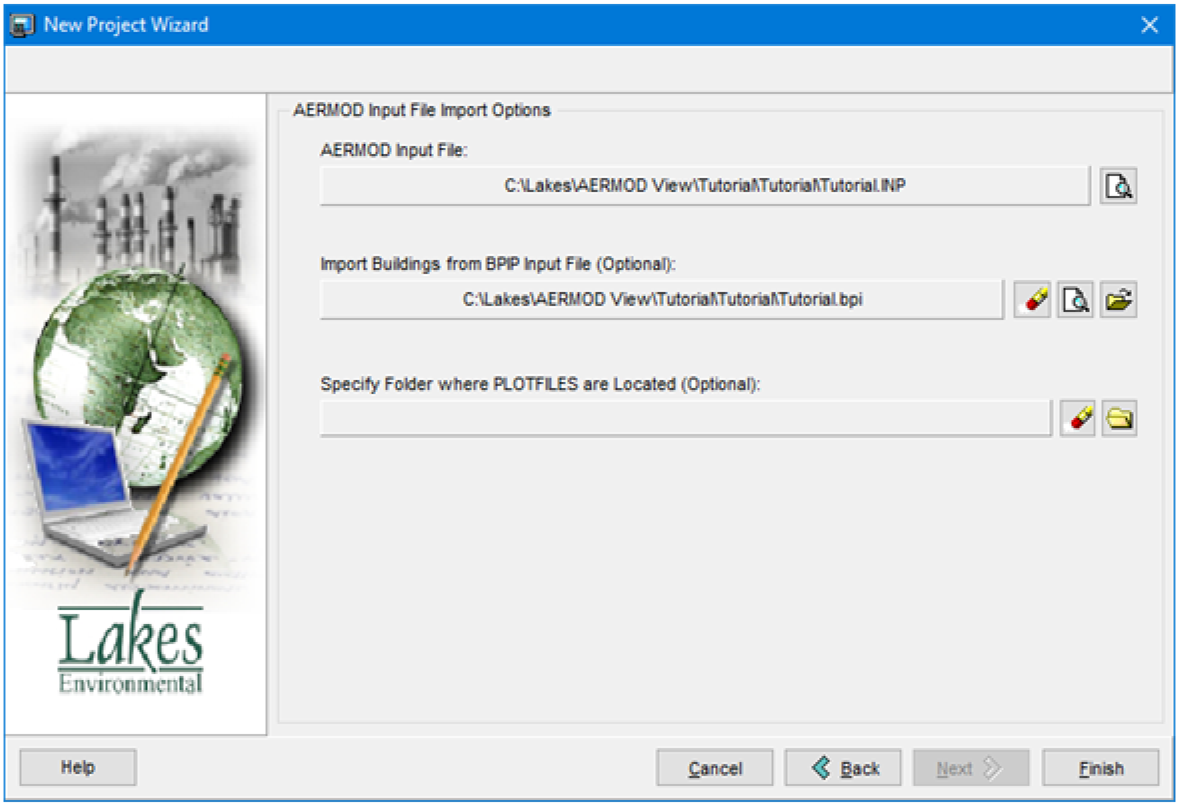


Coordinate System Parameters - Ordnance Survey National Grid Coordinate System



New Project Wizard - Input File Import Options

The **Input File Import Options** screen allows you to specify the location of the model input file, the BPIP input file, and the PLOT files on which you wish to base your new project.



New Project Wizard - AERMOD Input File Import Options

- **AERMOD Input File:** Displays the path to the selected AERMOD input file. You can only view the file contents in this screen. If you wish to select a different file, use the Back button to go back to the Create New Project screen.

> If importing ISCST3 or ISC-PRIME input files the caption will change accordingly, but the same options will be available.

- **Import Buildings from BPIP Input File (Optional):** Select the building profile input file.

- **Specify Folder where PLOTFILES are Located (Optional):** Navigate to and select where the PLOTFILES for the imported model are located.

- Click to preview the contents of the file.

 - Click to remove the currently selected file.

 - Click to navigate to a new file.

 - Click to navigate to a folder.

New Project Wizard - Project Reference Point

The **Project Reference Point** panel allows you to specify a reference point within your modeling area.

> The **Project Reference Point** panel only displays if you are creating an entirely new project.

How To Specify the Project Coordinate System:

1. Enter the coordinates of the **Reference Point**. The options available will depend on which coordinate system you are using. Reference the following table to determine which coordinates you will need to enter.

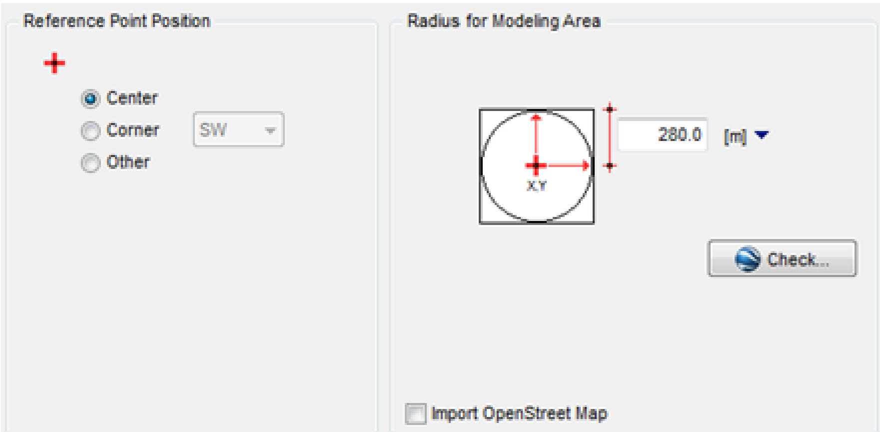
Projection	Parameter Field
Unknown	• Click to see the interface
UTM	• Specify X,Y in either UTM or Lat/Long You cannot enter negative numbers for Lat/Long. Use the radio buttons to the right of the Lat/Long fields to select hemisphere. • Click to see the interface
Other	• N/A - displayed only; set in previous panel • Click to see the interface

ITM	• Specify X, Y in either ITM or Lat/Long You cannot enter negative numbers for Lat/Long. Use the radio buttons to the right of the Lat/Long fields to select hemisphere. • Click to see the interface

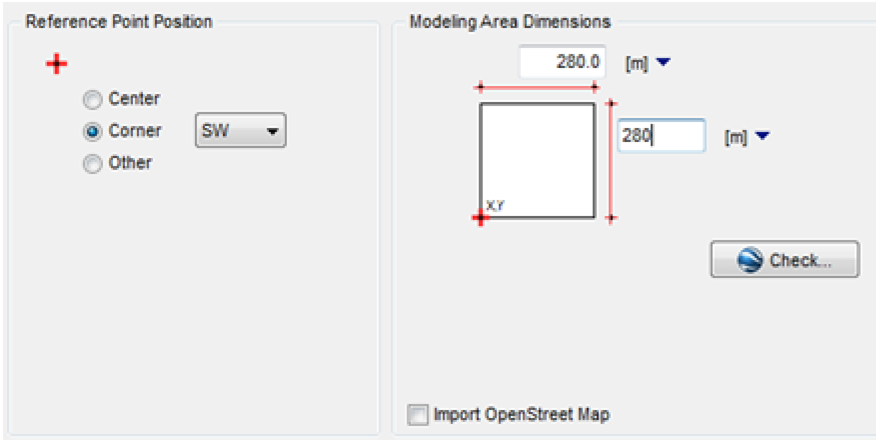
OSNG	• Specify X, Y in either BG or Lat/Long You cannot enter negative numbers for Lat/Long. Use the radio buttons to the right of the Lat/Long fields to select hemisphere. • Click to see the interface

2. In the **Specify Reference Point Position** and **Specify Modeling Area Dimensions** sections, select one of the reference point position options, and then set the size of the modeling area, as follows:

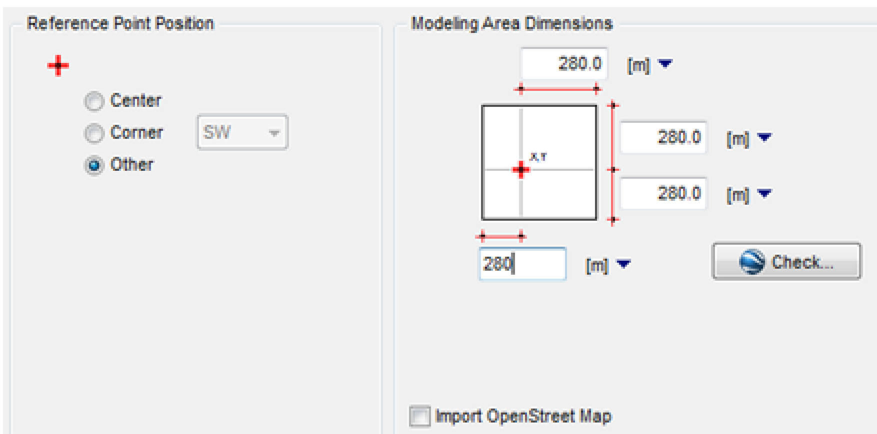
- Select **Center** to set the reference point at the center of the modeling area, and then specify a radius for the modeling area.



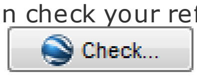
- Select **Corner** to set the reference point at a corner of the modeling area, then select the reference corner from the drop-down list (NW, NE, SE, SW) and specify the height and width for the modeling area.



- Select **Other** to set the reference point precisely within a rectangular modeling area, then complete the four fields to set the reference point.



3. If you are using coordinate systems that can be georeferenced, and have installed Google Earth™, you can check your reference point against Google Earth to ensure that it is accurate.

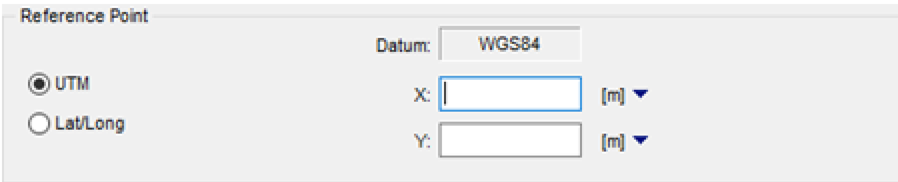
   Simply click the  button, which will launch Google Earth and navigate to the reference point.

> You can automatically import a tile map from OpenStreet server by checking the **Import OpenStreet Map** box before proceeding.

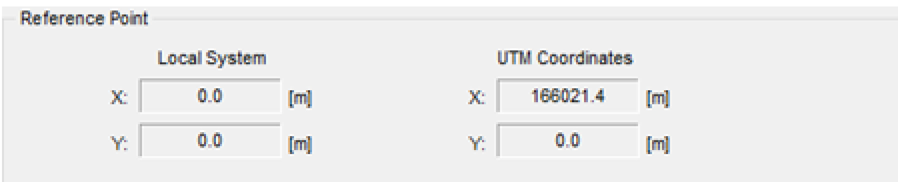
4. When you have completed all the information in the **New Project Wizard**, click on the **Finish** button.

Project Reference Point Options

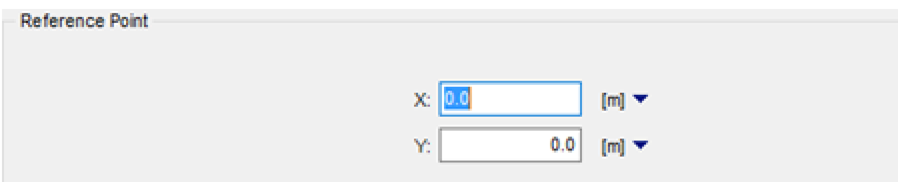
Reference Point Parameters - Universal Transverse Mercator



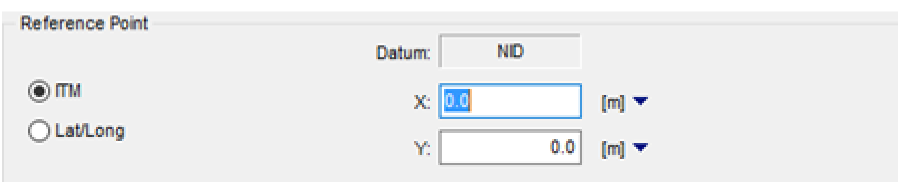
Reference Point Parameters - Local System with UTM Reference Point



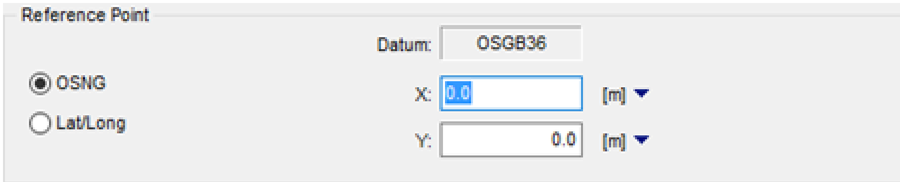
Reference Point Parameters - Unknown



Reference Point Parameters - Israeli Transverse Mercator



Reference Point Parameters - Ordnance Survey National Grid



Working with Base Maps

The **Working with Base Maps** section describes the AERMOD View's capabilities to import and export geographic data, including:

- Import Base Maps
- Import Tile Maps
- Import Multiple Base Maps
- Export Layer to Shapefile
- Export to Google Earth

Import Base Maps

The **Import | Base Maps...** option allows you to import image files in different formats as base maps for your.

The image files can be in the following formats:

- DLG
- DXF
- LULC
- MrSID and Raster Images
- Shapefiles

Importing DLG Base Maps

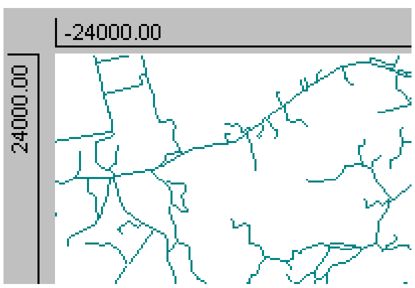
Digital Line Graph (DLG) is line map information in digital form produced by the U.S. Geological Survey (USGS). DLG files include information on planimetric base categories, such as transportation, hydrography, and boundaries.

How to Import DLG Base Maps:

1. Select **Import | Base Map | DLG...** from the menu or click on the Map Import tool ( ) located on the Annotation Toolbar. A floating menu is displayed. Select **DLG...** from the menu. The **Import DLG Base Map** dialog is displayed.

2. Enter the name and path for the DLG base map file (*.DLG) and click the **OK** button.

3. The base map you selected will be placed as an overlay on the drawing area.



DLG Base Map

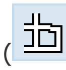
> It is important to note that the extents of the imported base map should be within the site/project domain area. Your base map will be placed on the same coordinates defined in your map file. If the extents of the base map are outside the defined site/project domain, then this base map will not be displayed on the drawing area.

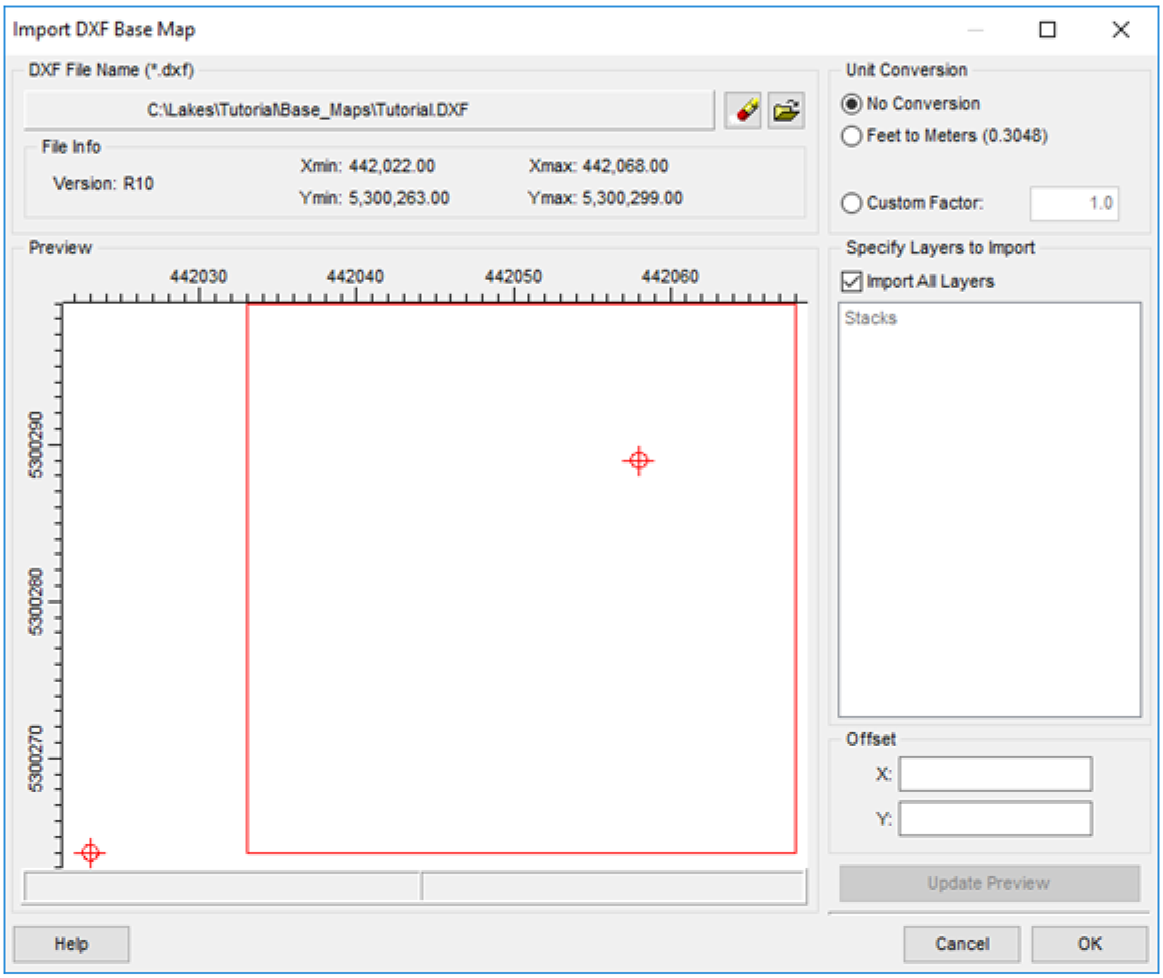
Importing DXF Base Maps

For easy reference and visualization of your results, you can import one or more digitized base maps for the area you are modeling. The function of the base map is to place the modeling results in context with the area to be modeled. The DXF (Drawing Interchange Format) file is a standard format for exchanging data between CAD systems. For instance, you can use Auto CAD software to convert your base map into the DXF format.

> The **Import DXF Base Map** utility can only read DXF objects that are consistent with the DXF Release 12 or earlier format. If your Auto CAD DXF file was created in a more recent release than R12, please re-save it as an R12 file before you attempt to import the file.

How to Import DXF Base Maps:

1. Select **Import | Base Map | DXF...** from the menu or click on the Map Import tool () located on the Annotation Toolbar. A floating menu is displayed. Select **DXF...** from the menu. The **Import DXF Base Map** dialog is displayed.



Import DXF Base Map dialog

The **Import DXF Base Map** dialog provides you with detailed control over the import of DXF files. Options available in this dialog include:

- **DXF File Name:** Clicking on the **Specify File** ( ) button allows you to specify the name and location of the DXF file you wish to import.

- **File Info:** Displays the version number of the application that was used to produce the file, as well as extents of the DXF map.

- **Preview:** The **Preview** window shows a preview of the DXF base map that has been specified. The axes of the preview area update automatically to the coordinates contained in the DXF base map.

- **Unit Conversion:** This section allows you to option of converting the existing units of the DXF base map. For the unit conversion, three options are available:

    - **No Conversion:** This is the default option used and the DXF will be imported without any conversions.

    - **Feet To Meters:** Will scale the DXF file by a factor of 0.3048, the equivalent conversion of feet to meters.

    - **Custom:** Allows you to specify a custom conversion factor to be applied to the DXF drawing.

> It is important to note that the extents of the imported base map should be within the site/project domain area. Your base map will be placed on the same coordinates defined in your map file. If the extents of the base map are outside the defined site/project domain, then this base map will not be displayed on the drawing area.

- **Specify Layers to Import:** Enables you to specify a particular layer from the DXF be imported rather than all of the layers contained in the entire DXF. The default option imports all layers. After making a layer selection you may wish to click the **Update Preview** button, which will visually show you in the **Preview** window what layers you are importing.

- **Offset:** This section allows you to shift the imported map in X and Y direction.

2. After specifying any import options, click the **OK** button to import the DXF into your project. Your **DXF Base Map** will be placed as an overlay on the drawing area.

The following objects are recognized from DXF files to be imported into your project:

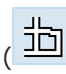
- **Lines and Poly lines**
- **Text**
- **Point Objects**

- **Circle**
- **Solid**
- **Face3D**
- **Arc**

Importing LULC Base Maps

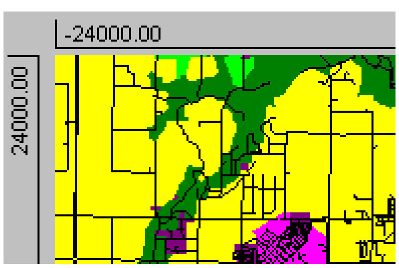
You can import **Land Use and Land Cover (LULC)** digital data to be used as base maps.

How to Import LULC Base Maps:

1. Select **Import | Base Map | LULC...** from the menu or click on the Map Import tool () located on the Annotation Toolbar. A floating menu is displayed. Select **LULC...** from the menu. The **Import LULC Base Map** dialog is displayed.

2. Enter the name and path for the LULC base map file (*.LUC) and click the **OK** button.

3. The base map you selected will be placed as an overlay on the drawing area.



LULC Base Map

> It is important to note that the extents of the imported base map should be within the site/project domain area. Your base map will be placed on the same coordinates defined in your map file. If the extents of the base map are outside the defined site/project domain, then this base map will not be displayed on the drawing area.

Land Use and Land Cover digital data provide information on nine major classes of land use such as urban, agricultural, or forest. Land Use and Land Cover data compilation is based upon the USGS

(United States Geological Survey) classification system which are generalized into first and second levels (i.e., Level I and Level II). The **Level II** classification category is used to define the code selection in the **Land Use Code Selection** section of the **Land User Creator** dialog. Such Land Use and Land Cover files are in CTG (Composite Theme Grid) format with the resolution of 200 meters. See the table below:

Table: USGS Land Cover Classification Categories

Level I		Level II	
10	Urban or Built-Up Land	11	Residential
		12	Commercial and Services
		13	Industrial
		14	Transportation, Communications and Utilities
		15	Industrial and Commercial Complexes
		16	Mixed Urban or Built-Up Land
		17	Other Urban or Built-Up Land
20	Agricultural Land	21	Cropland and Pasture
		22	Orchards, Groves, Vineyards, Nurseries and Ornamental Horticultural Areas
		23	Confined Feeding Operations
		24	Other Agricultural
30	Rangeland	31	Herbaceous Rangeland
		32	Shrub and Brush Rangeland
		33	Mixed Rangeland
40	Forest Land	41	Deciduous Forest
		42	Evergreen Forest

Level I		Level II	
		43	Mixed Forest
50	Water	51	Streams and Canals
		52	Lakes
		53	Reservoirs
		54	Bays and Estuaries
		55	Oceans and Seas
60	Wetland	61	Forested Wetland
		62	Nonforested Wetland
70	Barren Land	71	Dry Salt Flats
		72	Beaches
		73	Sandy Areas other than Beaches
		74	Bare Exposed Rock
		75	Strip Mines, Quarries and Gravel Pits
		76	Transitional Areas
		77	Mixed Barren Land
80	Tundra	81	Shrub and Brush Tundra
		82	Herbaceous Tundra
		83	Bare Ground
		84	Wet Tundra
		85	Mixed Tundra
90	Perennial Snow and Ice	91	Perennial Snowfields

Level I		Level II	
		92	Glaciers

Importing Raster Images

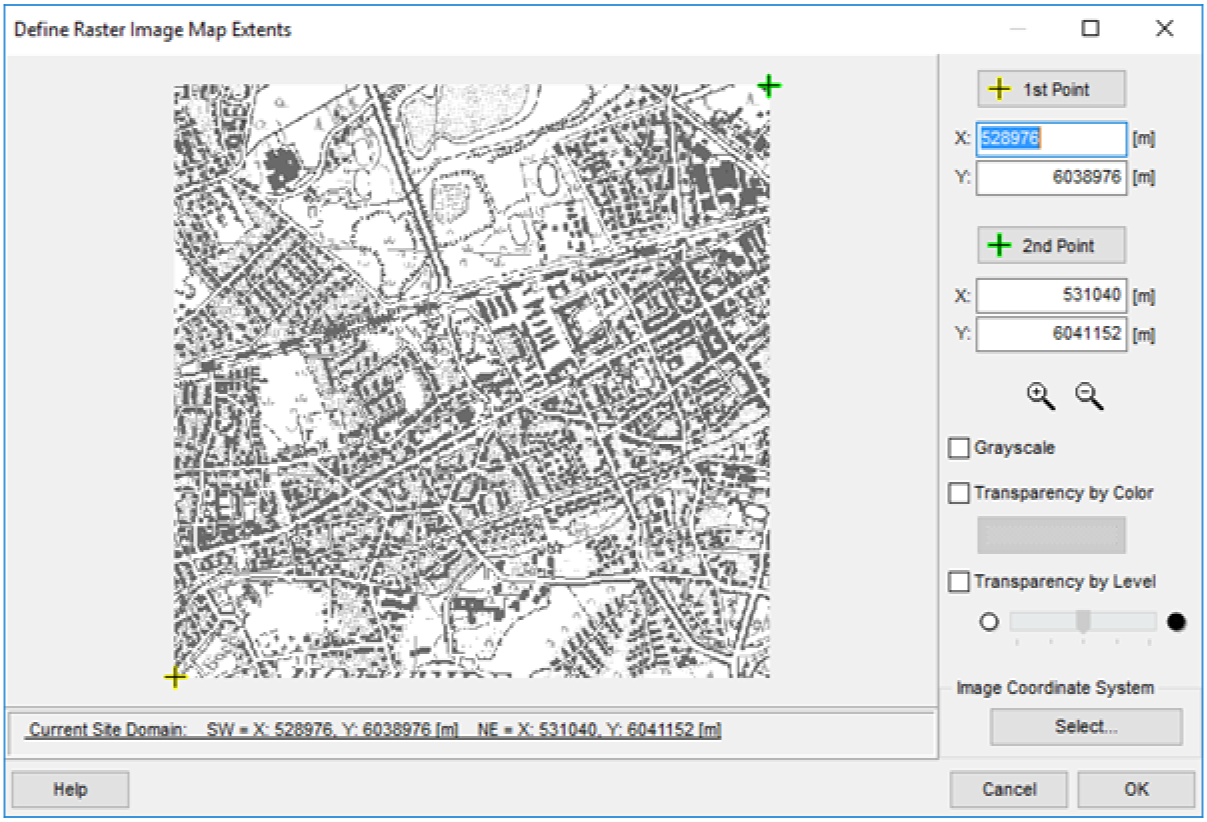
You can import graphical images to be used as base maps. These formats of graphics do not contain information on the site coordinates and extents of the image. However, you will be prompted for this information. If your image file has a World File associated with it, and is in the same directory as you image file, then the information in the world file will be used and you will not be prompted for site coordinates and extents.

How to Import Bitmap, JPEG, TIFF, GIF, PNG and MrSID Base Maps:

1. Select **Import | Base Maps | Raster Images...** from the menu or click on the Map Import tool () located on the Annotation Toolbar. A floating menu is displayed. Select **Raster Images...** from the menu. The **Import Raster Image** dialog is displayed.

> See also TIFF Images and MrSID Images for more information on these image formats.

2. Enter the name and path for the base map file (*.BMP, *.JPG, *.TIF, *.PNG, *.GIF or *.SID) and click the **Open** button. The **Define Raster Image Map Extents** dialog is displayed containing the image you have imported.



Define Map Extents dialog

3. Click on the **Zoom In** tool to select the **Zoom In** option. Your mouse pointer changes to a magnifying glass. Click on the location you want to zoom in. Click as many times as necessary, until you have the right magnification. To go back to the original size, right click on the image and select **Zoom Out** from the floating menu. You can also use the **Zoom Out** tool.

4. Click on the **1st Point** button and then click on the image where the location of your first point is. The Location dialog appears requesting you to input the **X** and **Y** coordinates for the first point. Repeat this for the second point, by clicking on the **2nd Point** button. Note that a yellow cross defines the location of your first point and a green cross defines the location of your second point. The specified coordinates for these two points are displayed in a text boxes on the right hand side of the dialog below the **1st Point** button and the **2nd Point** button respectively. You can adjust the displayed coordinates at any time.

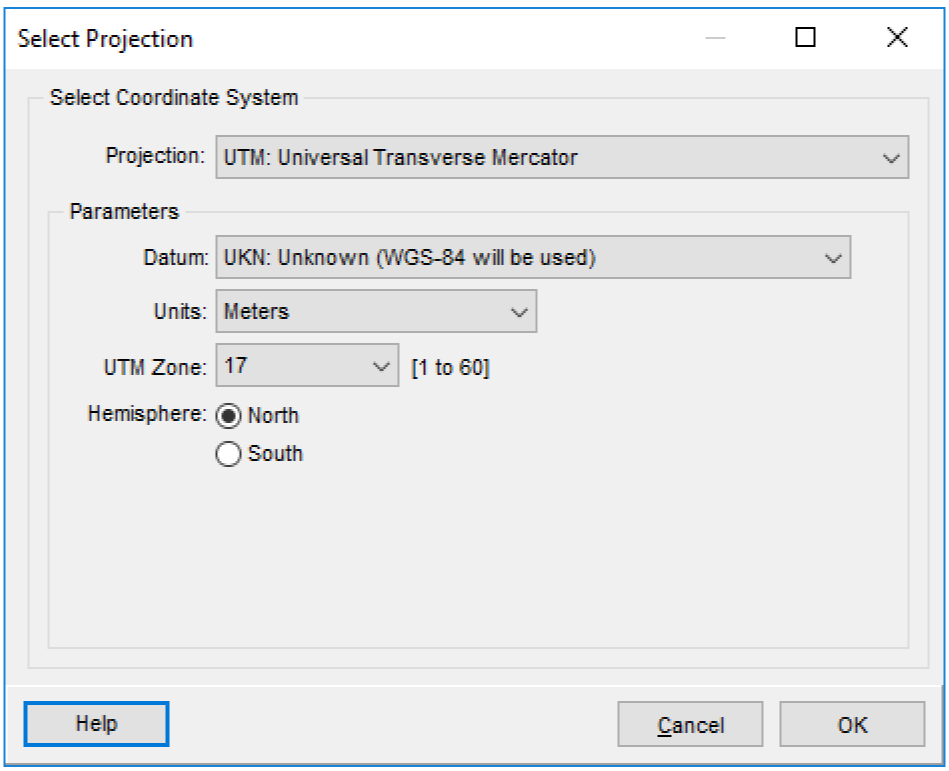
5. The **Grayscale** option allows you to import color maps as black and white.

6. The **Transparency by Color** option allows you to select a color in the bitmap that will be transparent when displayed in the drawing area. This option is usually used for black and white bitmaps, making the white portion of the map transparent. You can, however, click on the color bar to display the Color dialog from where you can select the color to make transparent.

7. The **Transparency by Level** option allows you to import a color map in which all colors are equally transparent. Set the level of transparency using the provided control (= completely transparent, = completely solid).

> The **Transparency by Color** and **Transparency by Level** options are mutually exclusive. You can adjust the settings after you have imported the map by double clicking on the map layer in the [Overlays tab](#) of the **Navigation Tree**, or by right-clicking on the layer and selecting **Edit** from the menu.

8. The **Image Coordinate System** option allows you to set the projection for your map. Click the Select... button to bring up the dialog.



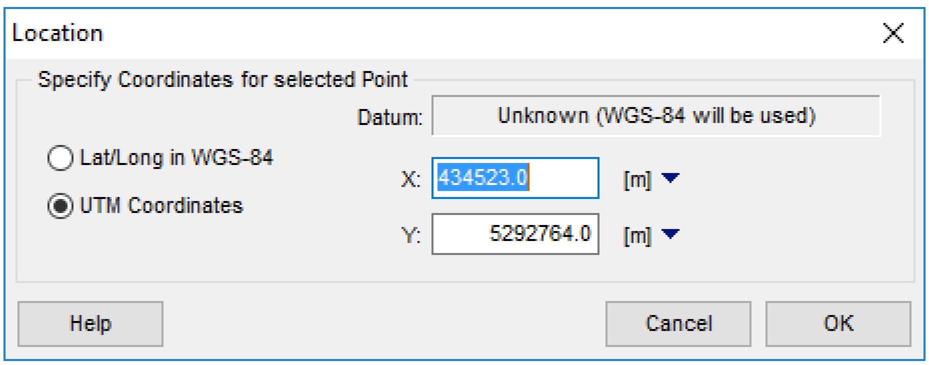
9. After defining the two points, click on the **OK** button. The imported image will be displayed as an overlay on the drawing area using the coordinates of the two points you have specified.

The image is imported using the Pyramid Method to minimize rendering and processing time. To learn more, refer to Pyramid Method.

Location

The **Location** dialog allows you to georeference base maps in raster format such as Bitmap, JPEG base maps. In this dialog you have the option of specifying the coordinates in Lat/Long or in the project's coordinate system (Gauss-Krueger or UTM).

You have access to this dialog by clicking on either the **1st Point** button or the **2nd Point** button in the Define Map Extents dialog.

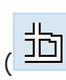


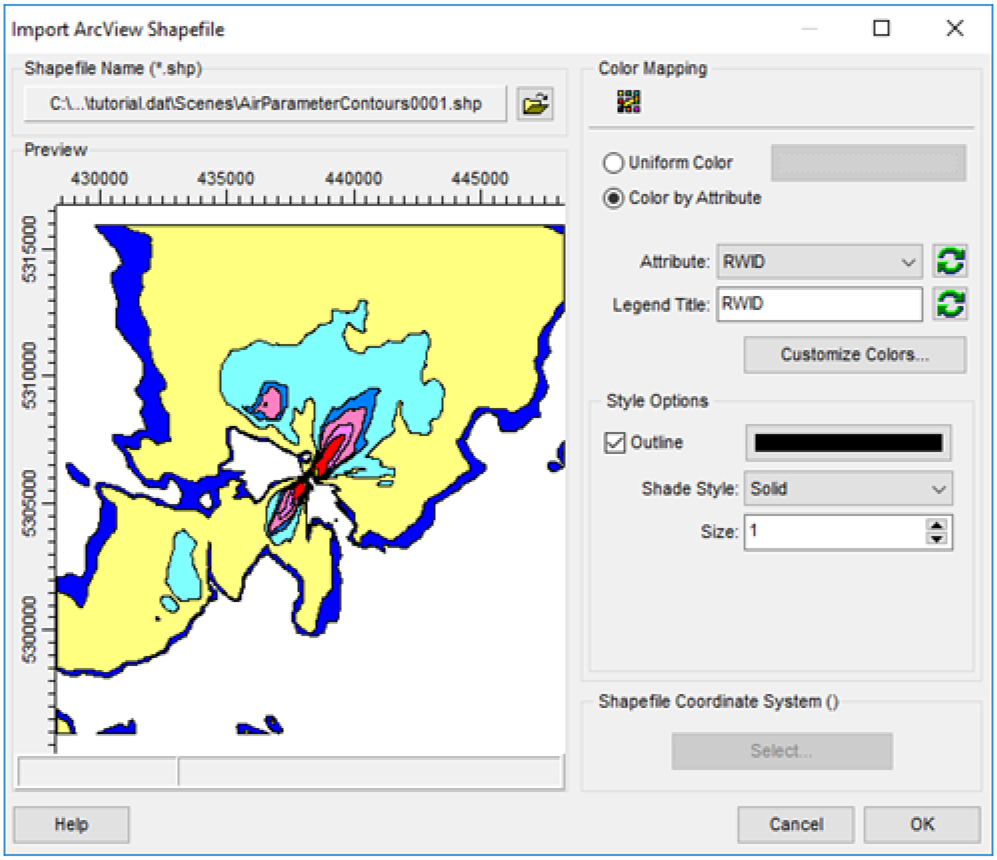
Location dialog

Importing ArcView Shapefiles

ArcView shapefiles store the geometric location and attribute information of geographic features.

How to Import ArcView Shapefiles:

1. Select **Import | Base Maps | Shapefile** from the menu or click on the Map Import tool () located on the Annotation Toolbar. A floating menu is displayed. Select **Shapefile...** from the menu. The **Import ArcView Shapefile** dialog is displayed.



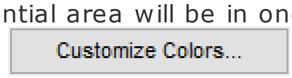
Import ArcView Shapefile dialog

2. Click the **Specify File** () button and specify the name and location of the shapefile (*.shp) you want to import. The **Preview** area displays the shapefile to be imported. The axes of the preview area update automatically to the coordinates contained in the shapefile.

3. The following graphical properties are available for you to customize the display of your shapefile:

- **Color Mapping:** You have two options in this panel -

  - **Uniform Color:** This option is the default. If this option is selected, then your shapefile will be imported in the fixed color you have selected. Click on the color bar to display the Color dialog where you can select a color of your choice.

  - **Color by Attribute:** This option allows you to select one of the shapefile's attributes as the color scheme. For example, suppose you have an attribute called LUCODE for your shapefile, which contains land use features such as residential areas, industrial areas, water and agricultural land following the Land Use and Land Cover classification system. If you select this attribute from the drop-down list, then random colors will be assigned for each geographic feature. This means that all the residential area will be in one color, the industrial areas in another color, and so on. Press the 

button

to display the Custom Attribute Colors dialog from where you can define colors for each attribute class.

- **Style Options:** In this panel you can customize the style of your shapefile. From the **Style** drop down list you can select a style to apply to the shapefile. If you would like to display the outlines of the shapefile, check the **Outline** box. The color bar becomes active which you can click on to display the Color dialog from where you can select a color for the outline.

- **Shapefile Coordinate System (GKL):** AERMOD View can import shapefiles created in a coordinate system that is different from the project's coordinate system. For example, if your project was setup in Gauss-Krueger (3-deg zones) coordinates, you will be able to import a shapefile that was created in UTM coordinates. You can click on the Select... button to open the Map Projection dialog where you can specify the coordinate system for the shapefile being imported.

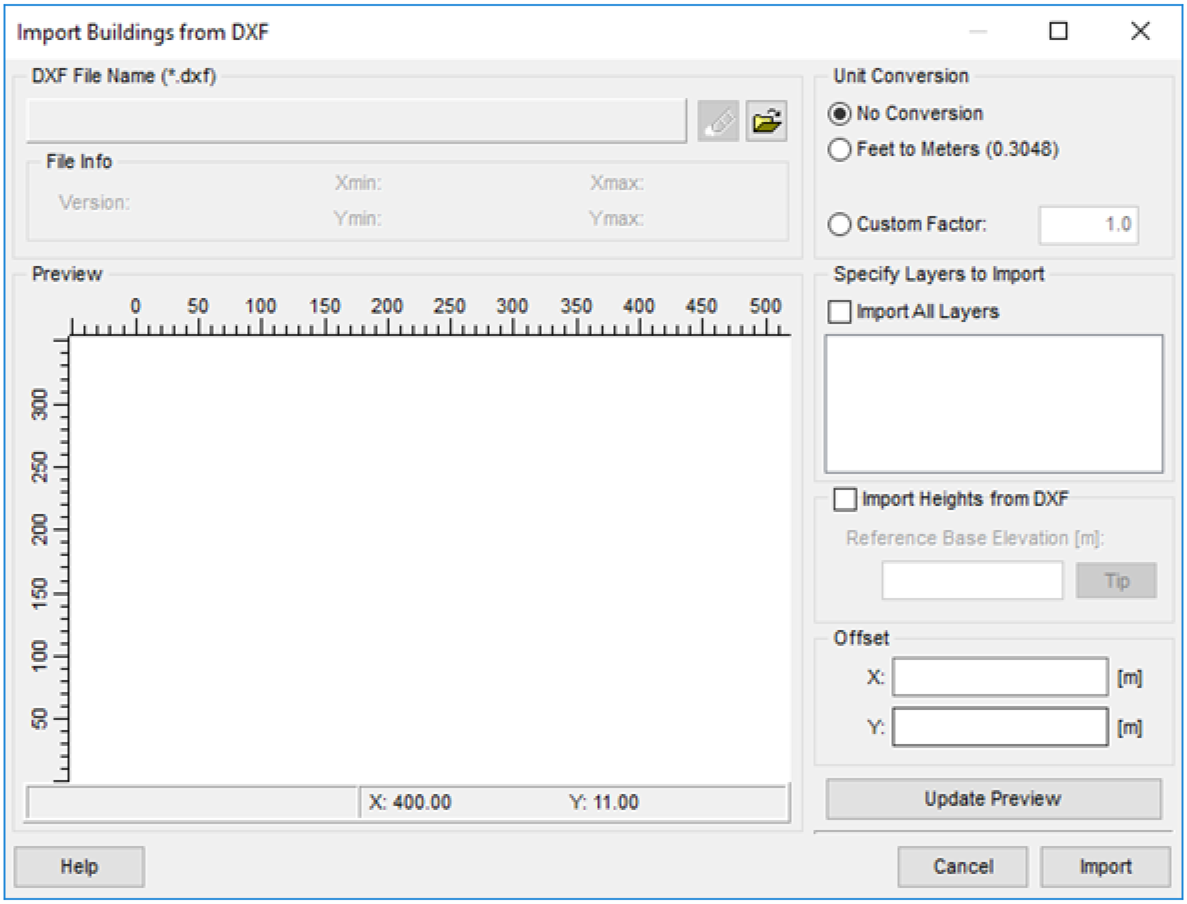
4. Click the **OK** button to import your shapefile.

Import Buildings from DXF

To minimize the time spent to draw buildings, you can import building locations and dimensions from an AutoCAD DXF file.

How to Import Buildings from an AutoCAD DXF file:

1. Select **Import | Buildings | AutoCAD DXF...** from the menu to display the **Import Buildings from DXF** dialog. Click on the button to specify the name and location of the DXF file containing the buildings to be imported. The name and path of the imported DXF file will be displayed in the **File Name** panel. The DXF map is displayed in the **Preview** panel and the DXF file information automatically displayed in the **Import Buildings from DXF** dialog.



Import Buildings from DXF dialog

2. The **Unit Conversion** panel provides you with the option of converting the existing units of the DXF file to meters in case your DXF map was created in units other than meters. For the unit conversion, three options are available:

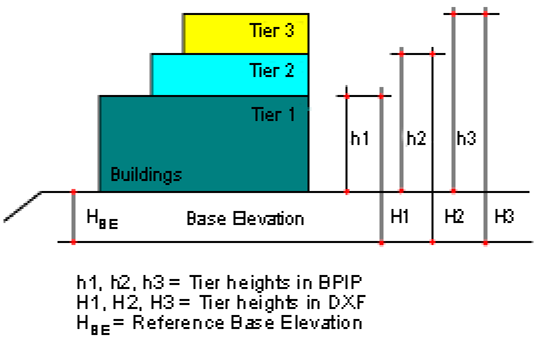
- **No Conversion:** This is the default option used and the DXF will be imported without any conversions.
- **Feet To Meters:** Will scale the DXF file by a factor of 0.3048, the equivalent conversion of feet to meters.
- **Custom:** Allows you to specify a custom conversion factor to be applied to the DXF drawing.

3. In the **Specify Layers to Import** section you can either **Import All Layers** by checking this box or uncheck it and select layers from the list. You can see any specific layer by selecting it in the **Specify Layers to Import** list and clicking the **Update Preview** button.

4. If your DXF file contains real world elevation values (z coordinate), this information can be imported as well. Check the **Import Heights from DXF** box and specify the **Base Elevation** for the building.

5. Use the **Offset** feature to shift your DXF in the X and Y direction, for example, to convert building coordinates from Local Cartesian coordinate system to UTM.

6. Click the Import button to import the building and close the dialog. All the imported buildings are automatically displayed in the drawing area.



In creating your DXF, a reference base elevation may have been included (ie: elevation above MSL). To subtract this value from your imported tier heights, enter the reference base elevation on which your DXF was created. In the case your DXF was created without a reference base elevation, you can enter a value of zero. You can also click on the Tip button to display the **Reference Base Elevation** dialog where you can specify your reference base elevation.

Lines or polygons with contingent end points are recognized from DXF files to be imported into projects when building data is imported from DXF

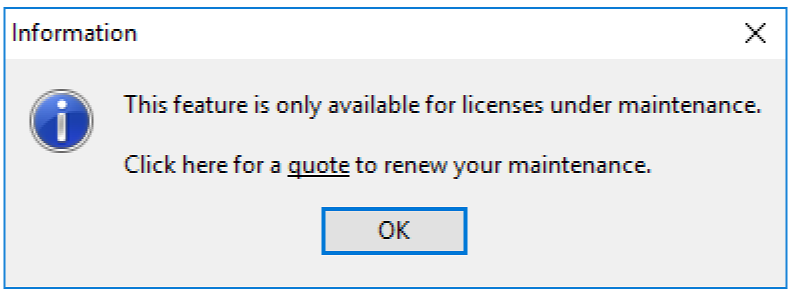
Import Tile Maps

You can import maps into your project directly from various online resources using the **Import Tile Map** function. To import a map, select **Import | Tile Maps...** The dialog below will be displayed.

> The **Open Street Map** and **Open Cycle Map** is only available for users with current maintenance.

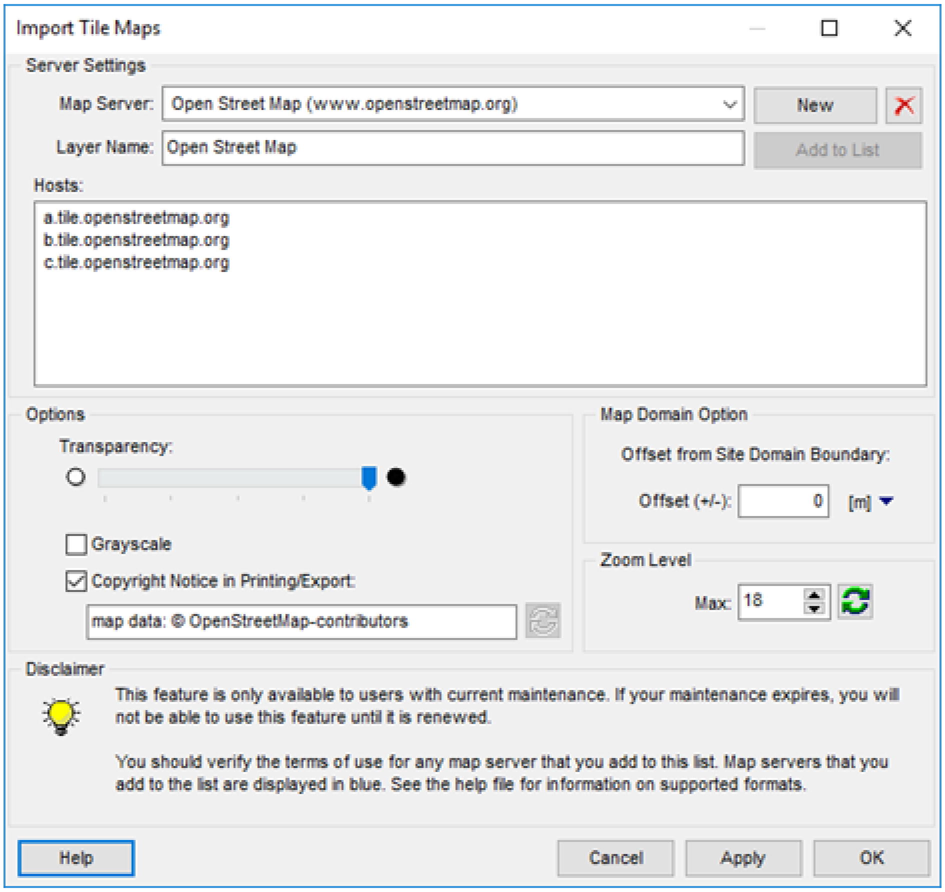
> Due to high acquisition and maintenance cost, the **Lakes Satellite** service is only available to users with current **paid** maintenance agreement.

If you do not have current maintenance, the following message will be displayed:



You may click on the "quote" to be redirected to the website where you can purchase or update your maintenance.

If your maintenance is current, the following dialog will be displayed:

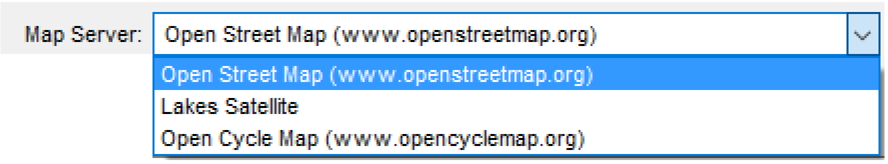


Import Image from Tile Map Server dialog

> In addition to the options/settings available in this dialog, you can also select the desired quality of the tile maps in the Preferences dialog (**File | Preferences | Settings | World Map Settings**). In the **World Map Options** found in this particular dialog, you can select the quality of the imported tile maps to be shown in 3D View or any printed materials. Note that higher tile map quality would result in the increase in processing time and the amount of data to be downloaded.

Tile Map Server Settings

- **Map Server:** Select the server from which you wish to import your map. There are several pre-programmed servers available, but you can also add a custom server by clicking the **New** button. If you use this option you will need to enter the host addresses in the **Hosts:** panel.



- **Layer Name:** Type in custom name for the layer. This name will appear in the Overlays tab.

- **Hosts:** Unique addresses for the servers that host the mapping services you wish to use. If you are creating an entry for a custom server you will need to fill in this panel.

> **Warning:** Not all tile map server formats are available. If you try to import maps from a custom server and are unable to do so, it may be because AERMOD View cannot accept this format. Please see Supported Tile Map Server Formats for more details, including on how to add new tile map servers. If the maps on this server are not copyright protected, please contact technical support and this format will be added to the next release.

> **Warning:** If you are adding a custom map server it is your responsibility to verify copyright requirements.

> **Warning:** Use of **Lakes Satellite** via third party mapping server is subject to the terms of use as stated in the end user license agreement.

Options

- **Transparency:** For some maps you can set the transparency, which will determine how opaque they appear in the project. A semi-transparent map will show features displayed on layers below it.

- **Grayscale:** If you are importing a color map, you can convert it to black and white by checking this box.

- **Copyright Notice in Printing/Export:** This option allows you to generate a copyright notice on the bottom left corner of any printing/exporting tile map image. The default notice for each type of tile map is available by pressing the button.

Map Domain Options

- **Offset (+/-):** Determines the extent of the imported map relative to the Site Domain Boundary. Positive offset will import a map greater than the site domain and a negative offset will import a smaller map.

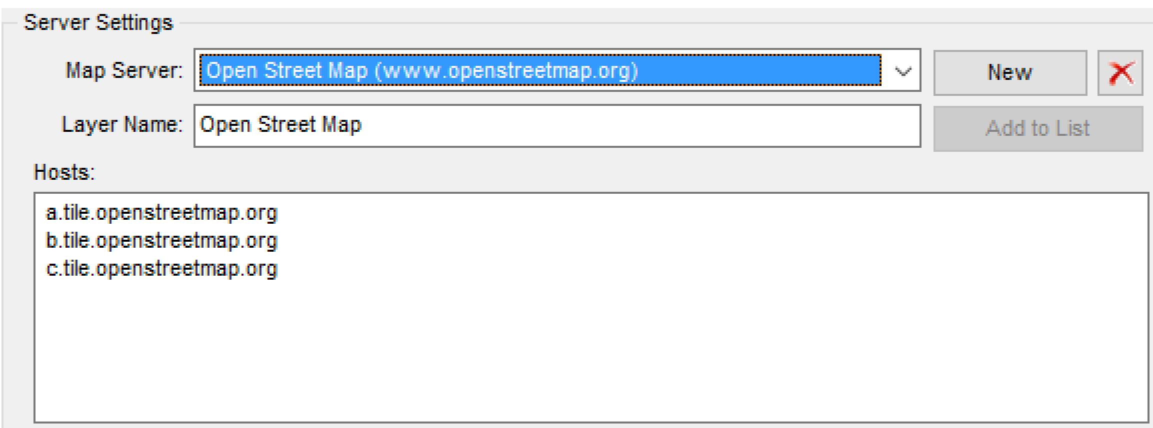
Zoom Level

- **Max Zoom Level:** The maximum number of times you can zoom into the server-provided map before loss of picture. If you continue to zoom, the image will be enlarged, but no new details will be available.

Supported Tile Map Server Formats

Currently AERMOD View tile map import utility supports the following format:

"server-host-address"/[z]/[x]/[y].png



In the image above one of the server host addresses would be **a.tile.opencyclemap.org/cycle**. To retrieve the image from this server, AERMOD View will automatically add the zoom, x-, and y-coordinates for the region you are importing. For any given server there are usually multiple redundant host addresses to provide a faster service and prevent interruptions in case one host is having technical difficulties.

How to add new Map Servers

1. Click the **New** button beside **Map Server** drop-down menu.
2. Type in **Layer Name**
3. In the **Hosts** panel enter the host address(es) for the map server you wish to add. If the host address does not conform to the format **"server-host-address"/[z]/[x]/[y].png** you have to enter the entire address, including the positions of x, y, and zoom and the extension of the image file.

4. Click **Add to List** button.

5. Click **OK**.

It is the user's responsibility to obtain the proper map server host address(es).

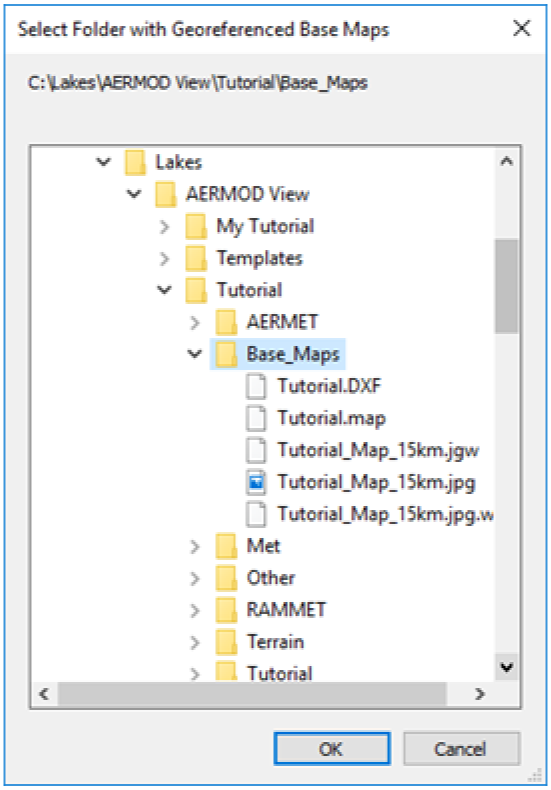
The dialog will close and the map will be imported. If the map does not load, one or more of the following may be an issue:

- Internet connection
- Proxy settings
- Error in the host address
- Host address format is not supported

If you are unable to resolve the issue, please contact technical support.

Import Multiple Base Maps

Selecting **Import | Multiple Base Maps...** allows you to easily locate any georeferenced base maps that are contained in the specified folder and quickly import them into an AERMOD View project.



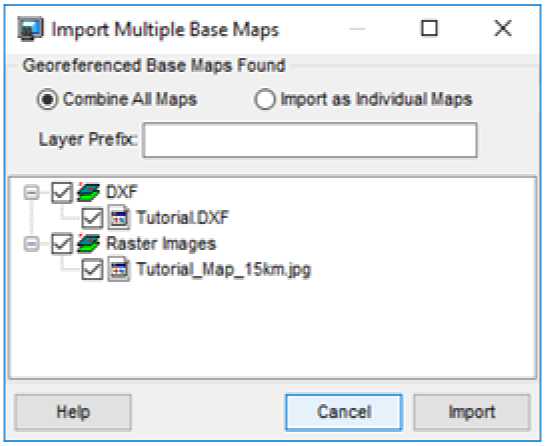
Select Folder with Georeferenced Base Maps dialog

How to Locate Georeferenced Base Maps

1. Select **Import | Multiple Base Maps...** from the menu.

2. The **Select Folder with Georeferenced Base Maps** dialog displays. Locate and select a folder that you wish to search. The subfolders will **NOT** be searched. Click **OK**.

3. When the search is complete, the **Import Multiple Base Maps** dialog displays a list of all of the located files, sorted by type.



Import Multiple Base Maps dialog

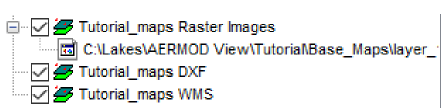
> Even if the folder contains more georeferenced maps, only maps that fall within the project domain will be displayed.

4. To select which files to import, place or leave a check mark beside the files or file types you wish to import. Clear the checkbox beside any files or file types you do not wish to import.

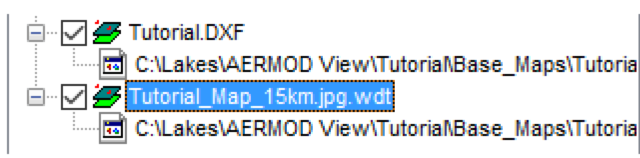
5. Select whether you wish to **Combine All Maps**, which will import all maps of any given type as one file, or to **Import as Individual Maps**, which will create separate nodes for each map file under the Overlays tab.

6. Type in a **Layer Prefix** you wish to use for the combined imported map nodes.

7. Click the **Import** button.



Imported combined maps



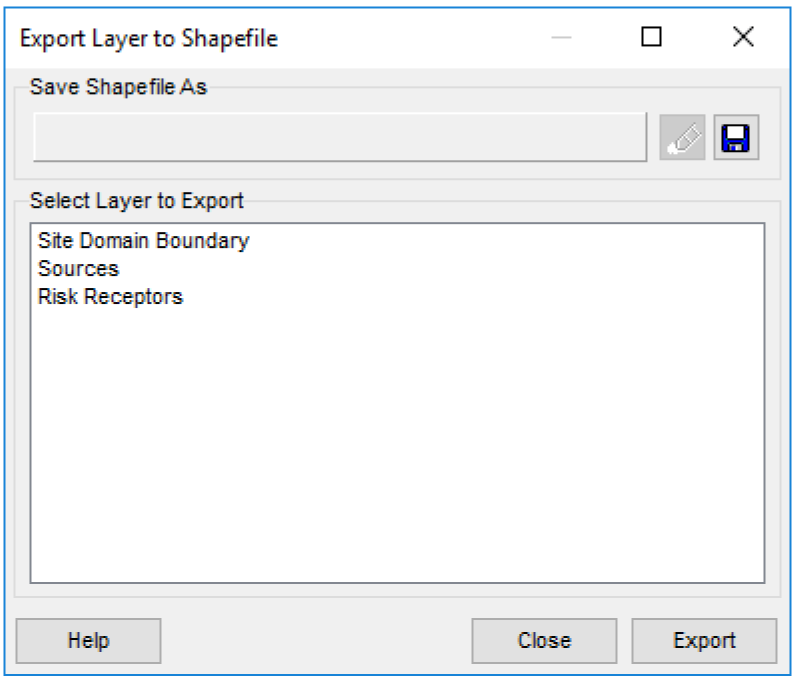
Imported individual maps

Export Layer to Shapefile

ArcView shapefile files (*.shp) store the geometric location and attribute information of geographic features.

How to Export Layers to Shapefiles:

1. Select **Export | Shapefiles...** from the menu. The **Export Layer to Shapefile** dialog is displayed.



Export Layer to Shapefile dialog

2. The **Export Layer to Shapefile** dialog provides you with detailed control over exporting data to shapefiles. Options available in this dialog include:

- **Save Shapefile As:** Here you specify the name and location for the shapefile you wish to create. Clicking on the  button enables you to browse your computer and define the file name and location.

- **Select Layer to Export:** This displays a list of layers currently defined in your project that can be exported to shapefile. Select the layer that is to be exported by clicking on it.

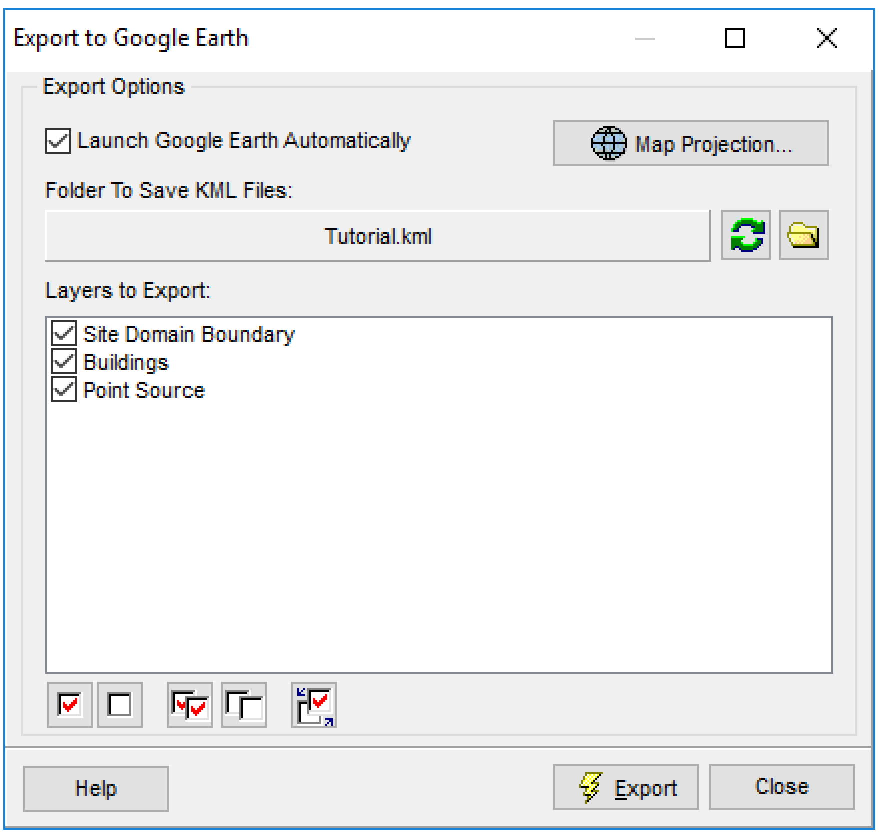
3. Once you define the name and location of the shapefile you wish to create, and have selected the layer containing the data you want to export, click the **Export** button to create the shapefile.

Export to Google Earth

You can export layers/overlays from your project to a KML (Keyhole Markup Language) file that can then be opened in Google Earth. This allows you to clearly envision your project and its results using the high quality images provided by Google Earth. Layers must be present in a project to be exported to Google Earth.

For information about using Google Earth, please see the Google Earth webpage.

> Refer to Google Earth Licensing for important information about using Google Earth.



Export to Google Earth dialog

How to Export to Google Earth:

1. Select **Export | Google Earth...** from the menu.

2. Select the folder in which to save the KML files; by default, the folder is named <projectName>.KML, and is located inside the project folder. To change this, select the **Select Folder** button ( ) and specify a new location.

3. In the *Layers to Export* list, leave checked all of the layers that you wish to include in the exported KML file. Click the checkbox to uncheck any layers that you do not wish to include.

Layers that can be exported from a modeling project.

- **Contours** - The currently displayed contours for the model results, also known as isopleths, are exported to Google Earth as they are shown in the drawing area. Each contour level/ color will be exported separately allowing you to turn on/off the display of a specific level. A representation of the color ramp is also exported and can be turned on/off under the Legend layer. Use the Alpha Blend color option for transparency effects in Google Earth.

- **Receptor Grids** - each receptor grid can be exported to Google Earth discretely

- **Receptors** - displays the receptors specified in your project

- **Site Domain Border** - This layer displays the site domain border, as defined in the Site Domain dialog

- **Links** - displays the links specified in your project

4. By default, Google Earth will be launched after the project is exported to a KML file. If you do not wish to launch Google Earth right away, uncheck the **Launch Google Earth Automatically** box; AERMOD View will only generate the KML files for future use.

5. If you wish to change the map projection/datum of the project, click the **Map Projection** button and set the Projection, Datum, and UTM Zone for the exported file in the Map Projection dialog.

> It is extremely important that before you export to Google Earth, you check the map projection/datum for the project. This ensures that AERMOD View will output your project layers to the correct position on Earth.

6. Click the **Export** button to export the project layers to the KML file.

What is Google Earth?

What Is Google Earth™?

Google Earth is a geographic browser that is available from Google™. It maps the earth by the superimposition of images obtained from satellite imagery, aerial photography and GIS 3D globe. Google Earth displays satellite images of varying resolution of the Earth's surface. Most land is covered in at least 15 meters of resolution.

What Are KML Files?

Google Earth is able to show images overlaid on the surface of the earth using Keyhole Markup Language (KML). KML files specify a set of features for display in Google Earth.

Google Earth Licensing

Google, Inc. provides three product levels to Google Earth, which you should investigate before using AERMOD View. For the most timely and complete Google Earth licensing options, please consult Google's website and license agreement. Please choose the Google Earth product that meets your organization's needs. **Lakes Environmental does not authorize or license any of these products for your use.**

Third-Party Data Licensing

When you use licensed data within AERMOD View to create files for export to Google Earth, make sure you have the appropriate rights from the data provider to share those exported files with third parties.

> Depending on the data you use, you may be responsible for displaying a copyright on any maps that result from exporting maps from Lakes Environmental software. Additionally, you may be required to purchase additional data licenses.

How to Acquire Google Earth:

1. Access the Google Earth website at http://earth.google.com/.

2. Determine your required product level.

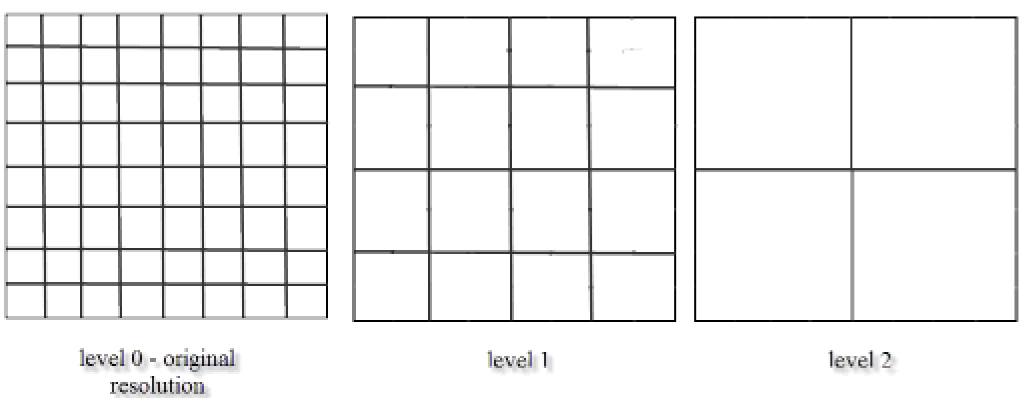
3. Download or purchase the appropriate Google Earth package, and install it according to the documentation provided by Google Earth.

Accessory Information

Raster Pyramid Method

The Pyramid method involves representing the raster as a series of images with progressively reduced resolution. This method allows for improved display performance of the rasters, since at low zoom, when high resolution is unnecessary, an image of lower resolution is used.

With this method the original raster image becomes the first level of the pyramid - level 0. With each successively higher level, four pixels are represented with one, reducing the resolution.



Pyramid Raster Method

The images with different levels of resolution are created upon import and stored in the folder where the original image is located. When they are needed, they can be readily displayed without having to do interpolation each time the zoom level is changed. This method allows for lower display processing times and greater detail in images under zoom.

About the World File

The image-to-world transformation is a six parameter affine transformation of the form:

$x' = Ax + By + C$

$y' = Dx + Ey + F$

where

x' = calculated x-coordinate of the pixel on the map

y' = calculated y-coordinate of the pixel on the map

x = column number of a pixel in the image

y = row number of a pixel in the image

A = x-scale, dimension of a pixel in map units in the x-direction

D,B = rotation terms (not supported for this release)

E = y-scale (this value is always negative, because image space is top-down, whereas map space is bottom-up)

C = translation term; x-Origin (x-coordinate of the center of the upper left pixel)

F = translation term; y-Origin (y-coordinate of the center of the upper left pixel)

The transformation parameters are stored in the world file, an ASCII format, in this order, A, D, B, E, C, F; for example:

2.22123393184959	A
0.00000000000000	D
0.00000000000000	B
-2.22123393184959	E
10383.13600759092515	C
11611.48117990907485	F

Note that the parameter characters are included in the example for clarity. They do not actually appear in the file.

If a world file is not present and there is no georeferencing information in the header of the image, a default mapping is still provided between image space and map space. MapObjects LT makes the origin of the image (0.0, 0.0), and sets the X and Y scale factors both to 1.0.

If you want to display a non-georeferenced image on a portion of your map, supply the georeferencing world file yourself, and make sure it maps the image to the portion of the map that you want.

World File Naming Conventions

The world file associated with an image is named by following the conventions in the table below. For example, if you have an image that's stored in a file named myimage.bmp, then the world file associated with it must be named myimage.bmpw or myimage.bpw.

World files are available for the following base map files: *.bmp, *.jpg, *.tif or *.sid

If the file extension of the image is:	The world file extension must be:
bmp	bmpw or bpw
jpg; jpeg	jpgw or jgw
tif; tff; tiff	tfw
sid	sdw

About Datums

AERMOD View supports the following datum:

1. **WGS-84:** WGS-84 GRS 80 Global Coverage
2. **NAS-C:** NORTH AMERICAN 1927 Clarke 1866
3. **NAR-C:** NORTH AMERICAN 1986
4. **NWS-84:** NWS 6370km Radius, Global Sphere
5. **ESR-S:** ESRI REFERENCE Normal Sphere (6371 Radius)
6. **ED50:** European Datum 1950 (EUR-M)

TIFF/GeoTIFF Images

Tag **I**mage **F**ile **F**ormat (TIFF) supports black-and-white, grayscale, pseudocolor and true color images, all of which can be stored in a compressed or uncompressed format.

The following types of TIFF images can be displayed:

- Single band black and white (1 bit)
- Grayscale (4, 8, 16, 24 or 32 bits)
- Pseudocolor (4, 8, 16 bits)
- Multiband images with 8 bits per band. There is no restriction on the number of bands in the image.

The following compression schemes are supported:

- CCITT Group 3 and 4 algorithms (TIFF Compression Scheme 2, 3, 4 and 32771)
- JPEG post-6.0-style DCT algorithms (TIFF Compression Scheme 7)
- NeXT 2-bit encoding scheme (TIFF Compression Scheme 32766)
- Macintosh PackBits algorithm (TIFF Compression Scheme 32773)
- ThunderScan 4-bit run-length encoding (TIFF Compression Scheme 32809)
- SCI's compression scheme for high-resolution color images (TIFF Compression Scheme 34676 and 34677)

The following TIFF variants are supported:

- TIFF 6.0: (TIFF revision 6.0). Enhancements to image definition, RGB colorimetry. Includes JPEG compression.
- GeoTIFF: (TIFF with a Geo header)

MrSID Images

MrSID (**M**ulti**R**esolution **S**eamless **I**mage **D**atabase) imaging format has the unique function of storing information about every pixel of an image in an image database. This allows each successive resolution of a MrSID file (*.SID) to be created by supplementing the currently displayed image data with only the new data necessary to create the next resolution. This results in better image detail and clarity with each successive resolution.

UTM Zone Locations and Central Meridians

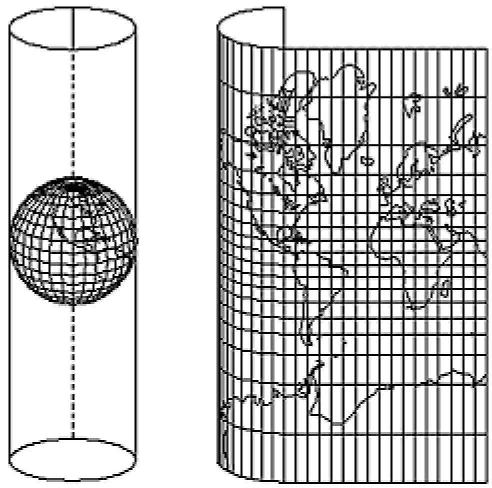
The following table indicates the degrees of longitude defining the 60 UTM zones around the world.

Zone	Central Meridian	Range	Zone	Central Meridian	Range
01	177W	180W - 174W	31	003E	000E - 006E
02	171W	174W - 168W	32	009E	006E - 012E
03	165W	168W - 162W	33	015E	012E - 018E
04	159W	162W - 156W	34	021E	018E - 024E
05	153W	156W - 150W	35	027E	024E - 030E
06	147W	150W - 144W	36	033E	030E - 036E
07	141W	144W - 138W	37	039E	036E - 042E
08	135W	138W - 132W	38	045E	042E - 048E
09	129W	132W - 126W	39	051E	048E - 054E
10	123W	126W - 120W	40	057E	054E - 060E
11	117W	120W - 114W	41	063E	060E - 066E
12	111W	114W - 108W	42	069E	066E - 072E
13	105W	108W - 102W	43	075E	072E - 078E
14	099W	102W - 096W	44	081E	078E - 084E
15	093W	096W - 090W	45	087E	084E - 090E
16	087W	090W - 084W	46	093E	090E - 096E
17	081W	084W - 078W	47	099E	096E - 102E
18	075W	078W - 072W	48	105E	102E - 108E
19	069W	072W - 066W	49	111E	108E - 114E

Zone	Central Meridian	Range	Zone	Central Meridian	Range
20	063W	066W - 060W	50	117E	114E - 120E
21	057W	060W - 054W	51	123E	120E - 126E
22	051W	054W - 048W	52	129E	126E - 132E
23	045W	048W - 042W	53	135E	132E - 138E
24	039W	042W - 036W	54	138E	138E - 144E
25	033W	036W - 030W	55	147E	144E - 150E
26	027W	030W - 024W	56	153E	150E - 156E
27	021W	024W - 018W	57	159E	156E - 162E
28	015W	018W - 012W	58	165E	162E - 168E
29	009W	012W - 006W	59	171E	168E - 174E
30	003W	006W - 000E	60	177E	174E - 180E

UTM Projection

The Universal Transverse Mercator (UTM) is a cylindrical projection, as depicted in the image below. In the UTM coordinate system the world is divided into 60 north-south zones, each covering a strip 6° wide in longitude. These zones are numbered consecutively beginning with Zone 1, between 180° and 174° west longitude, and progressing eastward to Zone 60, between 174° and 180° east longitude.

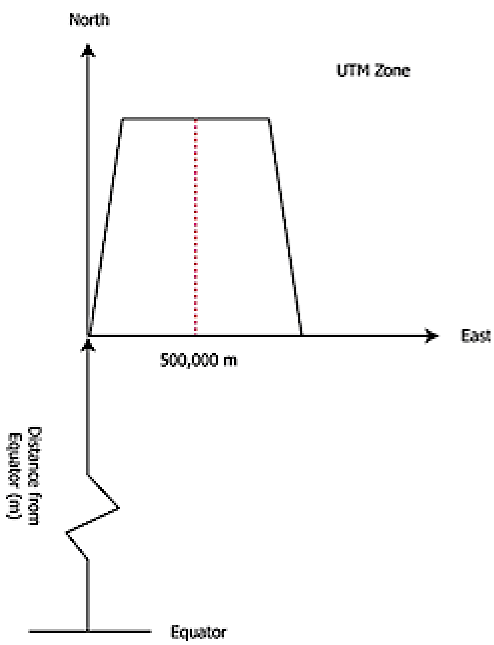


Cylindrical map projection system used in the UTM grid

In each UTM zone, coordinates are measured north and east in meters. The northing values are measured continuously from zero at the Equator, in a northerly direction. Southerly values are similarly measured from the Equator. A central meridian through the middle of each 6° zone is assigned an easting value of 500,000 meters, as shown in image below.

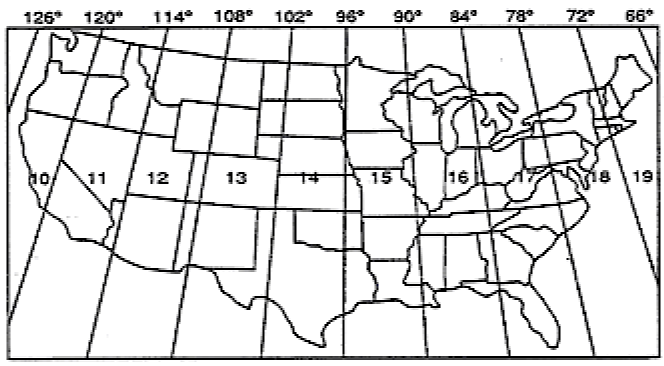
See also UTM Zone Locations and Central Meridians

UTM coordinates to the west of this central meridian are less than 500,000; and values to the east are more than 500,000.



UTM coordinate values within a UTM zone.

The image below shows the 10 zones that covers the continental USA.

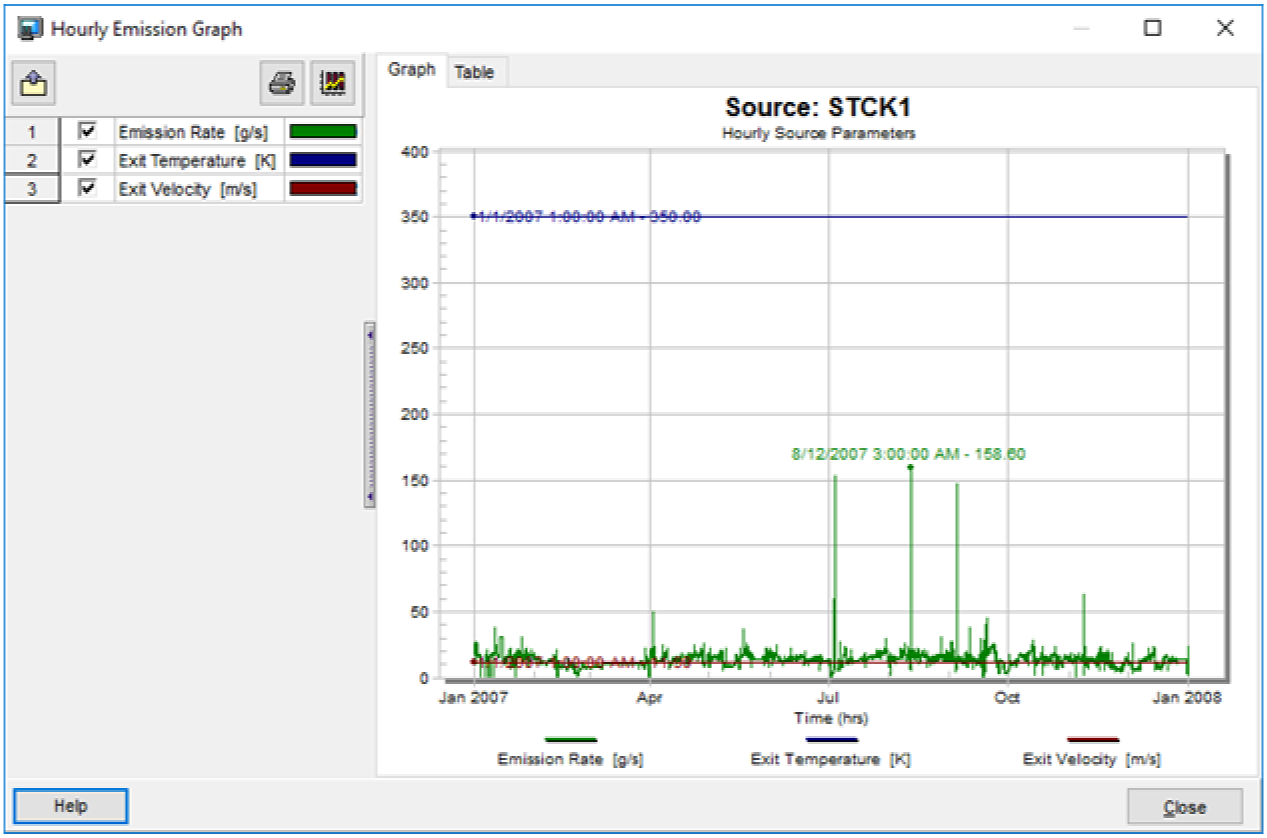


UTM zone covering the continental USA

Graph

Graph

The **Hourly Emission Graph** dialog allows you to view the time series emission data. To load the graph view, click the desired source in the Hourly Emission File Maker dialog and click the **Graph...** button.



Graph dialog

Viewing the Results as a Graph

The dialog opens with the results displayed as a graph. Check or uncheck the parameters in the navigation tree to display or hide them in the graph.

Right click on the graph to display a menu with formatting objects, including font size, plotting method, and grid display.

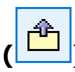
Viewing the Results as a Table

Select the **Table** tab to see the results displayed in a table. Check or uncheck the parameters in the navigation tree to display or hide them in the table.

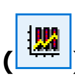
Using the Dialog

There are a number of options available in the **Graph** dialog as follows:

- **Navigation Tree:** The navigation tree allows you to select which monitor points will be displayed in the graph or table.
    - Right-click the navigation tree to view the popup menu, which allows you to select and unselect one or all of the monitor points.
    - Double-click the navigation tree to open the Subsets tab of the Graph Options dialog.
    - Hide the navigation tree by clicking the vertical pane control between the navigation tree and the results display.

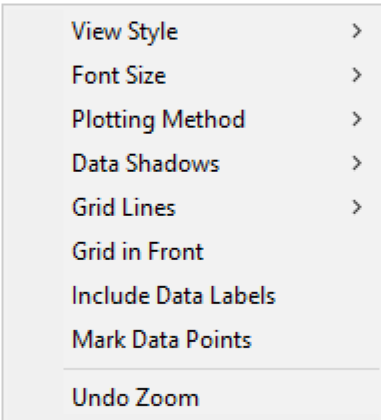
- **Export:** Clicking on the **Export (****)** button displays the following submenu options:
    - **Copy to File:** Displays the following sub-menu options -
        - **Bitmap.....:** Displays the **Save As** dialog, allowing you to save your graph as a Windows bitmap.
        - **Metafile...:** Displays the **Save As** dialog, allowing you to save your graph as a Windows metafile.
    - **Copy to Clipboard:** Displays the following sub-menu options -
        - **Bitmap.....:** Copies your graph to the clipboard as a Bitmap, allowing you to paste it into another windows application.
        - **Metafile...:** Copies your graph to the clipboard as a Metafile, allowing you to paste it into another windows application.
    - **CSV File...:** Allows you to export the hourly or daily concentration values at each monitor point to a *.csv file that can be opened in Microsoft® Excel®.
    - **Excel File...:** Allows you to export the hourly or daily concentration values at each monitor point to an *.xls file that can be opened in Microsoft Excel.

- **Print:** Clicking on the **Print (**) button opens the **Print** dialog which allows you to select options to print your graph.

- **Options:** Click on the **Graph Options (****)** button to open the Graph Options dialog, which allows you to select various display options for your graph. You can also enhance the appearance of your graph by clicking the right mouse button anywhere within the graph and choosing any one of the available Graph Menu Options.

Graph Menu Options

Using the mouse, click the right mouse button anywhere within the graph. The following menu appears:



- **View Style:** Adjusts the graph image to best suit printing on a monochrome or color printer. You can choose from Color, Monochrome, or Monochrome + Symbol. The default is Color. If you wish to print in color, choose the Color option from the adjoining list. If you wish to print in monochrome (black and white), choose Monochrome for graphs with less than four subsets, and choose Monochrome + Symbols for graphs with more than four subsets.

- **Font Size:** Adjusts the size of the graph image fonts for screen display and printing. You can choose from Large, Medium, or Small. The default is Medium. You should select either Medium or Small fonts when printing. For some graphs (i.e. generating an image for a highly rectangular graph), the font size may be automatically reduced in order to produce a higher quality image.

- **Plotting Method:** Sets the primary plot method of the graph. You can choose from one of 10 methods: Point, Line, Bar, Area, Stick, Points + Best Fit Line, Points + Best Fit Curve, Points + Line, Points + Spline, or Spline. The default is Line.

- **Data Shadows:** Toggles between drawing a black shadow behind the bars, points, and areas of a graph using a corresponding plotting method to produce a 3-D effect and no shadow. The default is shadow. The **Data Shadow** option has no effect on certain **Plotting Methods**, such as Line.

- **Grid Lines:** Displays graph grid lines. You can choose from Both X and Y Axes, Y Axis, X Axis, or No Grid. The default is Both X and Y Axes.

- **Grid In Front:** Toggles between drawing the graph grid lines on top of or behind the plotting method graphics and data points. The default is behind. If you choose to display no grid lines from the **Grid Lines** option, this option has no effect.

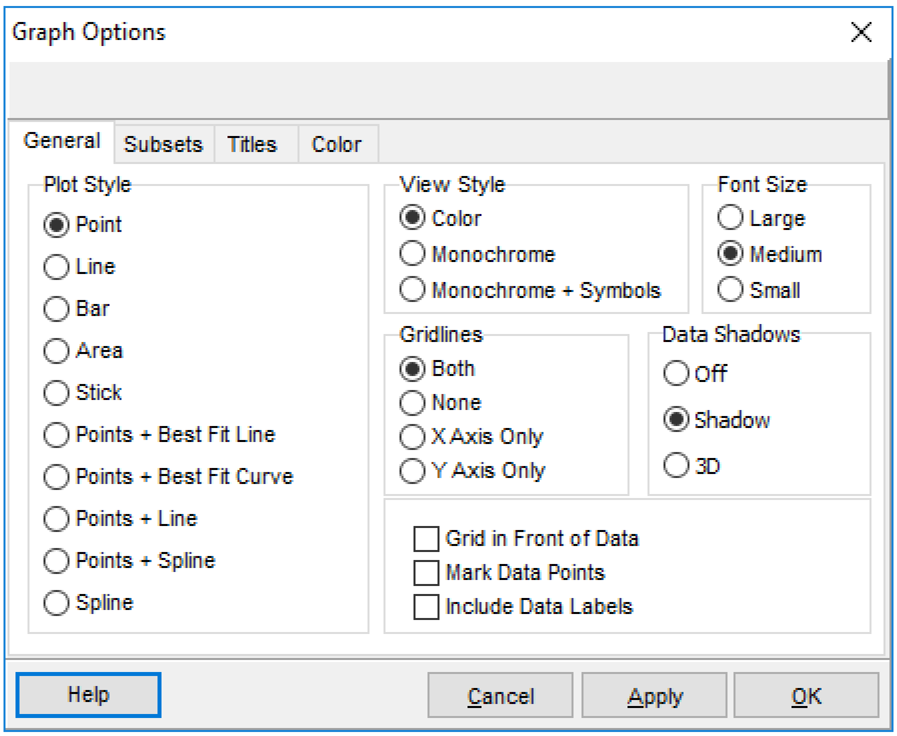
- **Include Data Labels:** Toggles whether or not the data labels are displayed on the graph. The default is not labeled. If you choose a **Bar Plotting Method**, this option has no effect.

- **Mark Data Points:** Toggles whether each data point on the graph is displayed on the graph with a dot or is not displayed. The default is not marked. If you choose a **Bar Plotting Method**, this option has no effect.

- **Undo Zoom:** Returns the graph area to its original view.

Graph Options

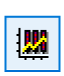
The **Graph Options** dialog contains options to set various graph attributes, such as all general settings, plot styles, subset display, points settings, axis settings, font settings and display, and color settings.



Graph Options dialog

The **Graph Options** dialog can be accessed in any one of the following ways:

- Double-click the mouse anywhere within the graph display; or

- Click on the  button in the **Graph** dialog

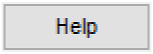
The **Graph Options** dialog has four tabs:

- General

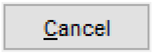
- [Subsets]
- [Titles]
- [Color]

Button Guide

At the bottom of the **Graph Options** dialog are a series of buttons. Their functions are described below:



Displays the Help for the **Graph Options** dialog.



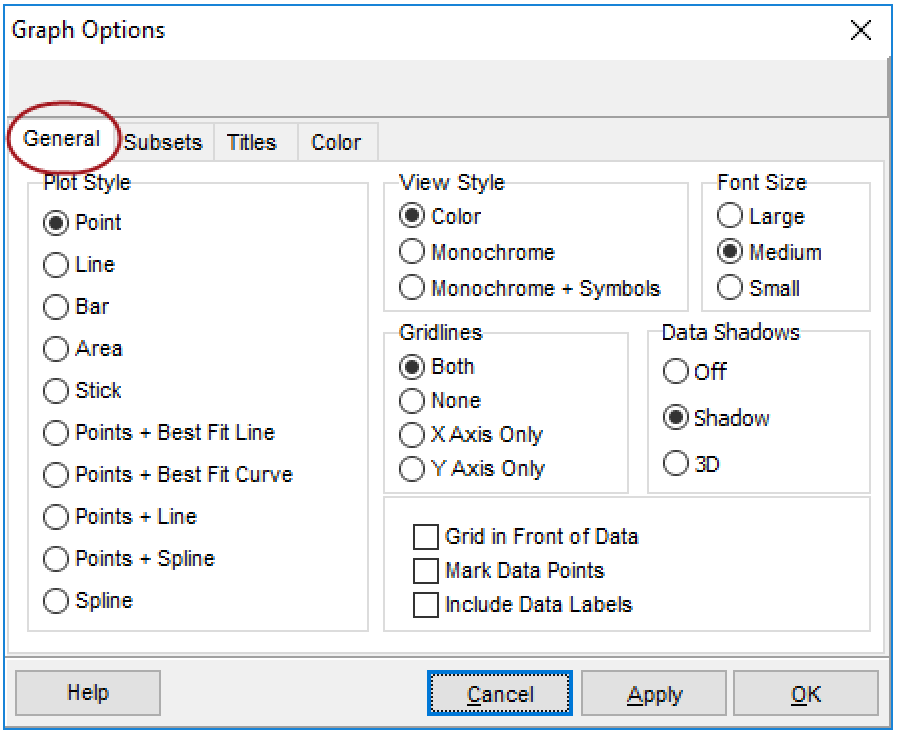
Exits the **Graph Options** dialog without implementing any changes to the graph.

Implements changes made to the graph but does not close the **Graph Options** dialog.

Implements changes made to the graph and closes the **Graph Options** dialog.

Graph Options - General

In the **General** tab of the Graph Options dialog, you define basic graph attributes and settings such as graph titles, viewing style, font size, grid lines, etc.



Graph Options dialog - General tab

The following options are available in this tab:

- **Plot Style:** Sets the primary plot method of the graph. You can choose from one of 10 methods: **Point**, **Line**, **Bar**, **Area**, **Stick**, **Points + Best Fit Line**, **Points + Best Fit Curve**, **Points + Line**, **Points + Spline**, or **Spline**. The default is **Line**.

- **View Style:** Adjusts the graph image to best suit printing on a monochrome or color printer. You can choose from **Color**, **Monochrome**, or **Monochrome + Symbol**. The default is **Color**. If you wish to print in color, choose the **Color** option from the adjoining list. If you wish to print in monochrome (black and white), choose **Monochrome** for graphs with less than four subsets, and choose **Monochrome + Symbols** for graphs with more than four subsets.

- **Font Size:** Adjusts the size of the graph image fonts for screen display and printing. You can choose from **Large**, **Medium**, or **Small**. The default is **Medium**. You should select either **Medium** or **Small** fonts when printing. For some graphs (i.e. generating an image for a highly rectangular graph), the font size may be automatically reduced in order to produce a higher quality image.

- **Gridlines:** Displays graph grid lines. You can choose from **Both**, **None**, **X Axis Only**, and **Y Axis Only**. The default is to display both gridlines.

- **Data Shadows:** Toggles between drawing a black shadow behind the bars, points, and areas of a graph using a corresponding plotting method to produce a 3-D effect and no shadow. The default is shadow. The **Data Shadow** option has no effect on certain plot styles, such as Line.

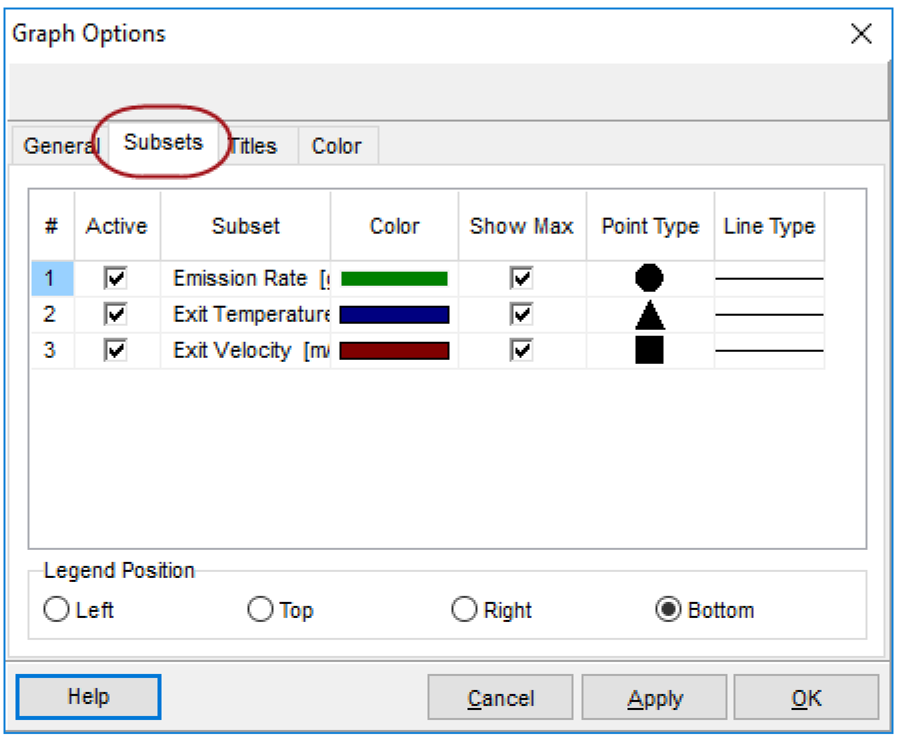
- **Grid in Front of Data:** Toggles between drawing the graph grid lines on top of or behind the plotting method graphics and data points. The default is behind. If you choose to display no grid lines from the **Grid Lines** option, this option has no effect.

- **Mark Data Points:** Toggles whether each data point on the graph is displayed on the graph with a dot or is not displayed. The default is not marked. If you choose the **Bar Plot Style**, this option has no effect.

- **Include Data Labels:** Toggles whether or not the data labels are displayed on the graph. The default is not labeled. If you choose the **Bar Plot Style**, this option has no effect.

Graph Options - Subsets

In the **Subsets** tab of the Graph Options dialog, you can specify viewing options for your subsets.



AERMOD View - Graph Options dialog - Subsets tab

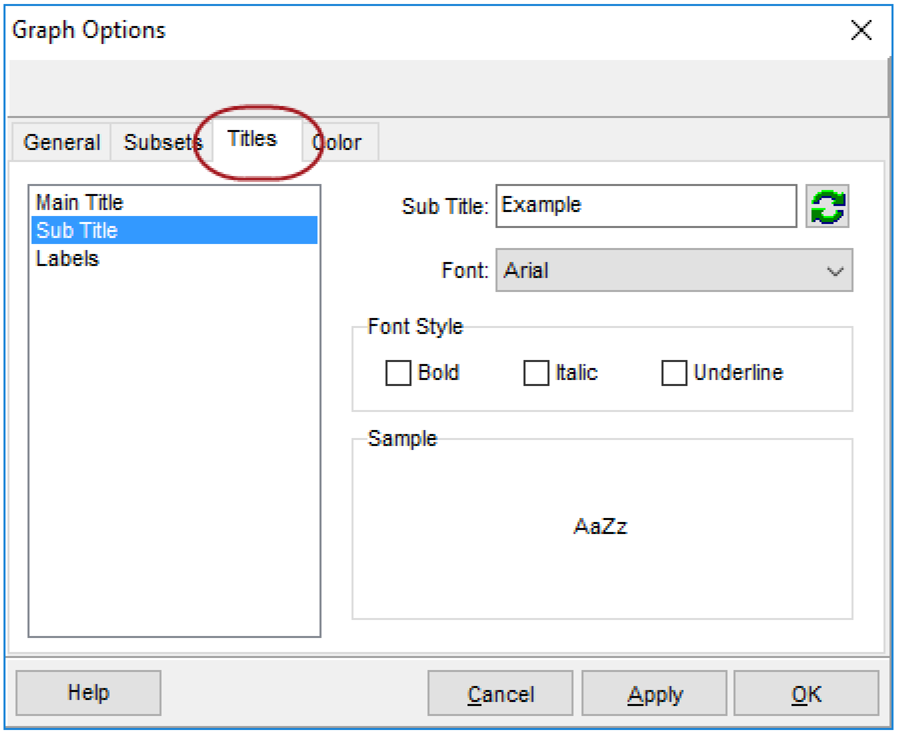
> The options found in the Subsets tab differ slightly between AERMOD View and Meteo View due to the differences in the data being plotted.

The following options are found:

- **Active:** You can chose to display or hide a series by checking/unchecking the **Active** box.
- **Subset:** Displays the name of the subset.
- **Color:** Change the point and line color by double clicking on the color bar to open the Color dialog where you can select the color of choice.
- **Point Type:** To change the point type, click in their respective cell in the table. A drop-down list appears, allowing you to select from a number of different styles.
- **Line Type:** To change the line type, click in their respective cell in the table. A drop-down list appears, allowing you to select from a number of different styles.
- **Legend Position:** Select where you would like the **Legend** displayed on the graph area.

Graph Options - Titles

The **Titles** tab of the Graph Options dialog allows you to make changes to the title, subtitle, and X and Y axis labels.



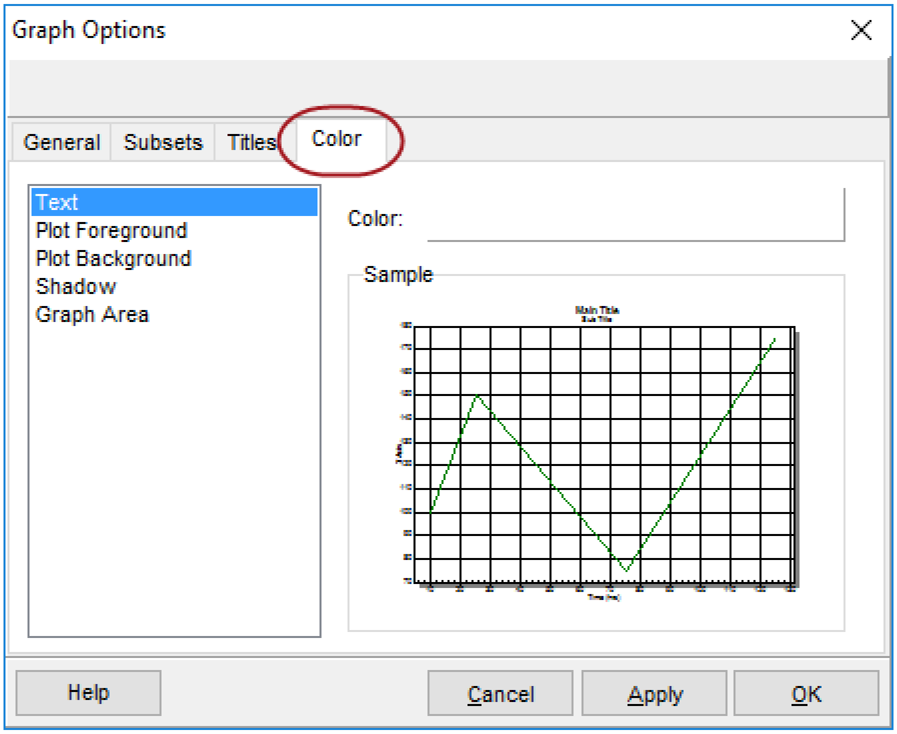
Graph Options dialog - Titles tab

The following options are available:

- **Title/Label:** Select the title or label you wish to change from the panel on the left. The default Title/Label will be displayed in the box on the right. You can change the title or if you wish to change it back to the default, click on the  button.
- **Font:** Select the font for your title/label from the drop-down list.
- **Font Style:** Select the font style for your title/label. You may apply or remove bold, italics and/or underline characteristics.
- **Sample:** Displays a sample of the font and styles you have selected.

Graph Options - Color

The **Color** tab of the Graph Options dialog allows you to change the color of graph objects.



Graph Options dialog - Color tab

How To Change the Color of a Graph:

1. Select the graph object you wish to change the color of from the list on the left.

2. Click on the color bar to display the Color dialog where you can select the color of your choice.

3. The change is displayed in the **Sample** panel.

Tools & Utilities

This section describes various tools and utilities available to you to facilitate the use of the software.

The following options are available in this section:

- Move Site
- Rotate Site
- Convert Receptor Grids to Discrete Receptors
- MAKEMET Utility
- MAXTABLE Viewer
- LEAD Post-PRocessor Utility (LEADPOST)
- Coordinate Converter
- Concentration Converter
- Concentration File Maker
- MAXIFILE Converter
- AERMET View
- RAMMET View
- WRPLOT View
- POST View

Move Site - Select Projection and Offset

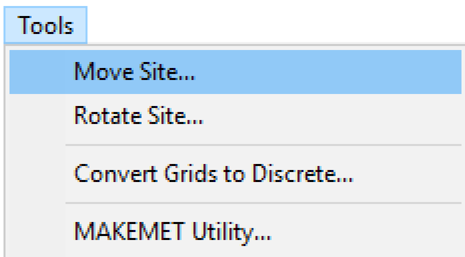
The **Move Site** function loads the **Select Projection and Offset** dialog, which allows you to move your entire site to a new location. This is especially useful if you started your project, for example, in a local Cartesian coordinate system and later on you need to switch to the UTM coordinate system. When you use the **Move Site** tool, the following objects will be automatically moved:

- Sources
- Receptors
- Annotations
- Bitmaps

Any existing contour plot files and terrain data will be permanently deleted from your project if you use the move site function, as they will no longer be valid. You will have to re-run your project to generate new contour plot files.

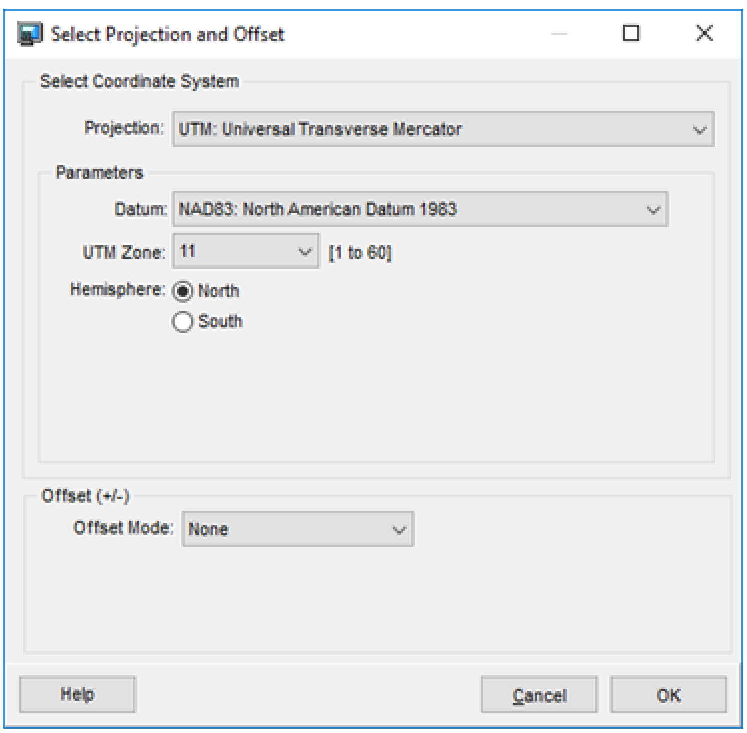
How to Move Your Site:

1. Select **Tools | Move Site...** from the menu.



Tools Menu

2. The **Select Projection and Offset...** dialog is displayed.



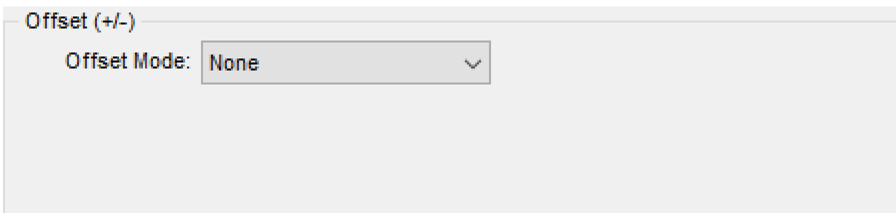
Select Projection and Offset dialog

3. Specify the **Projection** and other applicable parameters (e.g., **UTM Zone**) for the new project site.

4. In the **Offset** section, you can select one of the three **Offset Modes**, including **None**, **Single Point** and **Two Points**.

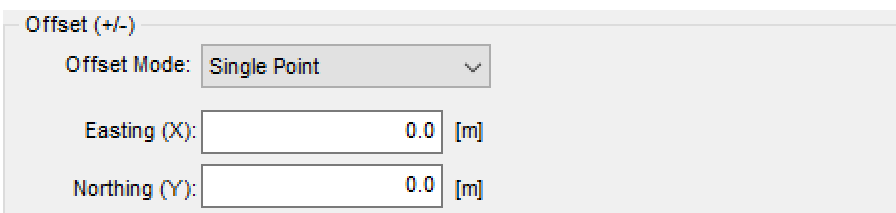
None

If **None** option is selected, the projection system will be updated to the one selected, but the site will not be moved.



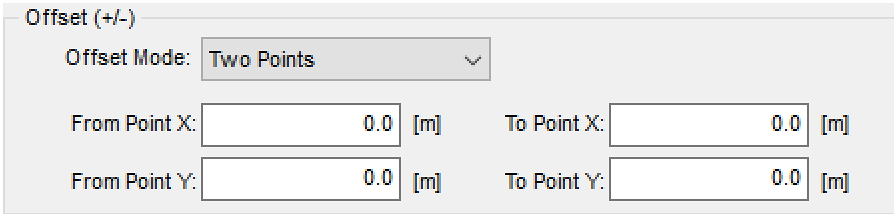
Single Point

If **Single Point** option is selected, you will be prompted to enter **Easting (X)** and **Northing (Y)** values, which will determine how far (in meters) each element of your site will be moved in the Eastern and Northern direction. If you need to move the site West and/or South, enter negative values.



Two Points

If **Two Points** option is selected, you will be prompted to enter the **From Point** and **To Point** coordinates. These need to be entered for a single element at the site (e.g. point source) and the offset will be automatically calculated and applied to all elements at the site.



5. When finished, click **OK**. The project coordinate system is then updated based on the specified options.

> Please note that base maps, which contain information on the coordinates and extents of the image, such as DXF, DLG, LULC, and Shapefiles, will not be moved to a new location when you use the **Move Site** tool. If for instance, if you have an AutoCAD DXF base map for your site in a local coordinate system, and you are moving your site to a UTM coordinate system, you will need to convert your DXF base map to the new coordinate system using the program in which the original map was created.

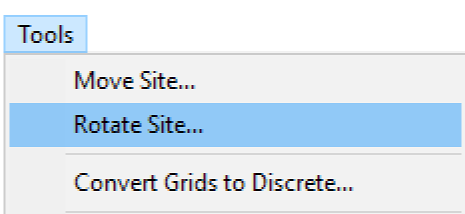
Rotate Site

The **Rotate Site** tool allows you to rotate certain objects (e.g. buildings and point sources) within your project. This tool is especially useful if the Plant North of your base map displaying your buildings does not coincide with True North. In this case, after digitizing or importing buildings from an AutoCAD (*.dxf) file, you would need to rotate your buildings in order to properly run the BPIP/ BPIP-PRIME, ISCST3, ISC-PRIME and AERMOD models. All of these models expect that your modeling objects (e.g. buildings, sources, receptors) are displayed in the drawing area with True North pointing upward.

> Any contour plotfiles will be permanently deleted from your project if you use the rotate site tool, as they will no longer be valid. You will have to re-run your project to generate new contour plot files.

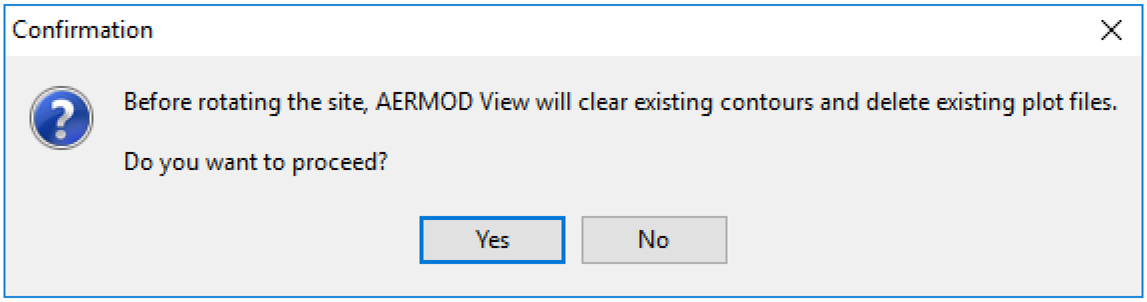
How to Rotate Your Site:

1. Select **Tools | Rotate Site...** from the menu.



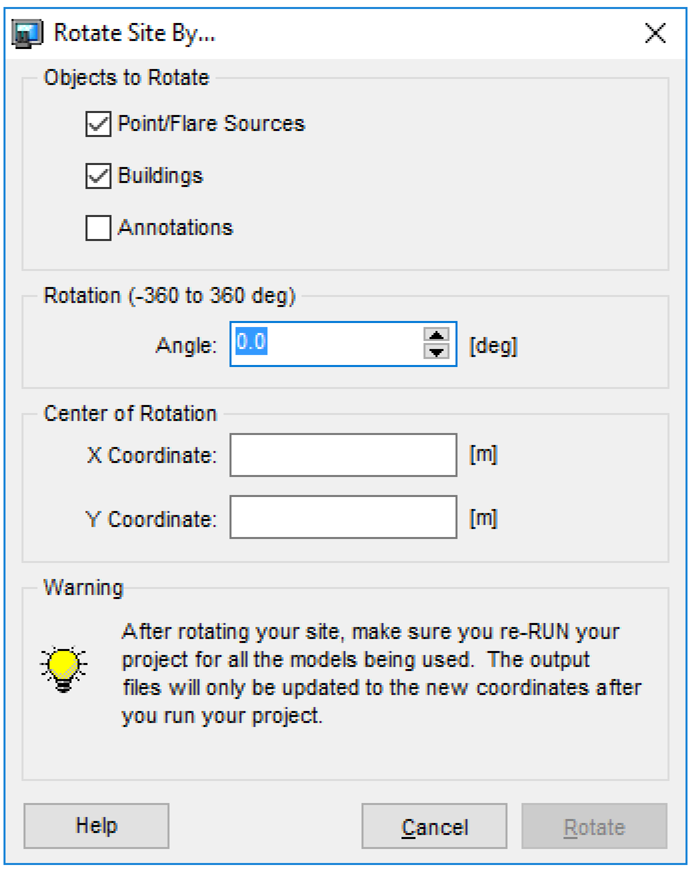
Tools Menu

2. A **Confirmation** dialog is displayed noting that, by rotating your site you will loose existing contour plotfiles. Click **Yes** if you want to proceed.



Confirmation Dialog

3. The **Rotate Site By...** dialog is displayed.



Rotate Site By... Dialog

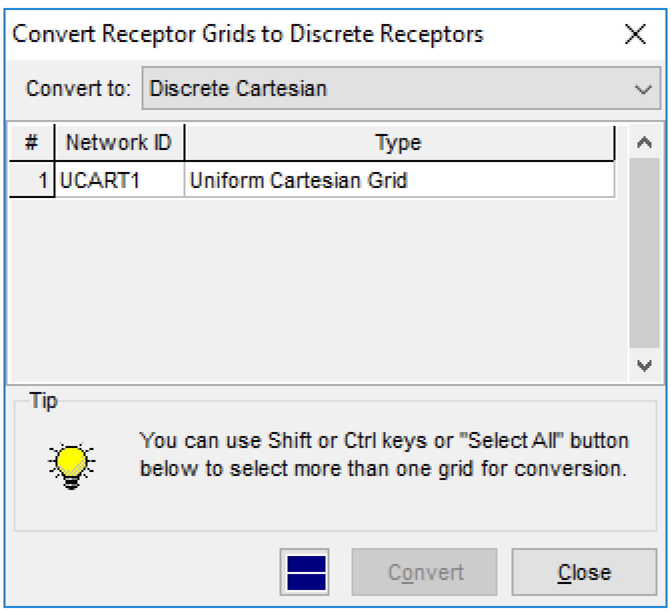
4. Specify the objects to rotate, the rotation angle in degrees and the X and Y coordinates for the center of rotation. Click on the Rotate button to rotate your site. A dialog will appear telling you that the rotation was successful. Click **OK** to exit.

> Positive rotation angles will result in counter clockwise rotations while negative rotation angles will produce clockwise rotations.

> Output files are only updated after you run your project. In order for your changes to take effect you must re-run your project after rotating your site.

Convert Receptor Grids to Descrete Receptors

This option allows you to convert any and all receptor grids in your project to discrete receptors. To use this option select **Tools - Convert Grids to Discrete...** from the main menu.



Convert Receptor Grids to Discrete Receptors dialog

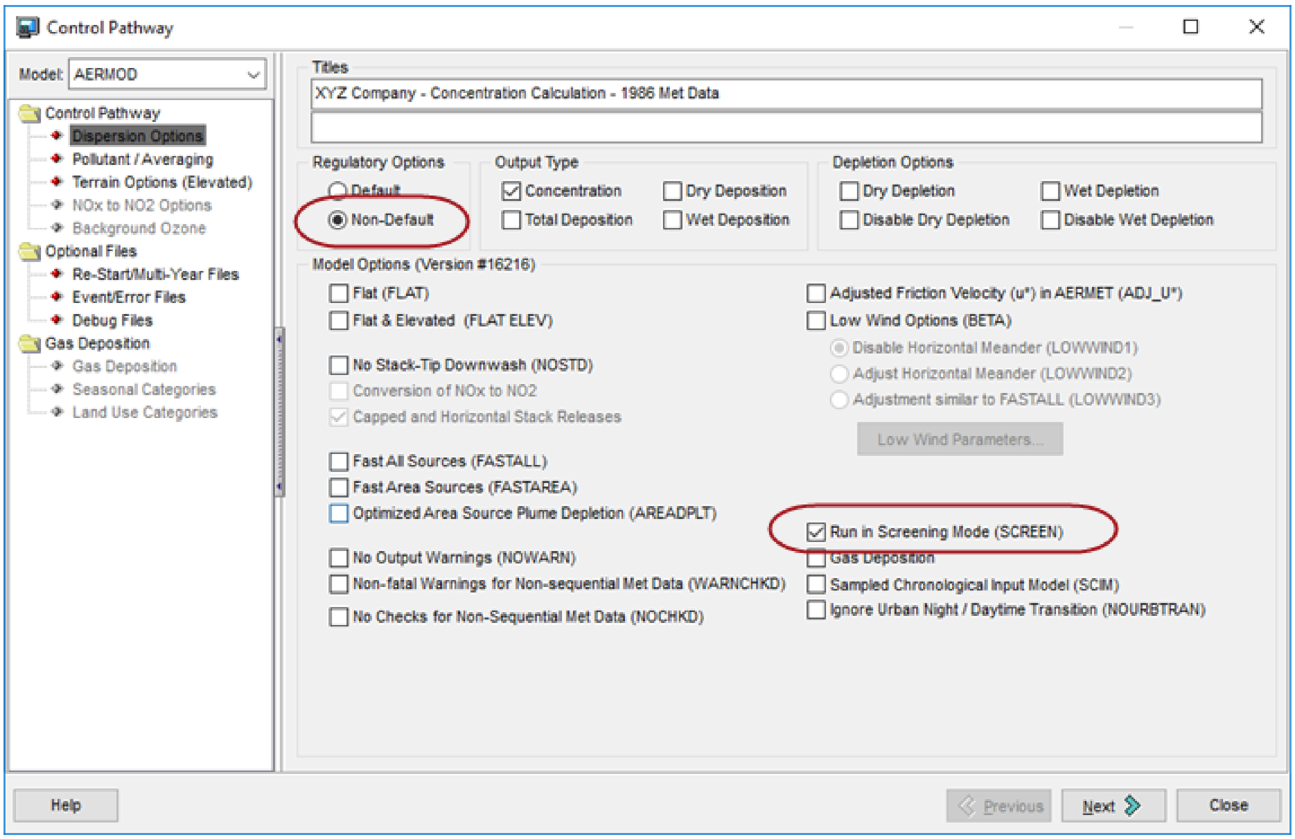
From the **Convert to** drop-down menu select the type of receptors you wish to convert your grid to. To convert multiple grids, mark them by holding down the Ctrl key and selecting them individually, or hold down the Shift key and click on the first and last grid of the range you wish to convert. When you have selected the desired grids, click the **Convert** button to finish the procedure.

MAKEMET Utility

The **MAKEMET Utility** allows you to easily specify the parameters necessary to run the US EPA MAKEMET program. The US EPA MAKEMET program generates a matrix of meteorological conditions, in the form of AERMET-ready surface (*.SFC) and profile (*.PFL) files, based on user-specified surface characteristics, ambient temperatures, minimum wind speed, and anemometer height.

You have access to this utility by selecting **Tools | MAKEMET Utility...** from the menu.

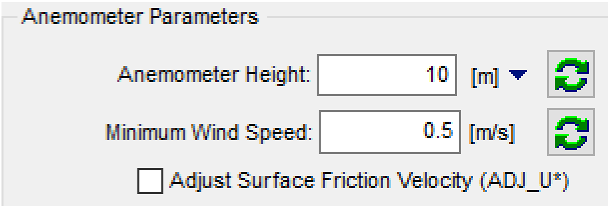
The screening met files generated by MAKEMET (*.SFC and *.PFL) can be used to run the AERMOD model in screening mode by selecting in AERMOD View, the **Run in Screening Mode** option available under the **Non-Default Options** section of the **Control Pathway** window.



AERMOD View - Control Pathway window

The following are the parameters that need to be specified under the MAKEMET Utility:

Anemometer Parameters

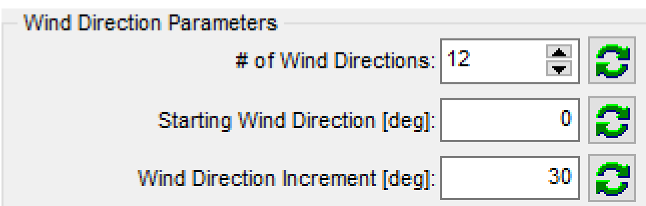


- **Anemometer Height [m]:** The suggested default value for the anemometer height is 10 meters. However, the user can specify any other value.

- **Minimum Wind Speed [m/s]:** The suggested default value for the minimum wind speed is 0.5m/s. However, the user can specify any other value.

- **Adjust Surface Friction Velocity:** You can select this option in order to adjust the surface friction velocity used in the utility.

Wind Direction Parameters



MAKEMET allows the user to specify a single wind direction or a range of wind directions for the meteorological matrix. This optn may be useful for applications involving building downwash to ensure that building dimensions for all sectors are included in the screening analysis.

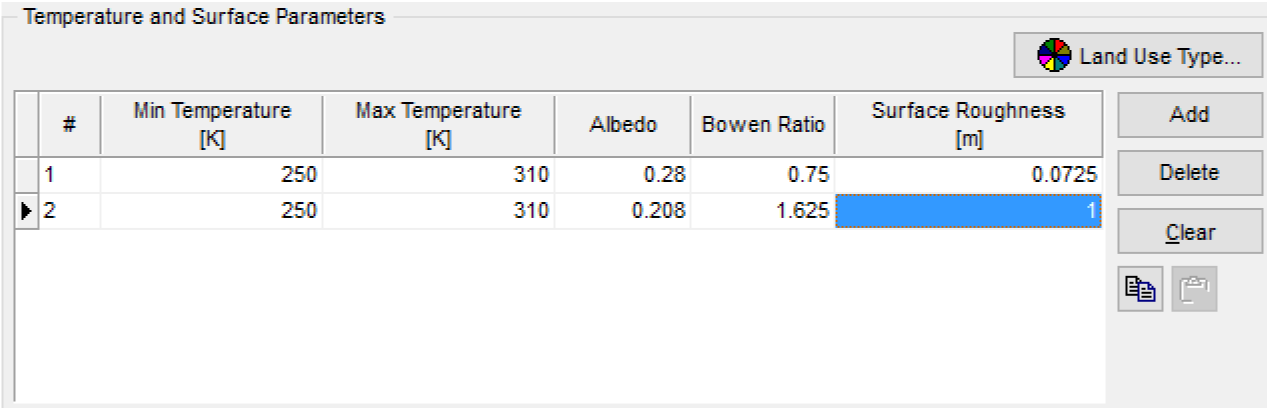
- **# of Wind Directions:** this parameter represents how many directions you want to specify.

- **Starting Wind Direction [deg]:** specify here the initial wind direction angle.

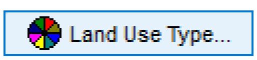
- **Wind Direction Increment [deg]:** specify the wind direction increment in case more than one wind direction was specified

Temperature and Surface Parameters

MAKEMET also allows the user to specify more than one set of ambient temperatures and surface characteristics. MAKEMET will concatenate the resulting meteorological matrices into single surface and profile files.

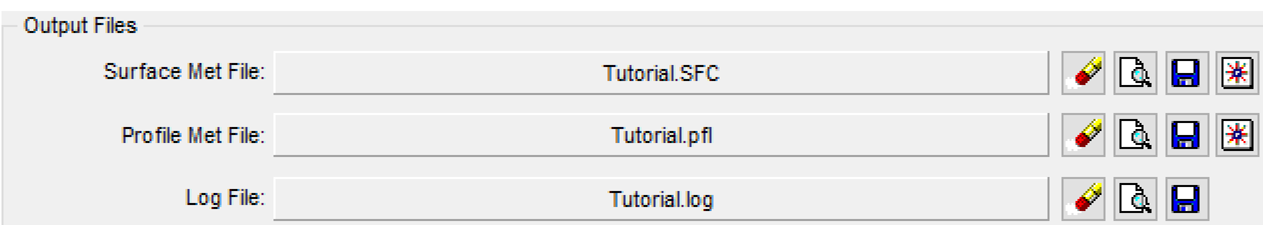


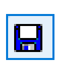
- **Min Temperature [K]:** Suggested default value for minimum ambient temperature is 250 deg Kelvin.
- **Max Temperature [K]:** Suggested default value for maximum temperature is 310 deg Kelvin.
- **Albedo:** Defined as the fraction of the incoming solar radiation that is reflected from the ground without absorption when the sun is directly overhead.
- **Bowen Ratio:** Defined as a measure of the amount of moisture at the surface.
- **Surface Roughness [m]:** Defined as the height at which the mean horizontal wind speed approaches zero.



Press this button to select suggested surface parameters for several land use types.

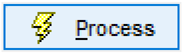
Output Files



- **Surface Met File:** Press the  button and specify the name and location where you want the Surface file to be generated.
- **Profile Met File:** Once you specify the name of the Surface file, the name and location of the Profile file will be automatically assigned for you.

- **Log File:** This file will be created by MAKEMET and it contains information on the input parameters specified for generating the Surface and Profile met files.

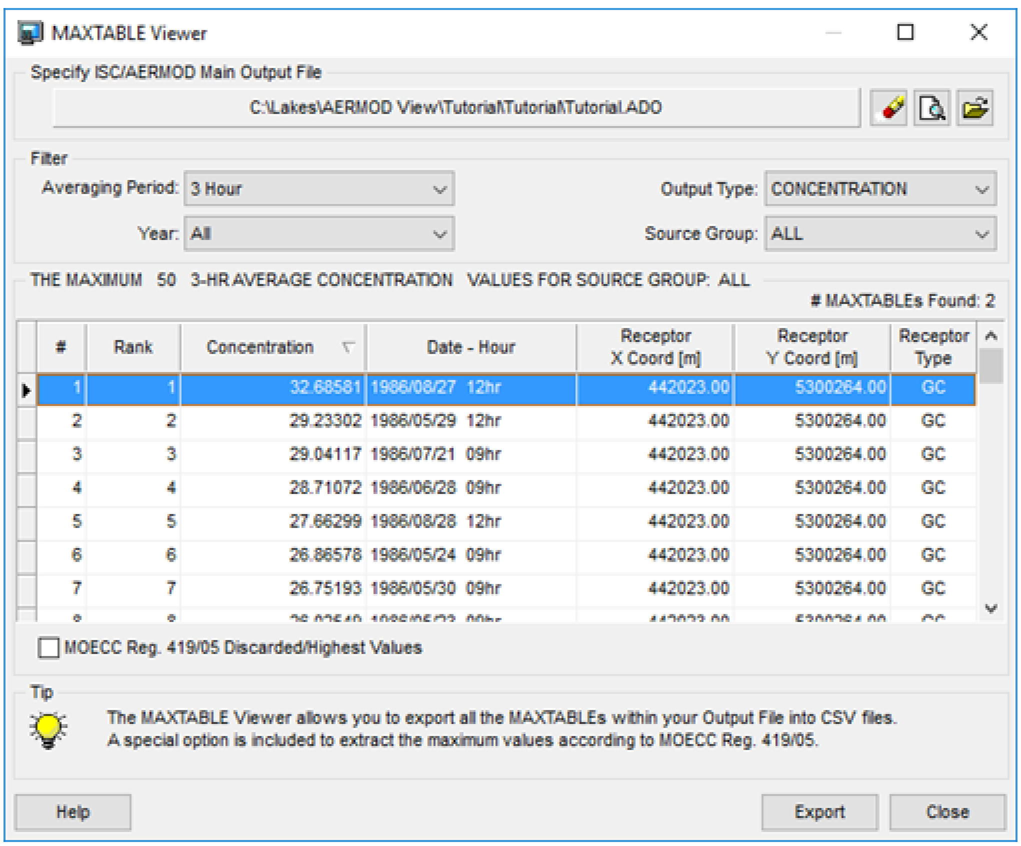
Processing MAKEMET



Press this button to process your input data using the US EPA MAKEMET program and generate the screening met files (surface and profile).

MAXTABLE Viewer

The **MAXTABLE Viewer** reads your AERMOD Main Output file (*.ado) or ISC Main Output file (*.out or *.pou) and displays all the available Maximum Value Tables within your output file allowing you to better analyze your results. You have access to this utility by selecting **Tools | MAXTABLE Viewer...** from the menu.



MAXTABLE Viewer

The following options are available:

- **Specify ISC/AERMOD Main Output File:** If you have already run your project and an output file is available, it will be automatically loaded into the MAXTABLE Viewer. However, you can click on the button to specify any output file you wish to view. If your output does not contain Maximum Values Tables, the output file will not be opened.

- **Averaging Period:** You can select from the drop-down list the averaging period you wish to view.

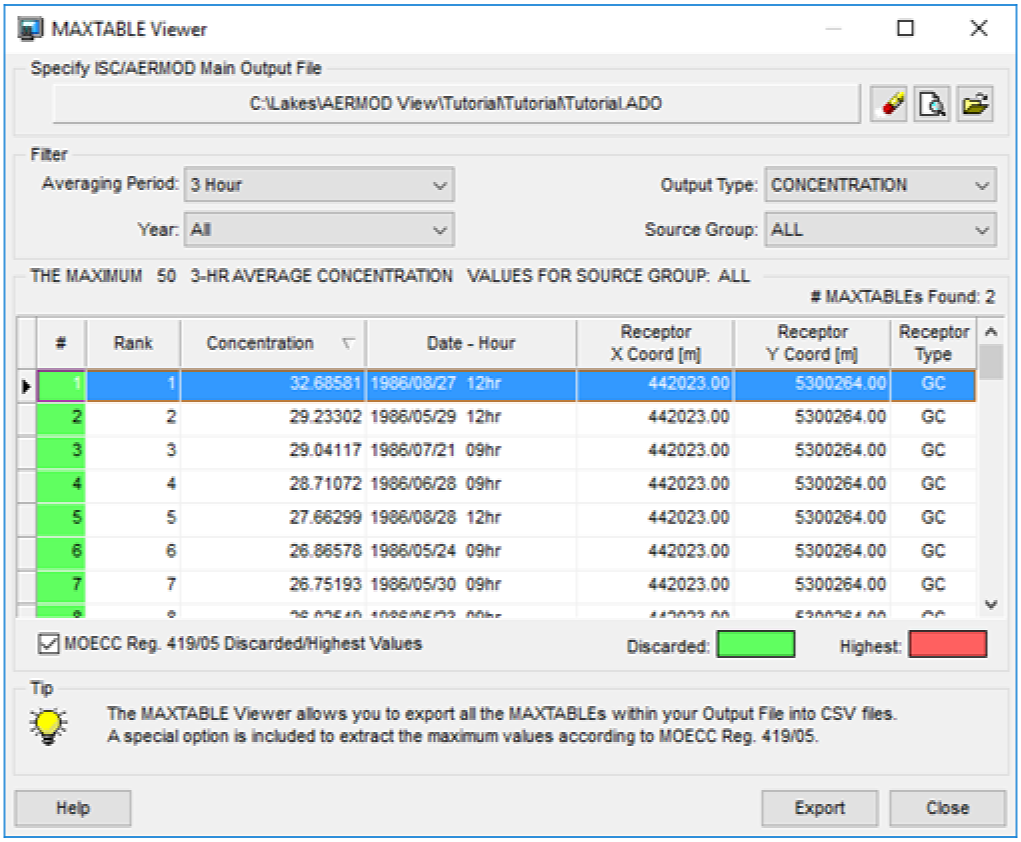
- **Year:** Select from the drop-down list the year you wish to view.

- **Output Type:** Select from the drop-down list the output type you wish to view.

- **Source Group:** If you have configured Source Groups, select from the drop-down list the source group you wish to view.

- **Display Window:** The results of your selection are displayed in a tabular format. It is possible to sort the data by Rank, Concentration or Date/Hour simply by clicking on the column header.

- **MOECC Reg. 419/05 Discarded/Highest Values:** This option was developed according to the *Air Dispersion Modelling Guideline for Ontario* (ADMGO), Canada, which states that the eight hours with the highest 1-hour average predicted concentrations in each single met year may be discarded. With a five year met file, this means that the Ontario MOE will consider for compliance assessment the highest concentration after the elimination of these forty highest hours over the five year period from the modelling results. Note that repeat listings of the same hour should be treated as one hour eliminated. When this option is checked, the highest (discarded) 8 hours are highlighted in green and the final accepted value highlighted in red.



MAXTABLE Viewer - MOECC Reg. 419/05 option

Click on this button to export the MAXTABLE results to a CSV file that can be opened in Excel.

LEAD Post-Processor Utility (LEADPOST)

The US EPA LEADPOST program was designed to read monthly concentration output from the US EPA AERMOD model or an alternative model and **calculate rolling 3-month averages** by receptor and source group.

The program will calculate:

- Maximum rolling 3-month average concentration for each receptor and source group
- Overall maximum concentration (across all receptors and source groups).

> For more information on how to generate POSTFILEs required for use in LEAD Post-Processor Utility, see LEAD Modeling Analysis.

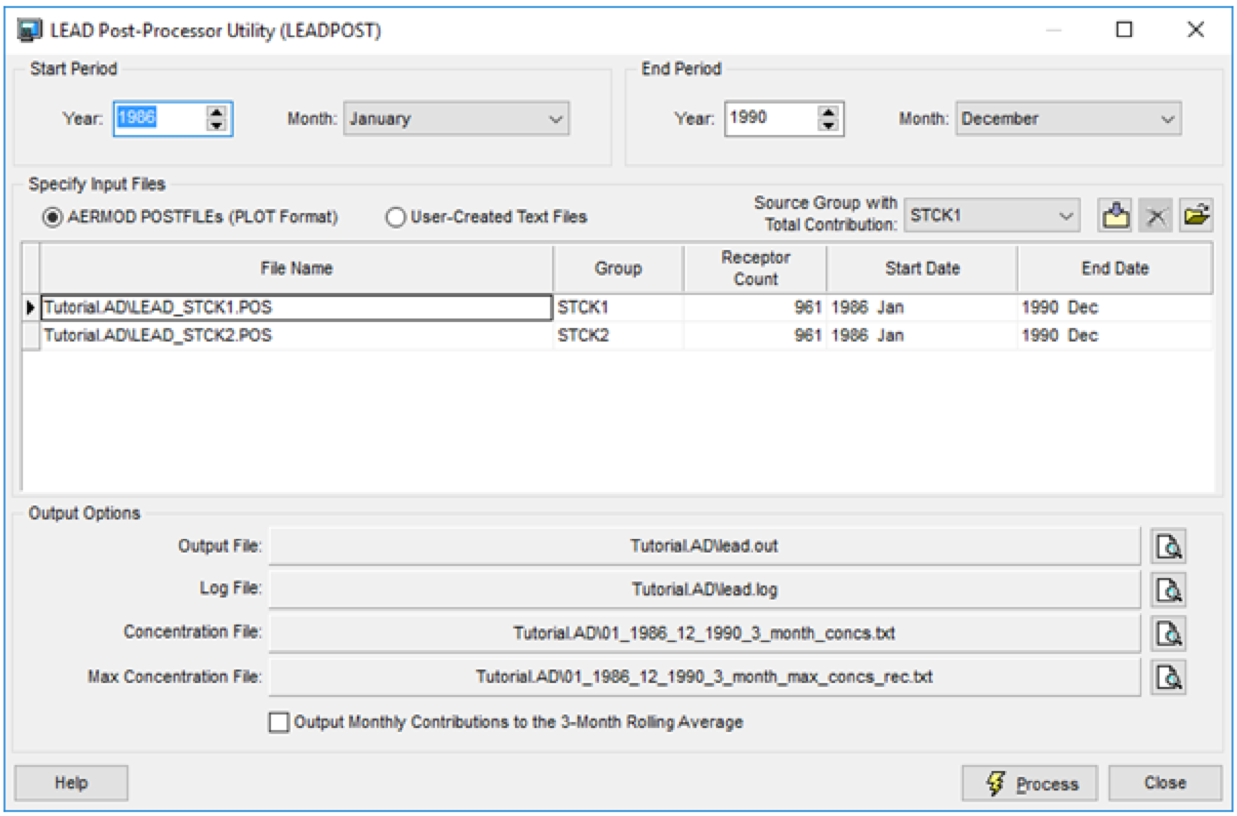
How to Use the LEAD Post-Processor Utility (LEADPOST):

1. Press the button and specify the monthly average POSTFILEs (PLOT Format) that were generated by the AERMOD model for your LEAD air dispersion modeling project. Information read from your POSTFILEs will be automatically displayed in the table.

2. Verify if the **Start Period** and **End Period** are correct.

3. Press the **Process** button. After processing finished, press the button to view the output file generated by LEADPOST. The **Concentration File** is the one that is suitable for contour plotting.

4. Press the **Close** button. The LEAD rolling 3-month average plotfile will be automatically loaded and displayed in the AERMOD View graphical area.



Start and End Period

- **Start Period:** The start year and month for the processing must be specified here.

- **End Period:** The end year and month for the processing must be specified here.

Specify Input Files

Press this button to automatically load POSTFILEs from your currently opened project.

Press this button after selecting one or more POSTFILEs to be removed from the List of files for processing.

Press this button to specify POSTFILEs or user-created text files for processing.

- **AERMOD POSTFILEs (PLOT Format):** Only AERMOD POSTFILEs with a monthly averaging period and in PLOT (ASCII) format are accepted.

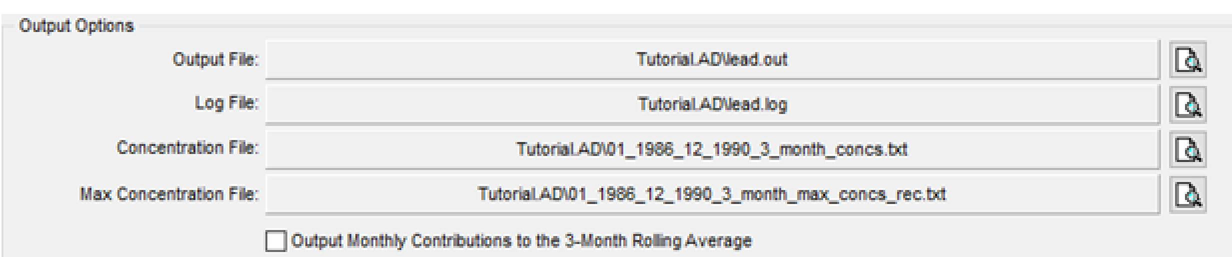
- **User-Created Text Files:** LEADPOST can also read user-created simple text files (see LEADPOST User-Created Text Files Format in Appendix: File Formats).

- **Source Group with Total Contribution:** If Source Group ALL is not present in the specified POSTFILES, then you will need to specify the Source Group ID that represents the group with total contribution.

> LEADPOST checks for the existence of source group "ALL", which is the source group with total contribution of all sources to a receptor concentration. If LEADPOST does not find the source group "ALL", the user will be notified and prompted for a source group that represents the total contribution from all sources. The entered group ID will be compared against the list of source groups from the POSTFILEs and if it is not in the list, the user is notified that the concentrations for the entered group will be calculated. If the entered group is to be calculated, LEADPOST assumes the individual source groups are mutually exclusive, i.e. an emission source is assigned to only one source group only.

> Receptors in all POSTFILEs must be the same and in the same order for each month and source group.

Output Files



- **Output File (lead.out):** This file contains a summary of the highest overall maximum 3-month rolling average concentration. Shown are the month and year, receptor coordinates, receptor elevation, receptor hill height scale, and receptor flagpole height of the maximum rolling average. Also shown, in descending order by concentration, are the rolling averages for the individual source groups at the same receptor and same month.

- **Log File (lead.log):** During execution, LEADPOST writes information to this file.

- **Concentration File (MM_YYYY_MM_YYYY_3_month_concs.txt):** This file contains the rolling 3-month averages for all receptors and source groups for each month. Concentration contours for this file can be visualized in AERMOD View main graphical area.

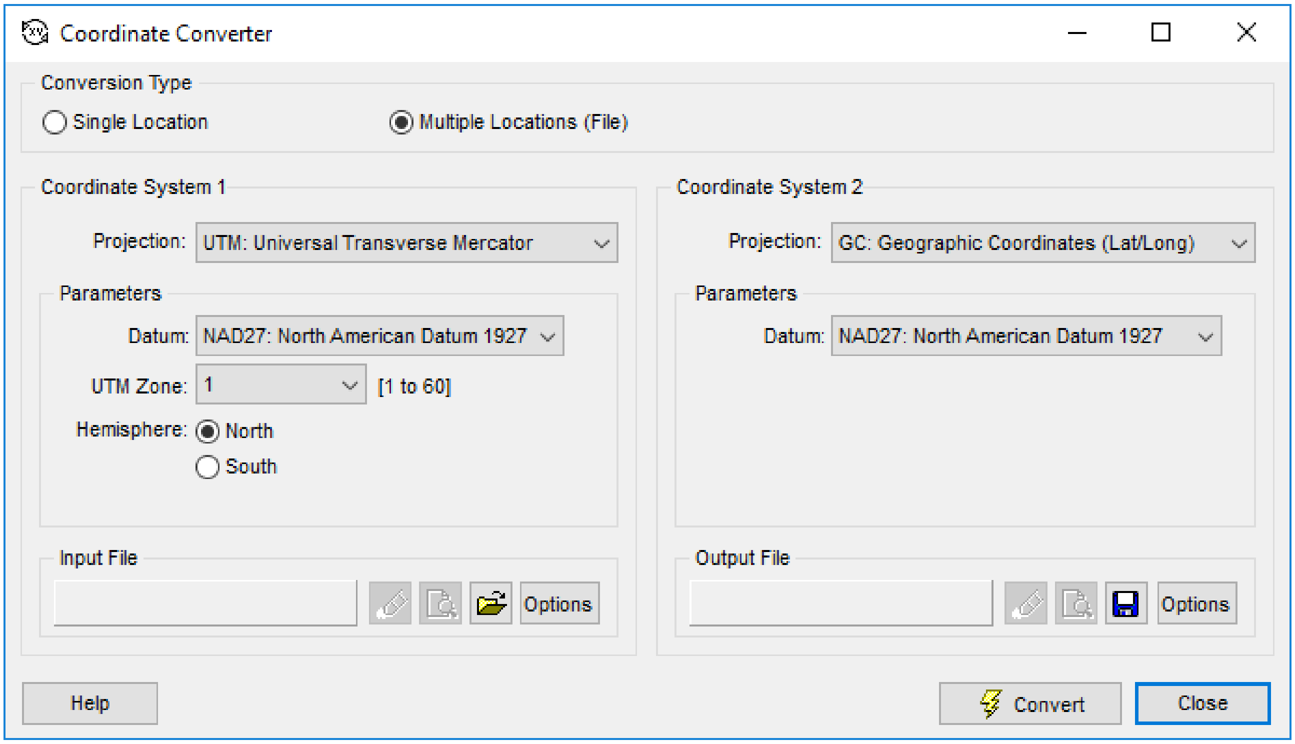
- **Max Concentration File (MM_YYYY_MM_YYYY_3_month_max_concs.txt):** This file contains the maximum 3-month concentrations by receptor.

- **Output Monthly Contributions to the 3-Month Rolling Average** - checking this option will include the monthly concentrations used to calculate 3-month rolling averages in the .TXT output files.

> **Warning:** LEADPOST assumes that concentrations are in micrograms per cubic meter. If concentrations are in units other than micrograms per cubic meter, concentration results from LEADPOST should be converted to micrograms per cubic meter to compare against the lead NAAQS standard. LEADPOST will also check the averaging period in the POSTFILE.

Coordinate Converter

The **Coordinate Converter** utility allows you to convert between different coordinate systems. You have access to this utility by selecting **Tools | Coordinate Converter...** from the menu.



Coordinate Converter utility

How to Convert between Coordinate Systems:

1. In the **Conversion Type** panel you can specify whether you want to convert a single location or multiple locations.

2. In the **Coordinate System 1** panel, select the **Projection** for the project from the drop down list.

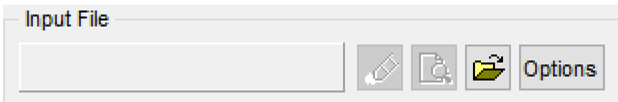
See: Map Projections

3. Complete the fields in the **Parameters** section, which displays the appropriate options for the Projection option that you have selected.

See: Datum Options

4. If you have selected to convert a single location, the Coordinates panel will be displayed. Here you must specify the coordinates for conversion.

5. If you have selected to convert multiple locations you can specify the input file, in .csv format, containing all the locations to be converted. Click on the button to specify the file.



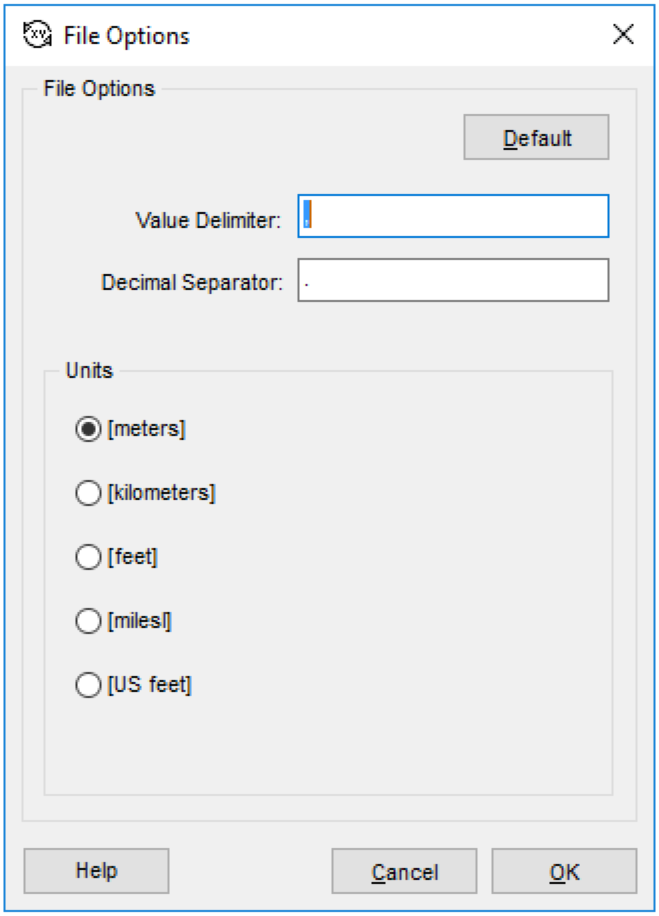
Click on the Options button to open the File Options dialog from where you can specify certain file options for the input file.

6. Repeat **Steps 2, 3** and **4** in the **Coordinate System 2** panel. If you have selected to convert multiple locations you will need to specify a location to save the output file which will contain all the converted coordinates. Click on the button to specify a location.

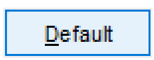
7. Click on the Convert button to convert your coordinates.

File Options

The **File Options** dialog is accessed by clicking on the Options button in the **Coordinate Converter** dialog. In this dialog you can identify the options used in you input file. This allows the file to be read easily and the conversion to be completed successfully.



File Options

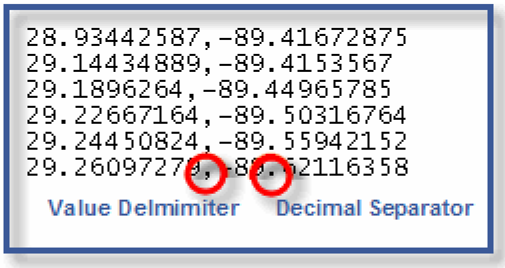
At any time, you can click on the 

button to return to the default settings. The following options are available:

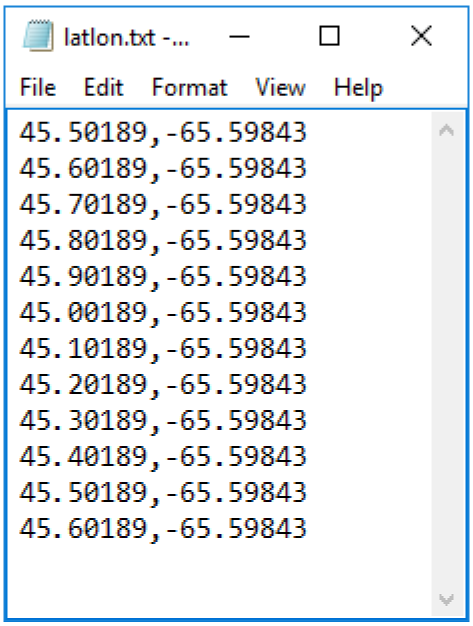
- **Value Delimiter:** A delimiter is a character or sequence of characters used to specify the boundary between separate, independent pieces of data. An example of a delimiter is the comma character, which acts as a field delimiter in a sequence of comma-separated values. The comma (,) is used as the default delimiter, but this can be changed to reflect what is being used in the input file.

- **Decimal Separator:** The decimal separator is a symbol used to mark the boundary between the integral and the fractional parts of a decimal numeral. The decimal point (.) is used as the default value, but this can be changed to reflect what is being used in the input file.

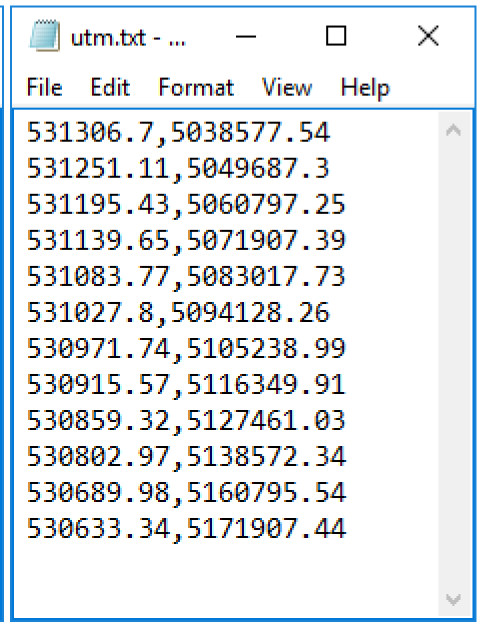
- **Units:** In this panel you can specify the units of the coordinates in the file.



See below two sample input files, one in Lat/Long and one in UTM Coordinates:



45.50189,-65.59843
45.60189,-65.59843
45.70189,-65.59843
45.80189,-65.59843
45.90189,-65.59843
45.00189,-65.59843
45.10189,-65.59843
45.20189,-65.59843
45.30189,-65.59843
45.40189,-65.59843
45.50189,-65.59843
45.60189,-65.59843



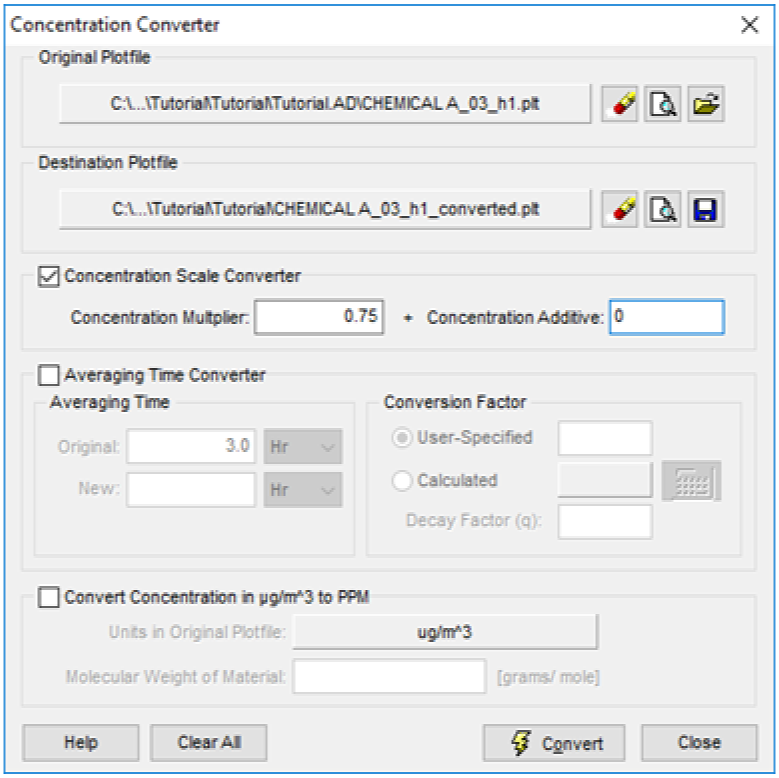
531306.7,5038577.54
531251.11,5049687.3
531195.43,5060797.25
531139.65,5071907.39
531083.77,5083017.73
531027.8,5094128.26
530971.74,5105238.99
530915.57,5116349.91
530859.32,5127461.03
530802.97,5138572.34
530689.98,5160795.54
530633.34,5171907.44

Concentration Converter

The **Concentration Converter** is a utility that allows you to generate new plotfiles from existing ones without having to re-run the ISCST3/AERMOD models. This utility allows you to apply a multiplier or an additive to the concentration values, change the averaging period, or change the concentration units to parts per million (ppm).

> The **Concentration Converter** utility can only be used for concentration values, not for deposition.

You have access to this dialog by selecting **Tools | Concentration Converter...** from the menu.



Concentration Converter

The following options are available:

Original Plotfile

Specify here the plotfile you wish to apply the changes to. Click on the  button to specify the name and location of the plotfile you wish to use. If a plotfile is currently being displayed in the AERMOD View window, by default the name and path of this open plotfile will be displayed in this panel.

Destination Plotfile

Here you must specify a name and location for the new plotfile that will be created after the conversion has been applied to the original plotfile. Click on the  button to specify where you would like to save the file and under what name.

Concentration Scale Converter

Check the **Concentration Scale Converter** box if you would like to apply a multiplicative or additive factor to the concentration values. This can be useful, for example, when you want to take into account background concentrations. Simply specify a value for the concentration multiplier and concentration additive.

Averaging Time Converter

Check the **Averaging Time Converter** box if you wish to specify a different averaging period for the selected plotfile. You must then specify the following:

- **Original Averaging Time:** Specify here the averaging period for your original plotfile in either hours or minutes. Your plotfile will be in one of the following averaging periods:

Averaging Period	Total No. of Minutes
1 hour	60 min
2 hours	120 min
3 hours	180 min
4 hours	240 min
6 hours	360 min
8 hours	480 min
12 hours	720 min
24 hours	1,440 min
Month	43,200 min
Annual	525,600 min
Period	Dependent on total number of years of met data

- **New Averaging Time:** Enter here the averaging period you want your plotfile to be converted to. The new averaging time can be given in either hours or minutes.

- **Conversion Factor:** You will need to either specify or calculate a conversion factor upon which the averaging time conversion will be based.
- **User Specified:** Specify in the text box a value for the conversion factor.
- **Calculated:** In the **Decay Factor** text box specify the decay factor to be used and click on the button to calculate the conversion factor. The calculated value will be displayed in the panel. The decay factor can be from 0.17 to 0.20, recommended in some jurisdictions to be 0.28. This is applied to the following equation (Wark, K. and C. Warner, 1981):

$$C_{new} = C_{old} * (T_{old}/T_{new})^q$$

where:

C_{new} = desired concentration

C_{old} = original concentration

T_{new} = new averaging time

T_{old} = original averaging time

q = decay factor

Convert Concentration in Micrograms per Cubic Meter to PPM

At some point, you may wish to convert concentrations in micrograms per cubic meter (ug/m3) to concentrations in parts per million (ppm) or parts per billion (ppb). For example, the ISC and AERMOD models generate output in ug/m3 while you may need results in ppm.

The conversion can be achieved using the following equations:

C_{ppb} = C µg/m3x 24.45/MW

C_{ppm} = C µg/m3 x 0.02445/MW

where:

C = Concentration of the pollutant

MW = Molecular weight of pollutant

24.45 = Constant representing the volume, in liters, of one mole of a gas at standard temperature (25°C) and pressure of 1 atm (101.3kPa).

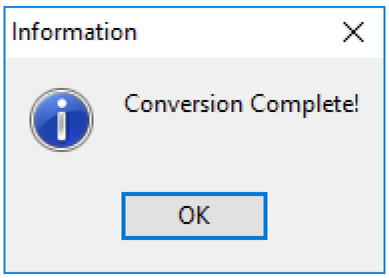
C_{ppm} = 100 x 0.02445/64

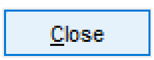
C_{ppm} = 0.0382 ppm = 38.2 ppb

AERMOD View is capable of performing the conversion of your plotfile concentration results from micrograms per cubic meter (ug/m3) to parts per million (ppm). Simply go to Tools | Concentration Converter and select Convert Concentrations in ug/m3 to PPM. The units in the original plotfile will be displayed in the panel, allowing you to verify whether this conversion is possible. All you need to do is specify the molecular weight (MW) in grams per mole of the specific pollutant you are modeling and click Convert. This will generate a new plotfile with concentrations in ppm.

After you have specified your conversion options you must press the 

button in order to generate the new plotfile. A message will be displayed indicating the conversion was successful.



Click on the 

button to close the **Concentration Converter**. You can then open and view your new plotfile.

> The conversion done by AERMOD View assumes a temperature of 25°C and 1 atmosphere pressure. If you want to make a conversion using specific temperature and pressure values, you will need to do this outside the interface using the following equations:

C_{ppb} = C µg/m3 x (MV/MW)

MV = 22.4 x (T/273) x (101.3/P)

where:

C = Concentration of the pollutant

MV = Molecular volume

MW = Molecular weight of pollutant

22.4 = This is the number of liters in one mole of an ideal gas at 0° Celsius (273° Kelvin).

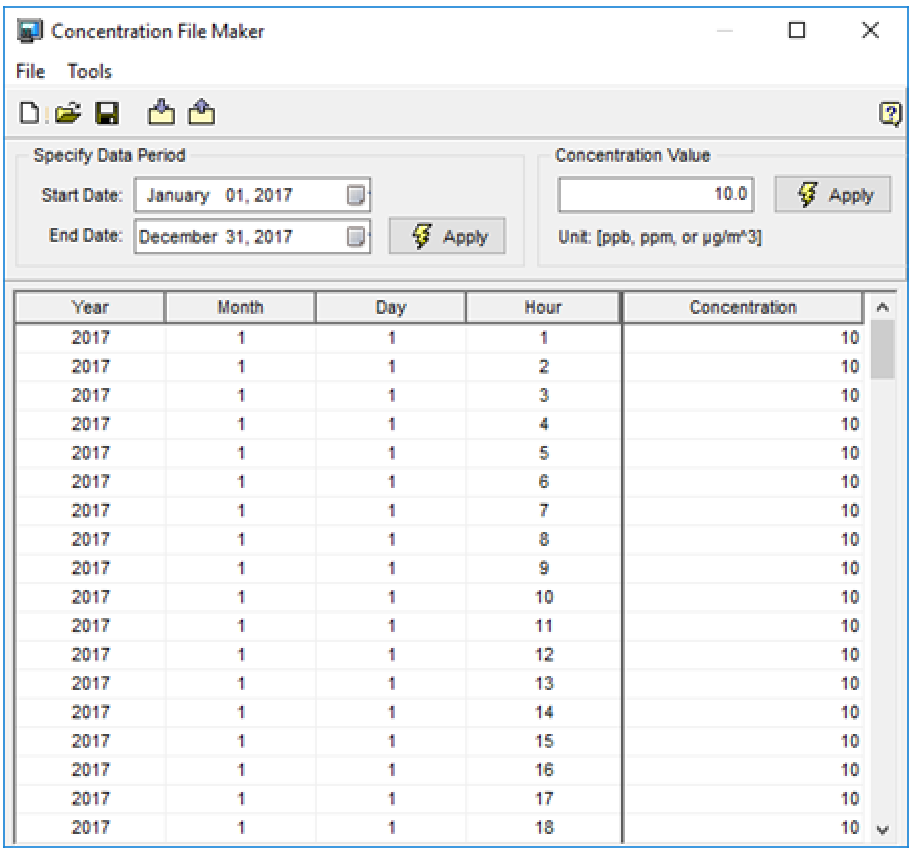
T = Temperature in Kelvin

P = Pressure in Kilopascal (kPa)

Please be aware that the values at each receptor in a plotfile can potentially be from different elevations and times. Thus it is necessary to take care when choosing temperature and pressure values.

Concentration File Maker

The **Concentration File Maker** utility allows you to create background concentration data files (e.g, ozone, SO2, NO2, etc). You have access to this utility by selecting **Tools | Concentration File Maker...** from the menu or by clicking on the File Maker... button in the NOx to NO2 Options screen of the Control Pathway.



Concentration File Maker Utility

How to Use the Concentration File Maker:

1. From the **Specify Data Period** panel, select the **Start Date** and **End Date** for the file. Click on the Apply button. The specified data period will be displayed in the window.

2. Highlight the hour rows to which you wish to apply identical concentrations.

3. In the **Concentration Value** panel, specify the concentration value. Click on the Apply button to apply this value to the selected rows (hours).

4. Repeat steps 2 and 3, if entering different values.

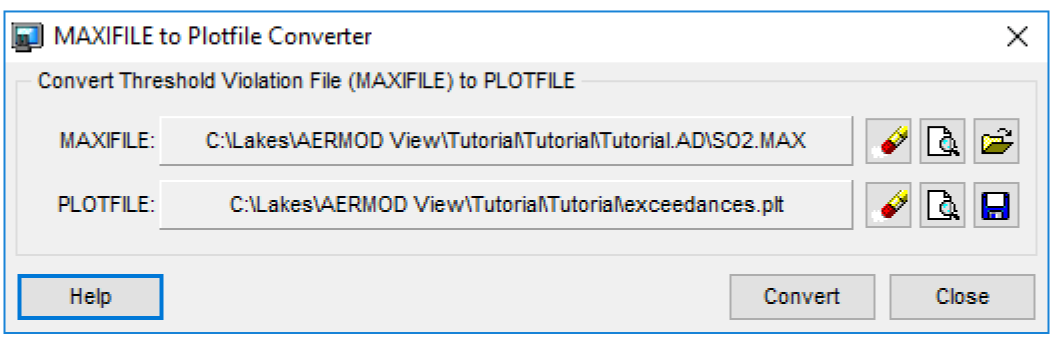
> You may select multiple rows in sequence by pressing down the Shift key or press the Ctrl key to make disjoint selections. To select all rows, press the Shift key + screen Down + Ctrl + End.

> A missing indicator of -99 will be applied to any hours for which you have not specified a concentration value.

5. Save the file by selecting **File | Save As...** from the menu or by clicking on the button. The **Save As** dialog will open allowing you to specify a name and location to save the concentration file (*.dat). You may now use this file in the NOx to NO2 Options screen of the Control Pathway if you wish.

MAXIFILE Converter

The **MAXIFILE Converter** allows you to convert any Threshold Violation Files (MAXFILEs) from any external projects into PLOTFILES. Plotfiles created from MAXIFILES contain information on the number of times a specified threshold was exceeded at each receptor location.



MAXIFILE Converter

How to Convert Threshold Violation File (MAXIFILE) into PLOTFILE:

1. Select the MAXIFILE you wish to convert.

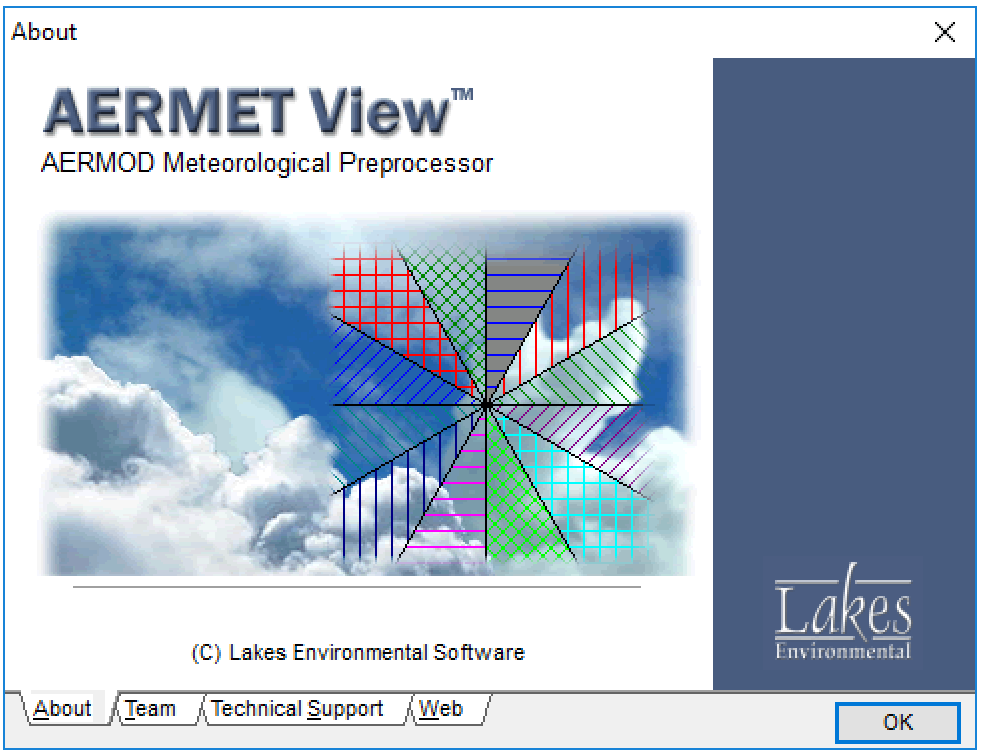
2. Specify the file name and location to save the plotfile.

3. Click on the 

button to convert the MAXIFILE to a plotfile.

AERMET View

AERMET View is the **Lakes Environmental** interface for the US EPA AERMOD meteorological preprocessor, AERMET. The AERMET program pre-processes meteorological data into a format suitable for use by the AERMOD air dispersion model. **AERMET View** guides you through easy steps to prepare your onsite and offsite meteorological data for use with AERMOD.



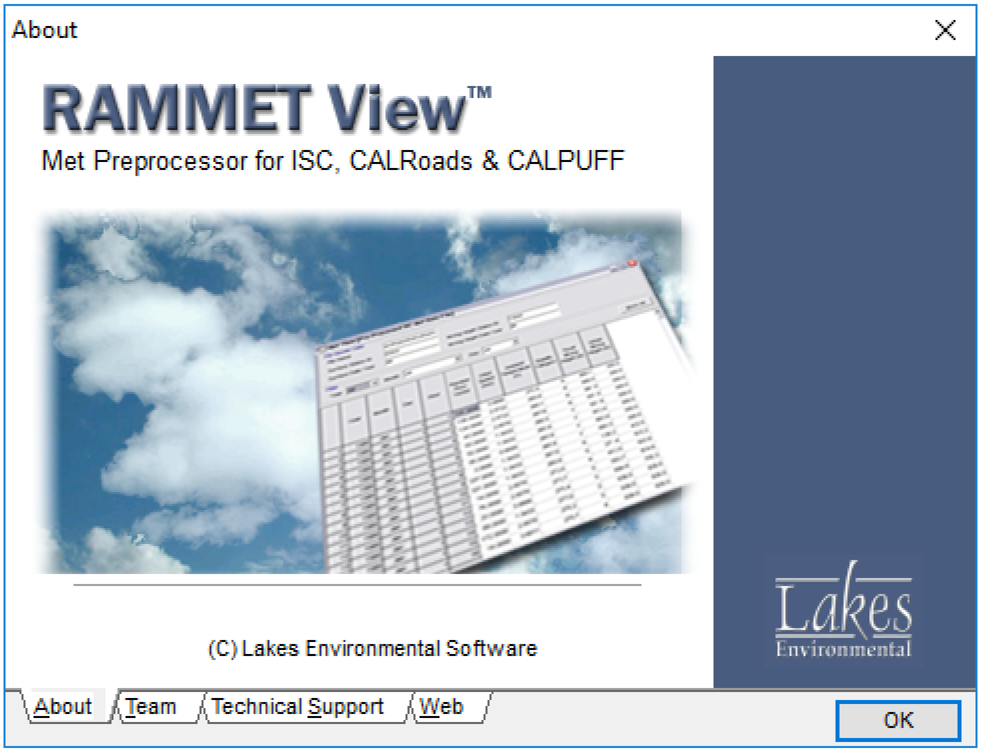
AERMET View will ease the process of pre-processing meteorological data. The interface is full of tips that will guide you through the steps to prepare your met data for use with AERMOD.

AERMET View will free you from buying expensive pre-processed met data from other companies and its easy-to-use utilities also allow you to convert and import non-standard meteorological data formats. Excel files can be imported and a **File Maker** utility guides you through creating your own meteorological data files.

> For more information about this product please refer to **AERMET View** online help.

RAMMET View

RAMMET View is the **Lakes Environmental** interface for the U.S. EPA meteorological preprocessor, PCRAMMET. PCRAMMET is the meteorological pre-processor used for preparing met data for use in the EPA's short term air quality dispersion models such as ISCST3, ISC-PRIME, and CAL3QHCR.

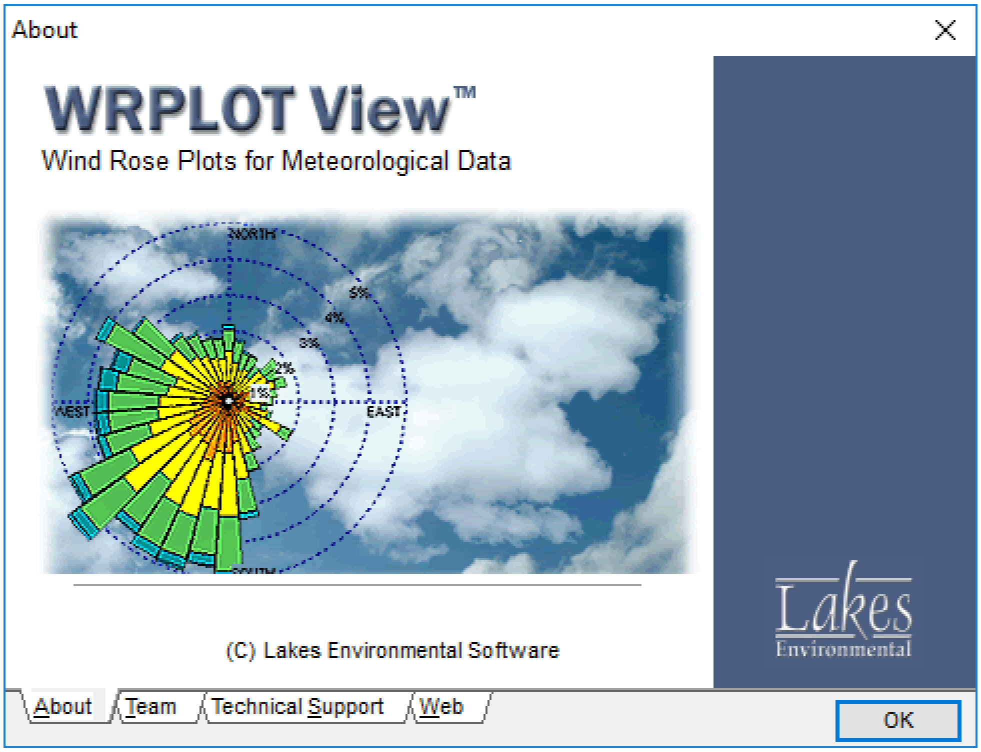


The **RAMMET View** interface will ease the process of pre-processing meteorological data. The interface is full of tips that will guide you through the steps to prepare your met data for use with ISCST3, ISC-PRIME, or CAL3QHCR models. **RAMMET View** will free you from buying expensive pre-processed met data from other companies and its easy-to-use utilities also allow you to convert and import non-standard meteorological data formats. Excel files can be imported and a File Maker utility guides you through creating your own meteorological data files.

> For more information about this product please refer to **RAMMET View** online help.

WRPLOT View

WRPLOT View is a Windows program that generates wind rose statistics and plots for several meteorological data formats. A wind rose depicts the frequency of occurrence of winds in each of the specified wind direction sectors and wind speed classes for a given location and time period.

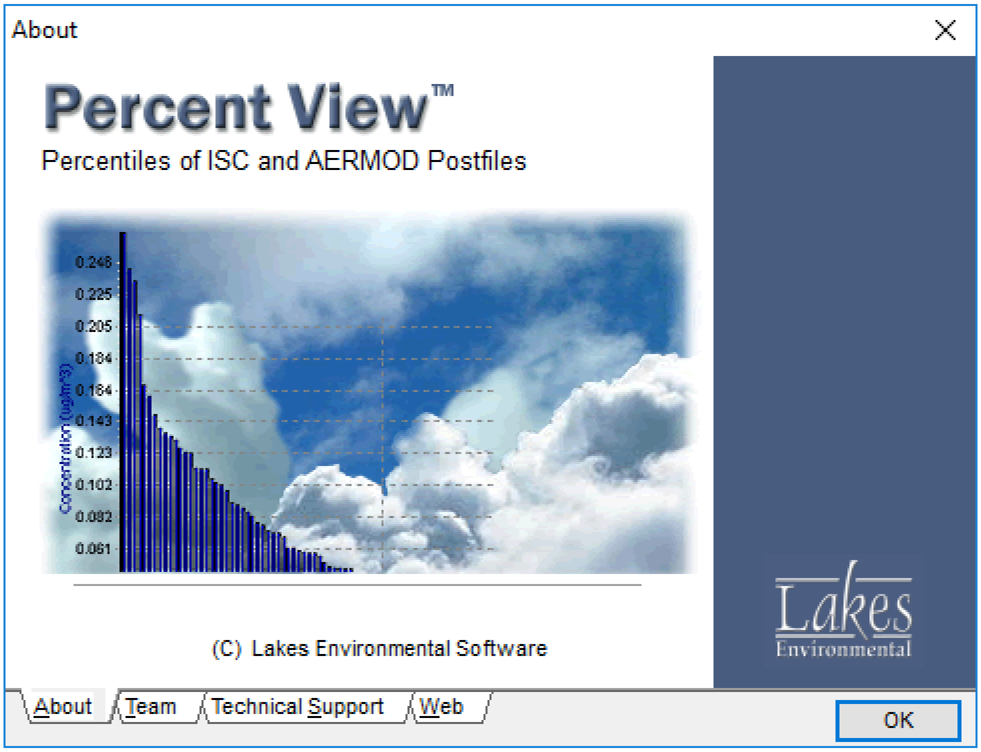


Wind roses can sometimes be used to depict graphically the dominant transport direction of the winds for an area. Due to the influences of local terrain, possible coastal effects, the exposure of the instruments, and the temporal variability of the wind, the wind rose statistics may not always be representative of true transport for an area. Other meteorological conditions may also be important for determining the formation and transport of certain atmospheric contaminants, particularly for reactive pollutants. The results of this program should therefore be used with caution.

> For more information about this product please refer to **WRPLOT View** online help.

Percent View

Percent View is a **Lakes Environmental** Windows program that takes the hassle out of performing modeling runs that require percentiles or rolling averages.



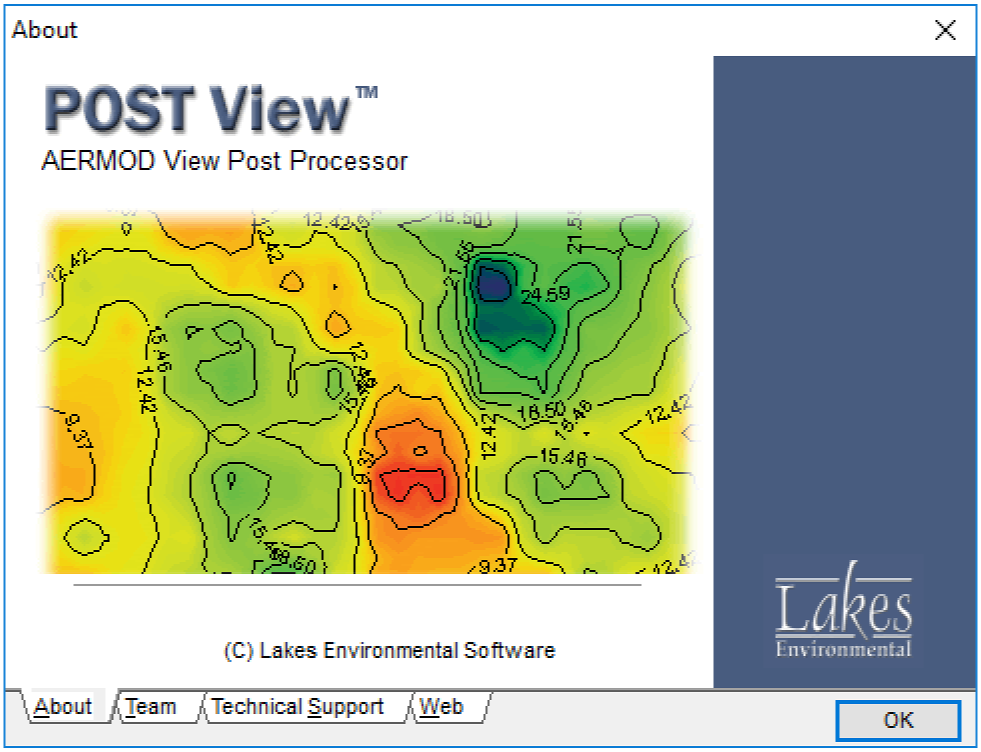
Use **Percent View** to generate percentile plots of a given averaging period contained within an ISCST3, ISC-PRIME, or AERMOD Post Processing File (POSTFILE). Percentile values are routinely used to express air quality standards in an international setting.

Percent View generates a plot file of concentration for a given percentile and pollutant averaging period. Isopleth plot files can be generated for use in **POST View** to demonstrate percentile based concentrations.

For more information about this product please refer to **Percent View** online help.

POST View

POST View is a Windows post-processor specially designed to handle ISC-AERMOD plotfiles. Plotfiles are generated for every AERMOD View run. In **AERMOD View**, plotfiles are defined in the OU–Contour Plot Files window. A Plotfile contains information on the concentration and/or deposition values at each receptor location for a specific averaging period, source group, and high value number.



POST View is a true Windows MDI (multiple-document interface) program which allows you to have multiple contour plots being generated in multiple windows, at one time.

POST View does all the time-consuming tasks of getting all the plotfiles generated by the models (ISCST3, ISC-PRIME, AERMOD, and AERMOD-DEP), gridding and displaying them for you. For easy reference and visualization of your results, you can import base maps and use them as a backdrop for your contour plots. For quality report presentation you can print the contour plots with the overlay site map using **POST View** project templates. You can also copy the contour plots on the clipboard and paste them into your favorite Windows application.

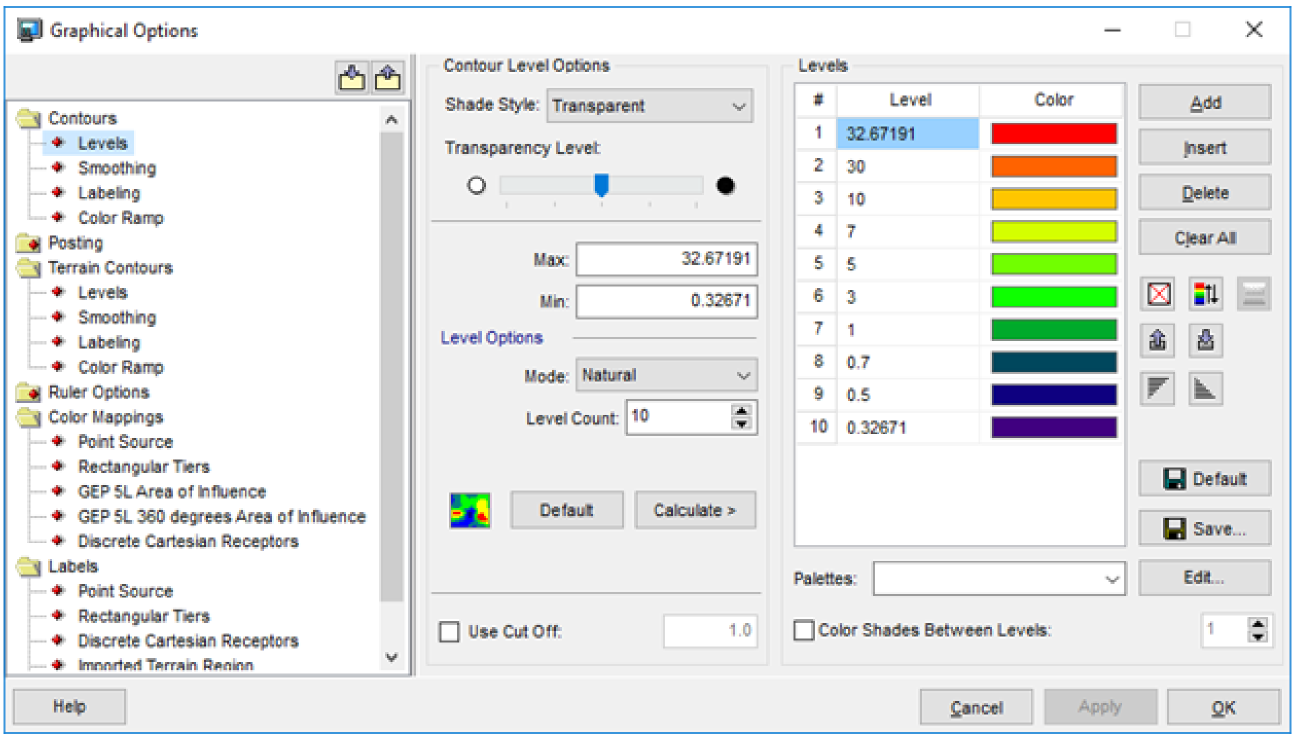
For more information about this product please refer to **POST View** online help.

Graphical Options

The **Graphical Options** dialog contains options to enhance the appearance of your project. Within the **Graphical Options** dialog you can define options such as contour levels, shading, contour line thickness and color, and labels.

You have access to the **Graphical Options** dialog by select **Output | Graphical Options...** from the menu.

The **Graphical Options** dialog uses a two-pane view. The tree, located on the left side of the dialog, is used for navigation and item selection. Select an item (marked as ✦) in the tree to display the available options on the right panel.



Graphical Options dialog

The **Graphical Options** dialog contains several options. These options are explained in the following sections:

- Contours
  - Levels
  - Smoothing
  - Labeling
  - Color Ramp

- Posting
- Terrain Contours
    - Levels
    - Smoothing
    - Labeling
    - Color Ramp
- Ruler Options
- Color Mappings
- Labels

You can create customized **Graphical Options** configurations using the following options:

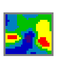
 Allows you to load previously saved configuration.

 Allows you to save current **Graphical Options** configuration as a layout.

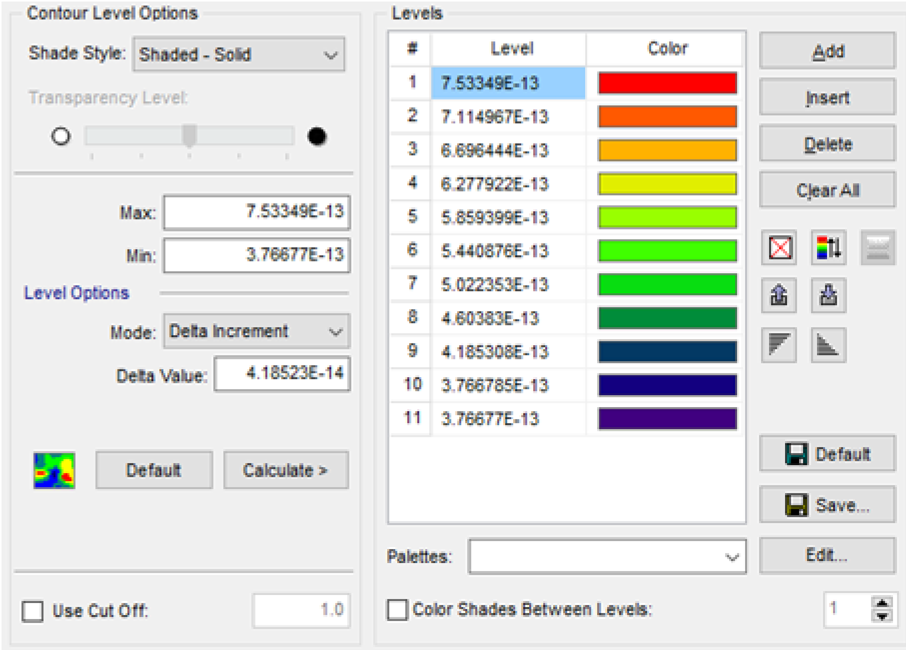
Levels

In the **Levels** page, you specify the contour levels for the current contour plot. Select **Contours - Levels** or **Terrain Contours - Levels** from the tree located on the left side of the Graphical Options dialog, to display the options available for contour levels, shading, and colors.

The **Levels** options are the same for both **Contours** and **Terrain Contours**.

 indicates you are working on the contour settings for your results.

 indicates you are working on the contour settings for your terrain data.



Graphical Options dialog - Levels

Contour Level Options panel

- **Shade Style:** Allows you to define the shading style for your contours. See Contour Shade Styles for a description of available options.

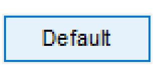
- **Transparency Level:** When you have selected the **Transparent** style, the **Transparency Level** option becomes available. Adjusting the transparency setting allows you to change the shading of the plots in your project.

- **Max.:** This is the maximum value you want to be displayed on your contours. If you press the **Default** button, the maximum value found in your model results is displayed in this field.

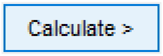
- **Min.:** This is the minimum value you want to be displayed on your contours. If you press the **Default** button, the minimum value found in your model results is displayed in this field.

Level Options panel

- **Level Options:** Some options are available to help you define the contour levels. These options can be selected under the **Mode** drop-down list. See Level Mode for a description of available options.

- 

: This button will get the minimum and maximum values available for the current model results.

- 

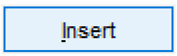
: This button will apply the selected **Level Options** to the table. Once these values are applied, they can be customized in the Levels table.

- **Use Cut Off:** The **Use Cut Off** option allows you to specify no shading for contours below a certain value. This is useful when you want, for example, to mask regions of a plume where the concentration is zero.

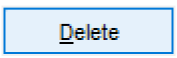
Levels panel

The following buttons are available from the **Levels** panel:

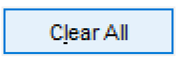
Adds another level to the bottom of the table.



Inserts a level above the selected level.



Deletes the selected level from the table.



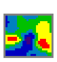
Clears the table.

For an explanation of the other buttons available in the **Ranges** table, see Specifying Colors.

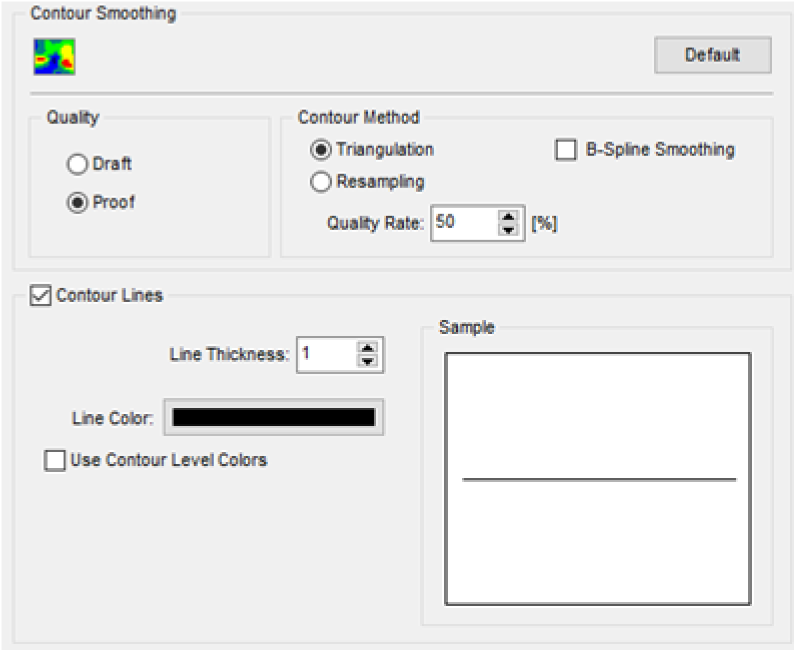
Smoothing

Select **Contours - Smoothing** or **Terrain Contours - Smoothing** from the tree located on the left side of the Graphical Options dialog, to display the options available to control the smoothing of the contour lines, the size of the contour mesh, and the color and thickness of the contour lines.

The **Smoothing** options are the same for both **Contours** and **Terrain Contours**.

 indicates you are working on the contour settings for your results.

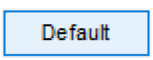
 indicates you are working on the contour settings for your terrain data.



Graphical Options dialog - Smoothing

The following options are available:

Smoothing

- 

: This sets your smoothing selections to the default.

Quality

- **Draft:** Select this option to generate contour plot lines with no smoothing applied to it. The rendering of the contour plots is much faster when using this draft mode.
- **Proof:** Select this option to get smooth contour lines. This option however will delay the rendering of the contour plots.

Contour Method

- **Triangulation:** Combines points of triangles and linearly interpolates between points in a triangle to create values which are then connected to create contours. If this option is selected then you need to specify the Quality Rate.

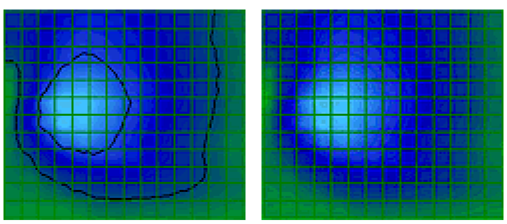
- **Quality Rate:** This option is only available when you have selected the triangulation contour method. For increased quality, additional points may be added in between the existing points, creating a finer resolution and a smoother result.

- **B-Spline Smoothing:** Check this option to get smoother contour lines. This option should be used to get high quality printed outputs; however, it can also be used for other purposes.
- **Resampling:**This option creates a regular grid and calculates values in each grid node using given points. If this option is selected then the Mesh Size should be specified.
  - **Mesh Size:** This option is available when you have selected the resampling contour method. The mesh size determines the size of the grid used to generate the contour plots. Each of the points in the mesh takes its value from the surrounding points. AERMOD View uses a mesh of 50x50 as the default; however, this can be changed at any time.

The higher the quality rate or mesh size, the longer it takes to render contours.

Contour Lines

This option allows you to turn on or off the display of contour lines, but still displaying the color shading.



- **Line Thickness:** Here you specify the thickness of the contour lines.
- **Line Color:** Click on the color bar to display the Color dialog where you can select the color for the contour lines.
- **Use Contour Level Colors:** Check this option to display contours in colors that would be used for color shading.

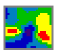
Sample

As you make selections for the line thickness, the sample is updated to reflect the changes.

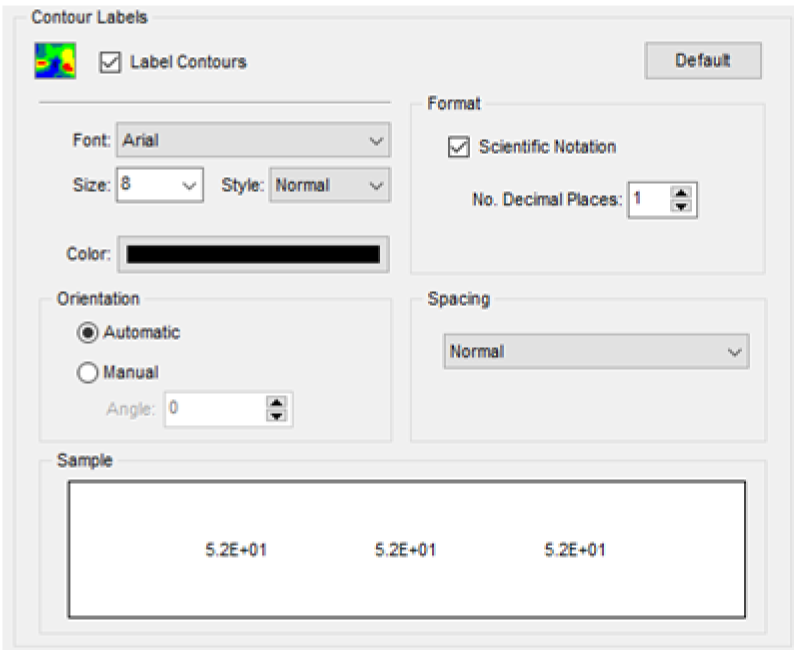
Labeling

Select **Contours - Labeling** or **Terrain Contours - Labeling** from the tree located on the left side of the Graphical Options dialog, to display the options available for contour labels. In the **Labeling** page of the **Graphical Options** dialog, you control the font type, font size, style, and color of the contour labels.

The **Labeling** options are the same for both **Contours** and **Terrain Contours**.

 indicates you are working on the contour settings for your results.

 indicates you are working on the contour settings for your terrain data.



Graphical Options dialog - Labeling

Labels

- **Label Contours:** Check this box if you want labels to be displayed at your contour lines.
- **Default:** This button sets your labeling selections to the default.
- **Font:** Select a font for the contour labels from the list of fonts installed on your Windows system.
- **Size:** Select a font size from the list of valid sizes for the font you selected.

- **Style:** Define a style for the selected font (Normal, Bold, Italic, Bold Italic).
- **Color:** Click the color bar to display the Color dialog where you can select the color for the contour labels.

Format

Allows you to specify the numeric format to use when displaying values for your contour labels.

- **Scientific Notation:** Check this box if you want the contour labels to be written in scientific notation.
- **No. Decimal Places:** Specify the number of decimal places for the contour labels.

Orientation

- **Automatic:** Check this box if you want the orientation of the contour labels to be displayed using the default settings (in-line with the contours).
- **Manual:** Check this box if you want to manually specify the orientation of the contour labels in your project. If you check this option, you must select the angle that you would like to orient the labels.

Spacing

There are a number of spacing options available for your contour labels. From the drop-down list, select the spacing option that best suits your needs. The available options are:

- **Very Infrequent**
- **Infrequent**
- **Normal**
- **Frequent**
- **Very Frequent**

Sample

As you make selections for color, font, size, and style, the sample is updated to reflect the changes.

Color Ramp

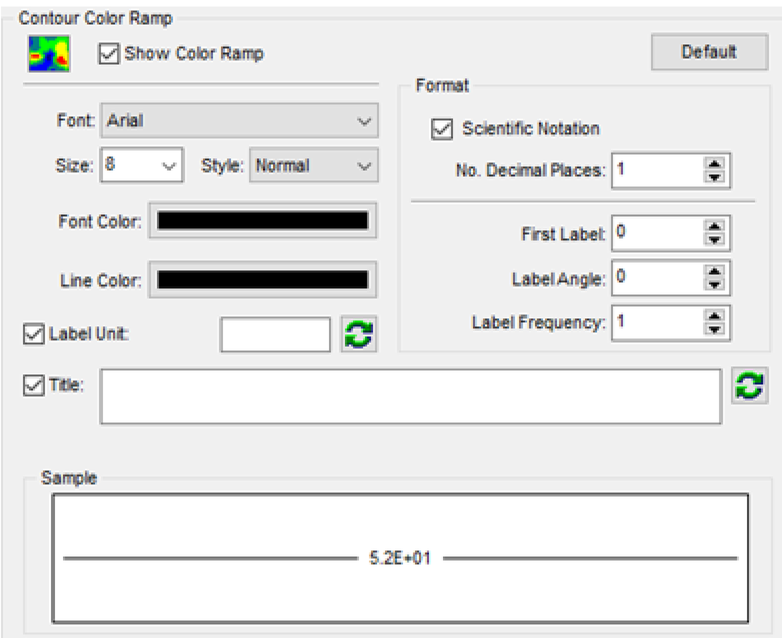
The **Color Ramp** page of the Graphical Options dialog displays the options available for customizing the display of the color ramp. Here you control the display, font type, font size, style, color and format of the color ramp labels.

The **Color Ramp** panel can be displayed in the following ways:

1. Expand the graphical feature (e.g. contours) in the tree located on the left side of the **Graphical Options** dialog and select **Color Ramp**.

2. Double-click on the **Color Ramp** located in the application main window.

3. Right-click on the **Color Ramp** and select **Edit** from the pop-up menu.



Graphical Options dialog - Color Ramp

See below the description of each one of these options:

Color Ramp

- **Show Color Ramp:** Check this box if you would like the Color Ramp to be displayed in the main application window.

You can display color ramps for multiple graphical aspects.

- **Default:** This returns all the color ramp settings to the default values.
- **Font:** Select a font from the list of fonts installed on your Windows system.
- **Size:** Select a size from the list of valid sizes for the font you selected.
- **Style:** Select a style for the selected font (Normal, Bold, Italic, Bold Italic).
- **Font Color:** Click on the color bar to display the Color dialog where you can select a color for the color ramp labels.
- **Line Color:** Click on the Color bar to display the Color dialog where you can select the color for the for the lines used on the color ramp.
- **Label Unit:** This option allows you to specify a label that identifies the unit for the values displayed on the color ramp. The button allows you to get the default unit label for the displayed contour values.

Plotfiles from Lakes Environmental AERMOD View version 5 and up contain information regarding units for concentration and deposition values, either the default output units or the units specified in the Emission Output Unit window. If you have imported a plotfile from a version earlier than Lakes Environmental AERMOD View version 5 or from another source, the default output unit of ug/m3 (micrograms per cubic meter) for concentration values and the default output unit g/m2 (grams per square meter) for deposition values will be used. You are responsible for specifying the appropriate units.

- **Title:** This option allows you to specify a title that describes the color ramp. The button allows you to get the default title for the color ramp.

Format (not available for Land Use)

Allows you to specify the numeric format to use when displaying values on the color ramp.

- **Scientific Notation:** Check this box if you want the color ramp labels to be written in scientific notation.
- **No. Decimal Places:** Specify the number of decimal places to display on the color ramp.

This setting also determines the number of decimal places in printed reports.

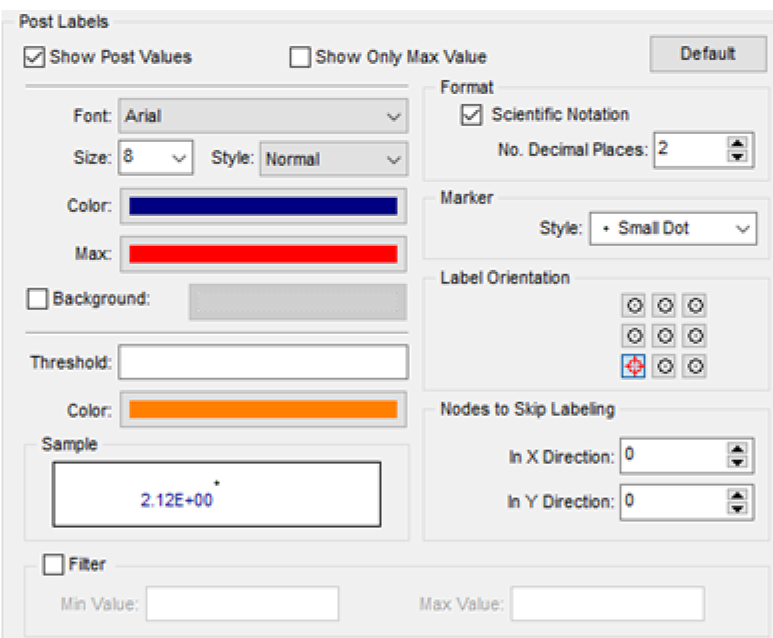
- **First Label:** Specifies the first color interval on the color ramp that is labeled. A value of 0 means that the first label will be placed at the minimum color interval.
- **Label Angle:** Specifies the angle at which the labels are displayed on the color ramp.
- **Label Frequency:** Specifies the interval in the color ramp between labels. A value of 1 means every contour level interval will be labeled.

Sample

As you make selections for color, font, size, and style, the sample is updated to reflect the changes.

Posting

In the **Posting** page, you control the display of posting values. Select **Posting** from the tree located on the left side of the Graphical Options dialog, to display the options available for posting of model results.



Graphical Options dialog - Posting

The following options are available:

Post Labels

- **Show Post Values:** Check this box if you would like the Post Values to be displayed at each receptor node. You can also turn on and off the display of posting with the **Show Concentrations** tool on the Graphical Output Toolbar.
- **Show Only Max Value:** Specify that only the maximum output value should be shown.
- **Default:** This sets your Posting selections to the default values.
- **Font:** Select a font for the post value labels from the list of fonts installed on your Windows system.
- **Size:** Select a size from the list of valid sizes for the font you selected.
- **Style:** Select a font style for the post value labels (Normal, Bold, Italic, Bold Italic).
- **Color:** Click the color bar to display the Color dialog from where you can select a color for your post value labels.
- **Max:** Click the color bar to display the Color dialog from where you can select a color to represent the maximum post value.
- **Background:** For better visibility of cell output values over base maps, the cell values can be represented with a background color.
- **Threshold:** Here you can specify a threshold value which can then be labeled in a different color for easy identification.
- **Color:** Click the color bar to display the Color dialog from where you can select a color for your threshold post value labels.

Format

- **Scientific Notation:** Check this box if you want post values to be written in scientific notation.
- **No. Decimal Places:** Specify the number of decimal places for the post values.
- **Marker:** You can specify the style of the marker to be used to mark the location where posting values are available (e.g, receptor nodes).
- **Label Orientation:** Specify the location of the post values relative to the marker by clicking on the button that best corresponds to the desired orientation.
- **Nodes to Skip Labeling:** This option allows you to specify a skip factor for posting of results. When the nodes to skip labeling in X and Y direction are set to 0, posting labels are shown at

every node. If you set the values for X and Y directions at 2 for example, two posting labels are skipped between nodes.

Sample

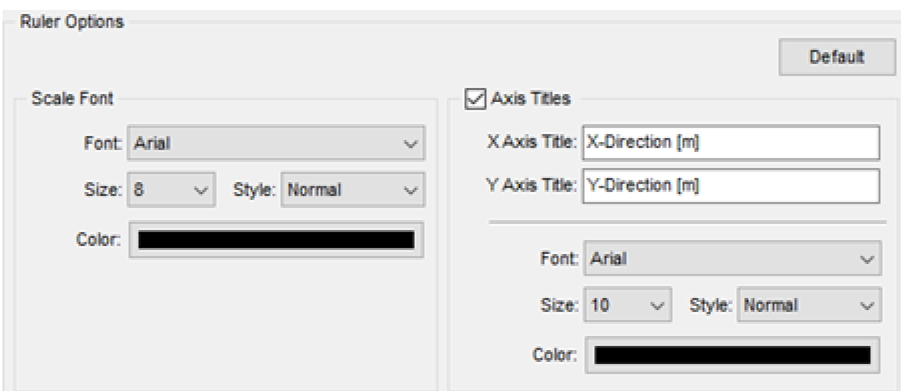
As you make selections for color, font, size, and style, the sample is updated to reflect the changes.

Filter

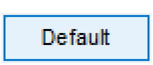
Check the box to enable this option. Here you can specify maximum and minimum values for posting. When this feature is enabled, only posting values in the specified range are displayed. The user can specify either a max value, a min value, or both a max and min value.

Ruler Options

The **Ruler Options** page of the Graphical Options dialog allows you to make changes to the axis labels along the top and left side of the drawing area. Here you can change the font, size, color of the ruler and apply axis titles.



Graphical Options dialog - Ruler Options



At any time, you can click on this button to return to the default settings.

Scale Font

In this frame you can determine the display of the scale font. The following options are available:

- **Font:** Select a font from the drop-down list.

- **Size:** Select a font size from the drop-down list.
- **Style:** Select a font style from the drop-down list.
- **Color:** Click on the color bar to open the Color dialog from where you can select a color for your text.

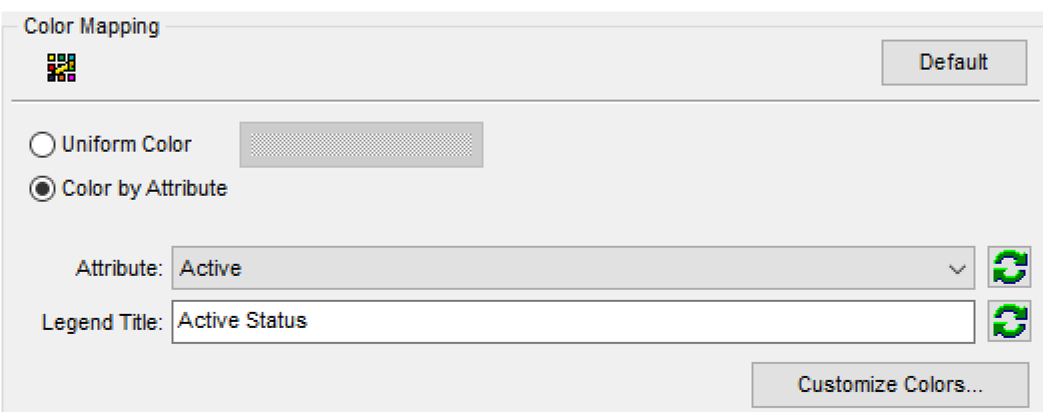
Axis

Here you can specify axis titles and make selections for color, font, size, and style for your text. The following options are available:

- **X-Axis Title:** Enter a title for the X-Axis.
- **Y-Axis Title:** Enter a title for the Y-Axis.
- **Font:** Select a font from the drop-down list.
- **Size:** Select a font size from the drop-down list.
- **Style:** Select a font style from the drop-down list.
- **Color:** Click on the color bar to open the Color dialog from where you can select a color for your text.

Color Mappings

The **Color Mappings** page of the Graphical Options dialog enables you to change the color and display of objects in the drawing area. Changes made to an object are specific to the object. This option is available only if you have imported a shapefile into your project or if you have defined buildings.



Graphical Options dialog - Color Mappings

Color Mapping

- **Uniform Color:** If this option is selected, then object will be displayed in the fixed, uniform color you have selected. Click on the color bar to display the Color dialog from where you can select the desired color.

- **Color by Attribute:** This option allows you to select one of the object's attributes as the color scheme. Press the **Customize Colors** button to display the Custom Attribute Colors dialog from where you can define colors for each attribute class or range. In the **Legend Title** field you can specify a title for your legend which will be displayed in the Tree View - Overlays tab.

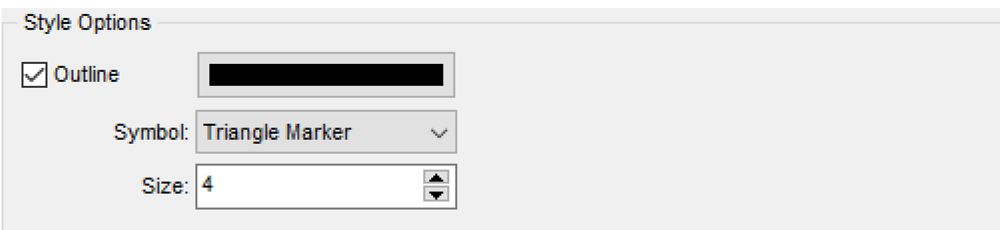
  If you wish to return to the default legend title click on the  button.

Style Options

The **Style Options** depend on the type of object, for which the colors scheme is being defined.

Point

The **Point** type objects have the following **Style Options**:



- **Outline:** To display the outline of each object, check the **Outline** box. You can then click on the color bar to display the Color dialog from where you can select a color for the outline

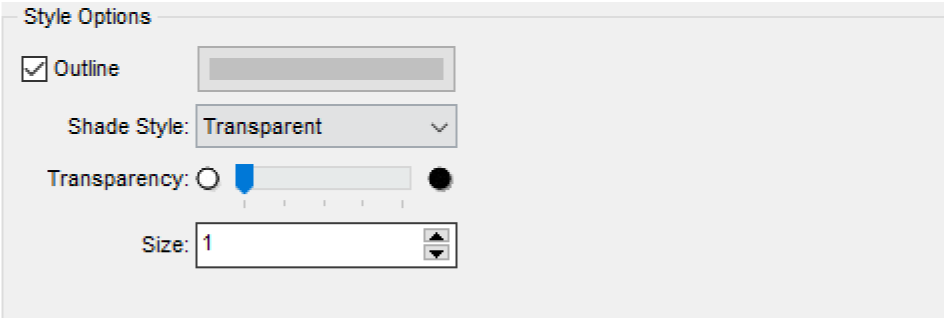
> **Circle**, **Square**, and **Triangle** symbols have an outline and a uniform color fill. The color of the **Cross** marker is set by the **Uniform Color**.

- **Symbol:** Select the symbol style from drop-down list.

- **Size:** Set the symbol size.

Polygon

The **Polygon** type objects have the following **Style Options**:



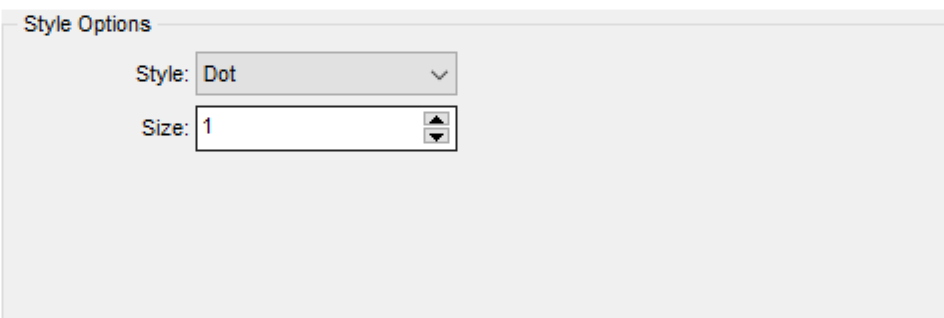
- **Outline:** To display the outline of each object, check the **Outline** box. You can then click on the color bar to display the Color dialog from where you can select a color for the outline.

- **Shade Style:** Select shading style from the drop-down list. If you select **Transparent**, you can use the **Transparency** slider to set the level of opacity/transparency.

- **Size:** Set the thickness of the outline.

Line

The **Line** type objects have the following **Style Options**:

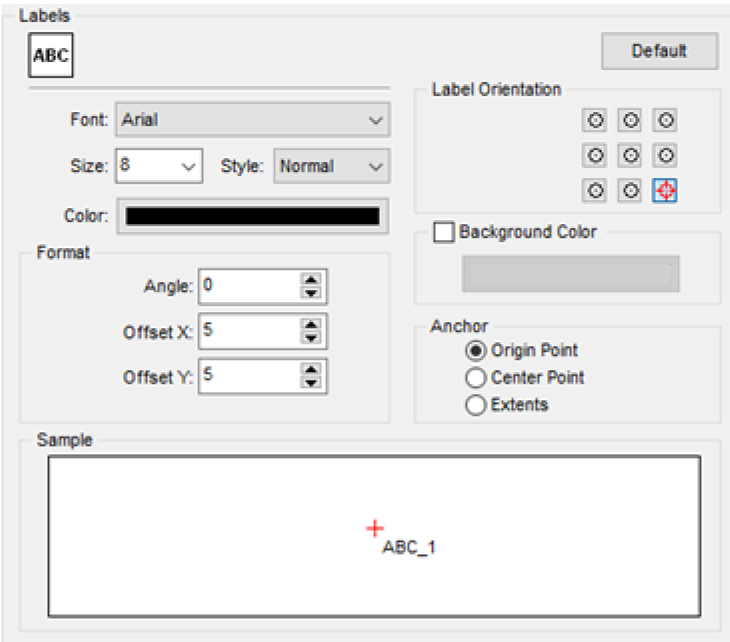


- **Style:** Select the line style you wish to use.

- **Size:** Set the line thickness.

Labels

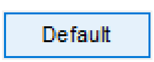
The **Labels** page of the Graphical Options dialog enables you to change the appearance of the labels for the objects in the drawing area. The selections available in the **Labels** tab are the same, regardless of which object type you are changing the labels for. Changes to your labels are object specific. For example, if you change the label settings for your point sources, you should note that the label settings of the other objects will not be changed.



Graphical Options dialog - Labels

The following options are available:

Labels

- 

: Click on this button to return all your font settings to the default.

> The default that is applied to the labels are the default settings you specified in the Default Settings page of the Preferences dialog.

- **Font:** Select a font from the drop-down list.
- **Size:** Select a font size from the drop-down list.
- **Style:** Select a font style from the drop-down list.
- **Color:** Click on the color bar to display the Color dialog from where you can select a color for the labels.

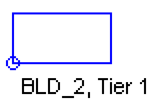
- **Label Orientation:** Specify the location of the labels relative to the object marker by clicking on the button that best corresponds to the desired orientation.

Format

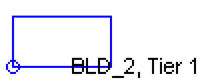
- **Angle:** If you would like your label placed at an angle relative to the marker, enter the angle value here.
- **Offset X:** This is the X distance between the marker and the label. The default is 5, but you can change this anytime.
- **Offset Y:** This is the Y distance between the marker and the label. The default is 5, but you can change this anytime.
- **Background Color:** Check the Background Color box if you would like to change the background color just around the label. Click on the color bar to open the **Color** dialog from where you can select a background color.

Anchor

- **Origin Point:** Select this option if you would like the label anchored around the origin point. The origin point is the SW corner of the object, such as a building or grid.



- **Center Point:** Select this option if you would like the label anchored around the center point of the object.



Sample

As you make selections for color, font, size, and style for your labels, the sample is updated to reflect the changes.

Graphical Output Toolbar

The **Graphical Output Toolbar** allows you to easily turn on and off contouring, posting of results, and acts as a quick reference to the to the maximum result value. This toolbar also allows you to select the desired plotfiles and output type to be displayed. As you select different output types, the drawing area is refreshed and results are displayed for the new option.

Graphical Output Toolbar

- **Collapse tool ():** The collapse tool allows you to remove the details of the parameters displayed in the drawing area.

- **Expand tool () :** The expand tool allows you to add the details of the parameters displayed in the drawing area.

- **Show Contours tool ():** With this tool, you have the option of turning on or off the display of the contours.

- **Show Concentrations tool ():** With this tool, you have the option of turning on or off the display of the result values at each receptor location.

- **Show Plume Animation Contours ():** With this tool, you have the option of turning on or off the display of contours in the plume animation mode.

- **Show Terrain tool ():** If you have elevated terrain, this tool gives you the option of turning on or off the display of the terrain contours.

- **Plotfile List:** This drop-down list contains all the plotfiles generated for the current run. Select from this list the plotfile you wish to be displayed.

- **Preview Tool ():** With this tool, you have the option of previewing the contents of the selected plotfile in WordPad.

- **Output Type:** The **Output Type** drop-down list displays the output types available for the current AERMOD Plotfile. This depends on what output options you selected in the Dispersion Options screen in the Control Pathway. The possible output types for each plotfile are:

    - **Concentration:** Select this option to show concentration results for the current plotfile.

    - **Deposition:** Select this option to show deposition results for the current plotfile.

    - **Dry Deposition:** Select this option to show dry deposition results for the current plotfile.

    - **Wet Deposition:** Select this option to show wet deposition results for the current plotfile.

    - **Elevation:** Select this option to show elevations for the current plotfile.

    - **Hill:** Select this option to show hill profile for the current plotfile.

Select the desired output type from the drop-down list. The plotfile for the selected output will be displayed in the drawing area.

- **Max:** AERMOD View reads the plotfile, which gives the calculated concentration and/or deposition values at each receptor location, and then displays in the Max panel the maximum concentration or deposition value along with the X and Y coordinates for the position of receptor where this maximum value occurs.

> Plotfiles from Lakes Environmental AERMOD View version 5 and up contain information regarding units for concentration and deposition values, either the default output units or the units specified in the Emission Output Unit window. If you have imported a plotfile from a version earlier than Lakes Environmental AERMOD View version 5 or from another source, the default output unit of ug/m3 (micrograms per cubic meter) for concentration values and the default output unit g/m2 (grams per square meter) for deposition values will be used. You are responsible for specifying the appropriate units, if different than the default, in the Color Ramp screen in the Graphical Options dialog.

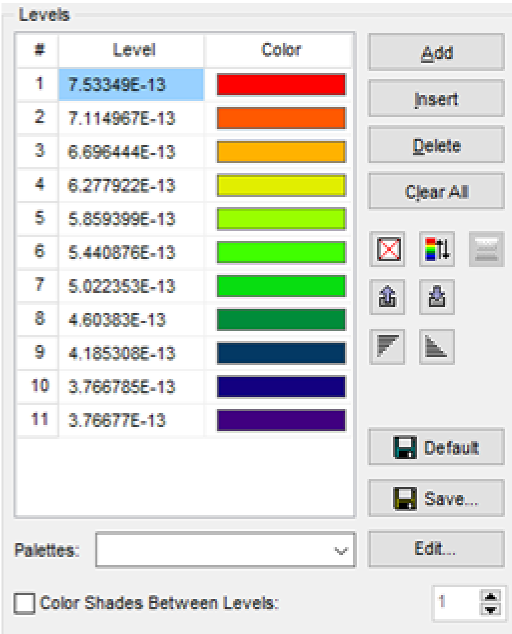
Accessory Information

Specifying Colors

Contour or **Range** colors are specified in one of the following ways:

- By **double-clicking** on the Color cell for each level. This will display the Color dialog, where you can select the desired color from a wide range of colors.
- By **selecting a color palette** from the Palettes drop-down list. Palette names displayed in blue color font are built-in color palettes. Palettes in black font are user-defined palettes. See Palette Manager for a description on how to create user-defined color palettes.

> The **Default** palette is the palette that is loaded for all your projects by default.



Some tools are available to allow you to clear cells, to invert the entire range of colors, and to interpolate colors. See the description of these tools below:

Clear Color In Selected Row: Press this button to remove the color from the selected row.

Invert Colors: Press this button to invert the order of the colors currently displayed on the Levels table.

Interpolate Colors: Select this button to interpolate the colors. See Interpolating Colors.

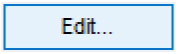
Export Color Settings: Allows you to save the value classes and selected color settings to a file. This option allows users, for example, to save pollutant related classes based on limit values.

Import Color Settings: Allows you to load color settings from a file.

Default If you have created a palette and wish to save the new colors you have selected to be the Default palette for all your projects, click on this button.

> You may also specify your **Default** palette colors in the Default Palette page of the Preferences dialog to be used in all your projects.

Displays the **Save Palette As...** dialog. This dialog allows you to save the current color selection as a user-defined palette.



Displays the Palette Manager dialog where you can edit a palette or create new palettes.

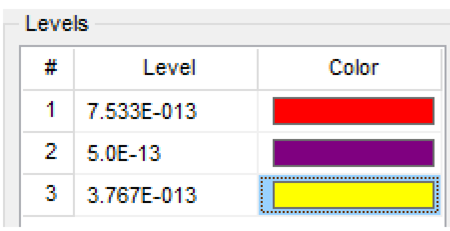
- **Color Shades Between Levels:** Checking this box allows for a more graduated color scheme. You can specify up to 100 shades between levels.

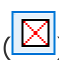
Interpolating Colors

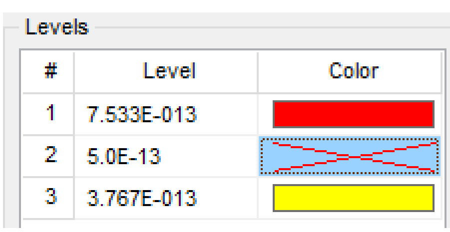
This option easily allows you to create an attractive color gradient for your contour plot.

How to Interpolate Colors:

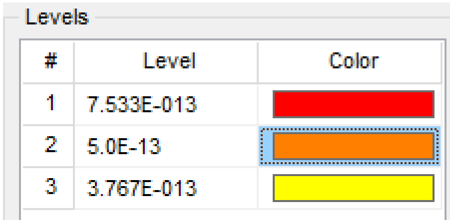
1. Select the color that you would like to interpolate.



2. Select the **Clean Color in Selected Row** button (). You need at least one blank (no color) level to be able to interpolate colors.

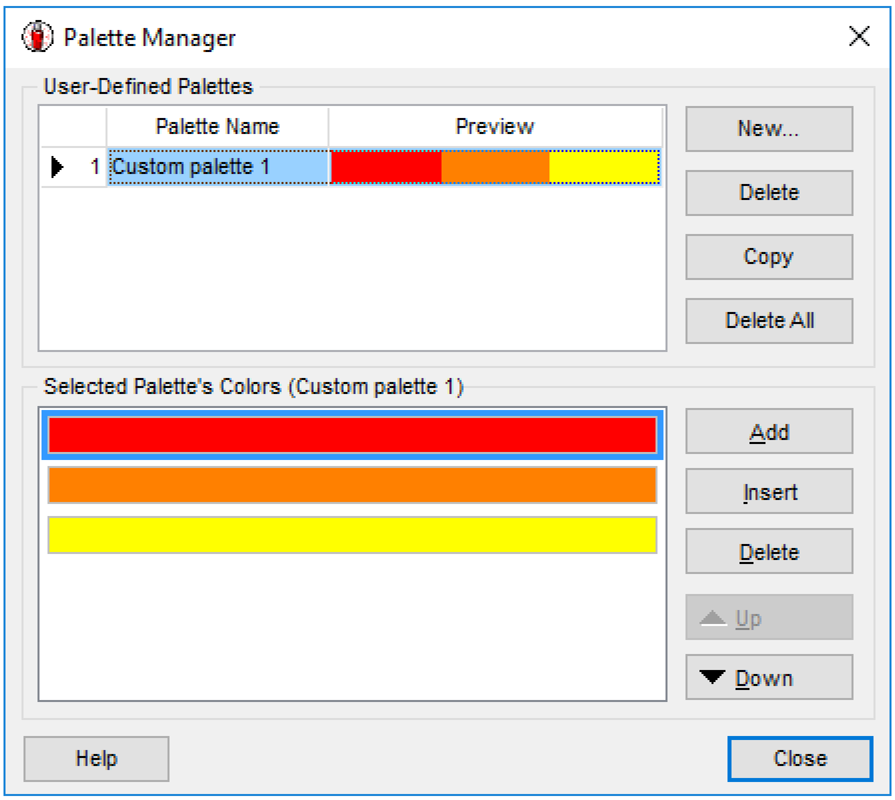


3. Select the **Interpolate Colors** button () to interpolate between two colors and create a custom color scheme.



Palette Manager

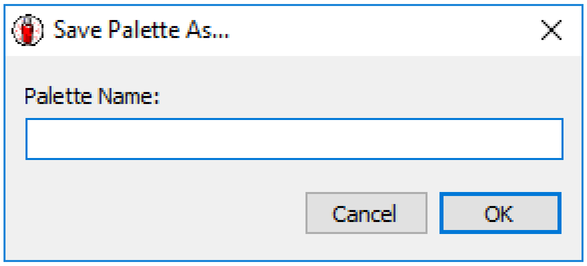
The **Palette Manager** dialog is where you can define, edit and save your own selection of colors to be used in all your projects. Color palettes saved in this dialog are stored in the palettes.xml file located in the same folder as your application.



Palette Manager

How To Create Color Palettes:

1. Press the **New** button located on the right side of the **User-Defined Palettes** frame. The **New Palette** dialog is displayed.



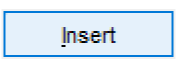
New Palette dialog

2. Type a short description for your color palette and press the **OK** button.

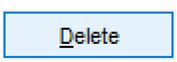
3. Note that the bottom frame (**Selected Palette's Colors**) is blank. This is the place you specify the colors to be saved with your color palette.

The following buttons are available to help you specify and manage your palettes and selection of colors:

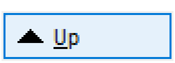
Press this button to display the Color dialog where you can select the desired color. The new color will be added at the end of the table.



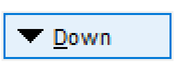
Press this button to insert a color above the selected color.



Deletes the selected color from the custom palette.



Moves the selected color up one level in the color order.



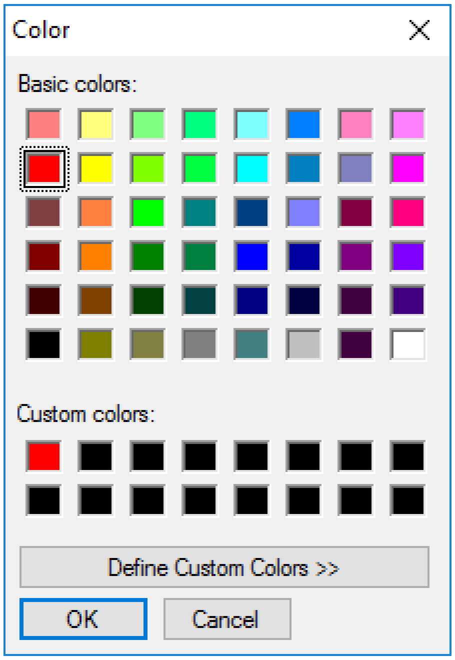
Moves the selected color down one level in the color order.

4. Once the colors are selected, press the close button to close the **Palette Manager** dialog. Note that the new user-defined palette is displayed under the *Palettes* drop down list.

> If you have created custom palettes for your contours in previous versions of the software, you can chose to keep those saved palettes. By copying the "palettes.xml" from the install directory of the previous version to the install directory of the new version, you can keep all your previously saved palettes.

Color Dialog

The **Color** dialog allows you to modify colors of objects or layers.

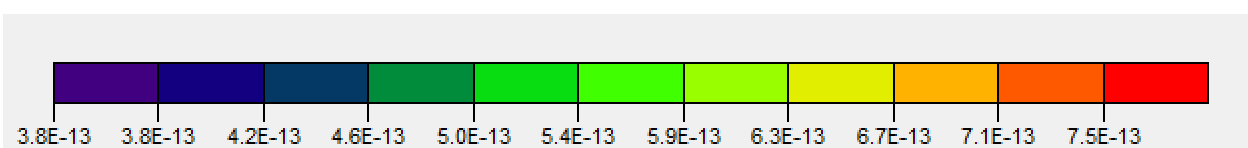


Color dialog

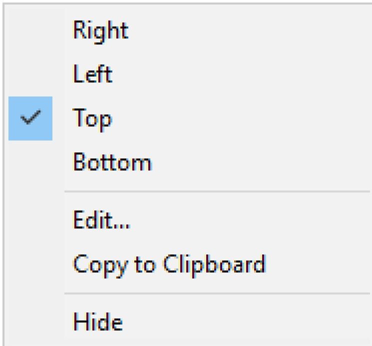
Select a color from the table of Basic colors or from Custom colors, and click on the **OK** button to apply that color.

Color Ramp

The **Color Ramp** displays the number of colors selected from the Levels page of the Graphical Options dialog. In the **Color Ramp** page of the **Graphical Options** dialog, you control the display, font type, font size, style, color and format of the **Color Ramp** labels.



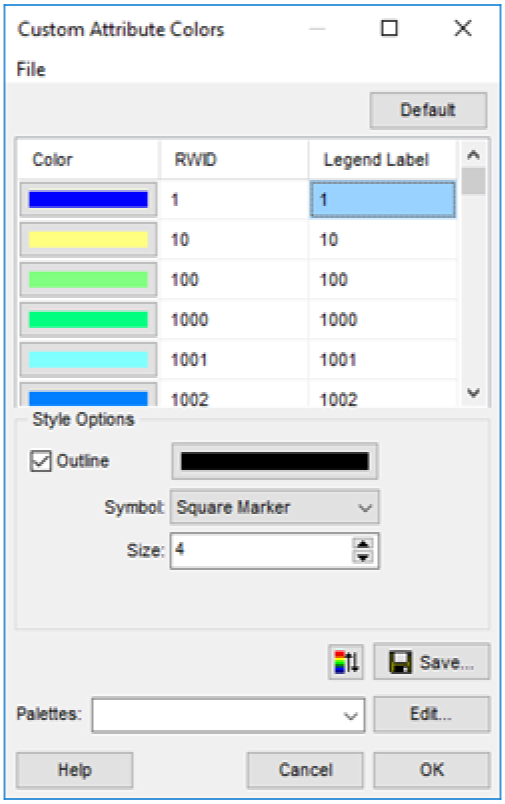
In addition to all these options, you can also right click with the mouse on the **Color Ramp** in the main window to display a floating menu from where you can select various options described below:



- **Right:** Displays the Color Ramp on the right side of the main window.
- **Left:** Displays the Color Ramp on the left side of the main window.
- **Top:** Displays the Color Ramp at the top of the main window.
- **Bottom:** Displays the Color Ramp at the bottom of the main window.
- **Edit...:** Displays Color Ramp page of the **Graphical Options** dialog.
- **Copy to Clipboard:** Copies the Color Ramp to the clipboard, allowing you to paste it into another Windows application.
- **Hide:** Select this option if you do not want to display the Color Ramp in the main window.

Custom Attribute Colors

The **Custom Attribute Colors** dialog allows you to define colors for each attribute class of a shapefile.



Custom Attribute Colors dialog

In the **Color** column, colors have been randomly allocated for each class or range. You can change the colors at any time by clicking on a color bar to display the Color dialog from where you can select a color.

The attribute column or, if there is a range of values, max and min columns are displayed. The information in these columns cannot be altered. In the **Legend Label** column, you may specify the label you would like displayed for each attribute or range in the legend of the Tree View - Overlays tab. For example, you may enter a name to be displayed instead of an ID number, as this would be more meaningful.

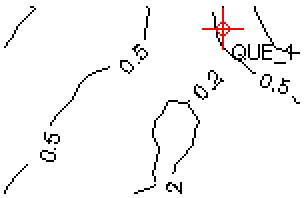
At the bottom of this dialog is the **Palettes** drop-down list from where you can choose a color palette. Palette names displayed in blue color font are built-in color palettes. Palettes in black font are user-defined palettes.

Shade Styles

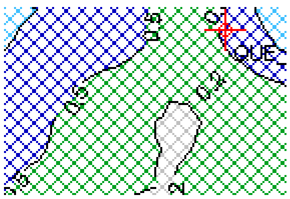
Shade Style options are available under the Graphical Options dialog, allowing you to choose the shading style that best applies to your case.

The following **Shade Styles** are available:

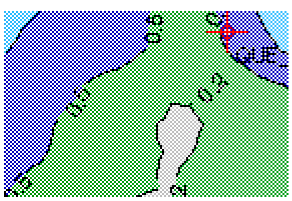
- **None:** Only contour lines are displayed without any color shading.

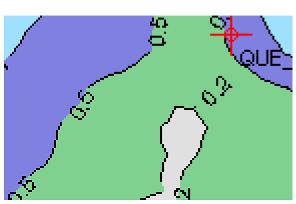
- **Hatch:** Contours are displayed with a transparent hatch shading, allowing your base maps on the background to still be visible.

- **Shaded:** Contours are displayed using the color scheme you selected. Four levels of transparency are available:
  - Shaded - Solid
  - Shaded - Dark
  - Shaded - Medium
  - Shaded - Light

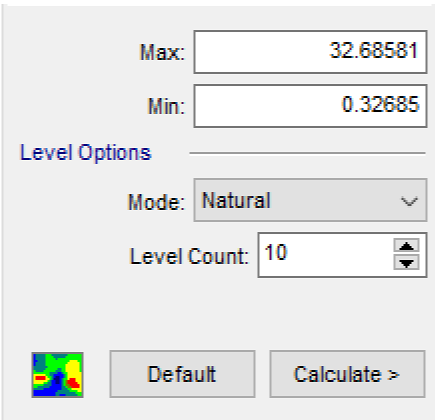
  

- **Transparent:** Contours are displayed in the color scheme you selected with the transparency level you set using the **Transparency Level** control. If you set the **Transparency Level** all the way to the left, the contour colors will be completely transparent. If you set **Transparency Level** all the way to the right, the contour colors will be completely solid.

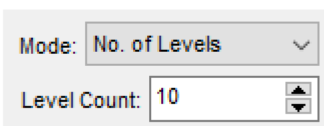
  

Level Mode

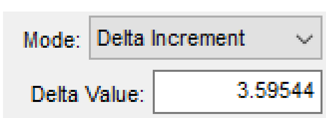
Contour level options are available under the Graphical Options - Levels page to help you define contour levels to be used for plotting your model results. The following options can be selected under the **Mode** drop-down list.



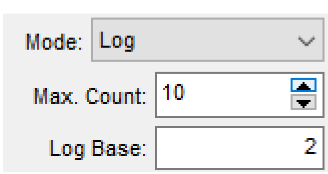
- **No. of Levels Mode:** Allows you to specify the total number of contour levels to be used. By selecting this option, you are requested to specify the Level Count. After pressing the **Calculate** button, contour levels will be calculated based on the specified Max, Min, and Level Count values.



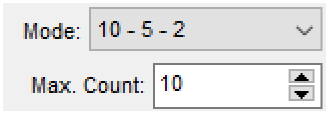
- **Delta Increment Mode:** Allows you to specify the interval to be used between contour levels (delta increment). By selecting this option, you are requested to specify the desired Delta Value. After pressing the **Calculate** button, contour levels will be calculated based on the specified Max, Min, and Delta values.



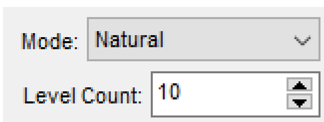
- **Log Mode:** Allows you to specify the interval to be used between contour levels based on a logarithm scale. By selecting this option, you are requested to specify the desired Log Base and the Max. Count (meaning maximum number of levels). After pressing the **Calculate** button, contour levels will be calculated based on specified Max, Min, Max. Count, and Log Base values.



- **10-5-2 Mode:** Allows you to specify the contour levels based on a 10-5-2 scale. By selecting this option, you are requested to specify the desired Max. Count (meaning maximum number of levels). After pressing the **Calculate** button, contour levels will be calculated based on the specified Max, Min, and Max. Count values.



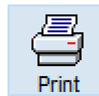
- **Natural Mode:** Allows you to set up the levels ranges with only one non-zero digit in the number (e.g. 0.01, 0.2, 3, 40, 500, 6000, etc. 250 would be disallowed as it has 2 non-zero digits). By selecting this option, you are requested to specify the Range Count. When the **Calculate** button is pressed, shaded range will be calculated based on the number of ranges you have specified, distributing the number of ranges evenly through each log range (0-10, 10-100, 100-1000, etc.). However, the algorithm always keeps certain key numbers, which may result in a greater number of ranges than specified. The key numbers are: lowest value, highest value, and numbers beginning with 1 and 5.



Printing

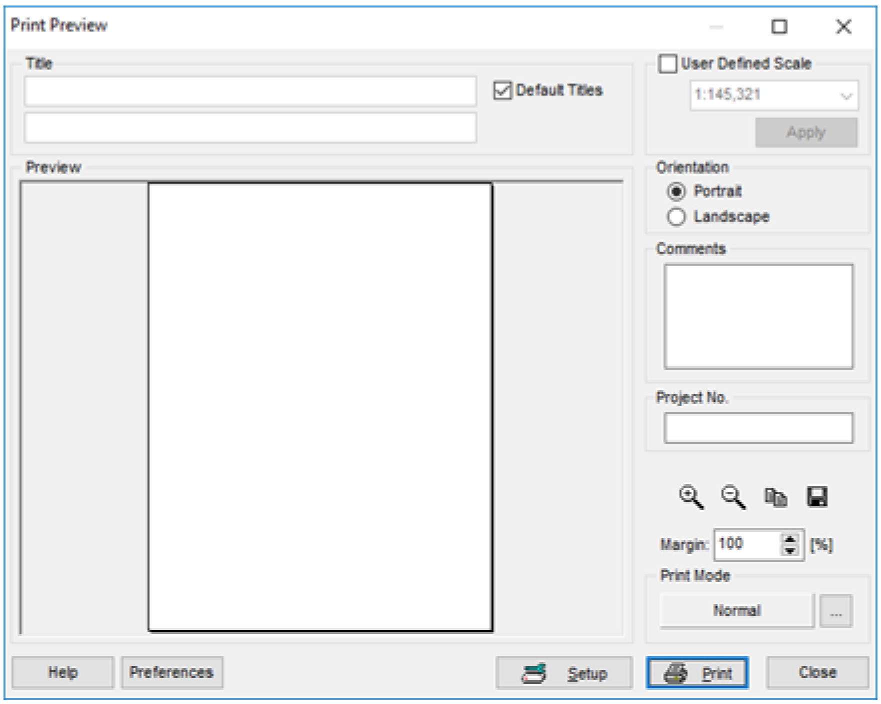
The AERMOD View allows you to print and export your data and results in a variety of ways.

Print Preview

The **Print Preview** dialog is displayed by clicking on the 

button located on the **Menu Toolbar** or when you select **File | Print...** from the menu.

Before you can print the contents of the drawing area, AERMOD View displays the **Print Preview** dialog. Here you can select printing options and preview how your printouts will look. AERMOD View prints the contents of the drawing area into templates. These templates were designed so you can have important information about your project automatically added to your printouts.



Print Preview dialog

> The number of decimal places in the printed report is controlled in the [Graphical Options | Color Ramp]().

The **Print Preview** dialog contains the following options:

- **Title:** AERMOD View places the name of your project as the default title. However, you have the option of specifying different titles. To specify different titles, uncheck the **Default Titles** box and type in the titles.

- **User Defined Scale:** Here you can the scale for your printout. The default scale used is 1:14848 but if you wish to change the scale, check the **User Defined Scale** box which then allows you to select the scale you wish to use from the drop-down list. You can also type in a scale and click on the Apply button to apply it.

- **Orientation:** This is the orientation for your printout. Note that the preview area shows a preview of your results in both orientations- portrait and landscape.

- **Comments:** In this field you can type any comments or notes you want to be printed along with your printouts.

- **Project No.:** In this field you can specify your project number.

- **Margin:** Enter a percentage for the margin. The default is 100%. Specifying a percentage <100% will produce a narrower margin than the default. Specifying a percentage >100% will produce a wider margin than the default.

- **Print Mode:** Displays which Print Mode option you are using. Click on ... to open the **Print Mode** dialog where you can select whether to print in Normal or Bitmap Mode.

Button Guide

At the lower right corner of the dialog a series of buttons are available. See the function of these buttons below:

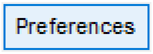
Zoom In: If you select this tool, your mouse pointer changes to a magnifying glass. On the Preview area, click on the location you want to zoom in. Click as many times as necessary, until you have the right magnification.

Zoom Out: Select this tool to go back to the original image size.

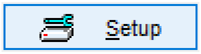
Copy to Metafile: Select this tool if you want to copy the image to the clipboard as a Microsoft® Windows® Metafile. You can then paste into any Windows application that supports pasting of a Windows Metafile from the clipboard.

Save to File: Select this option if you want to save the print preview image to a file. You can save the printout as an Enhanced Windows Metafile (*.emf) or a bitmap (*.bmp).

Help Displays the **Help** contents for the options contained in the **Print Preview** dialog.

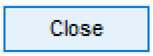


Displays the Preferences dialog where you can specify printing and labeling options for your printouts.



Displays the **Print Setup** dialog where you can specify printing options such as paper size and orientation.

Displays the **Print** dialog where you can specify printing options such as printer and number of copies.



Closes the **Print Preview** dialog.

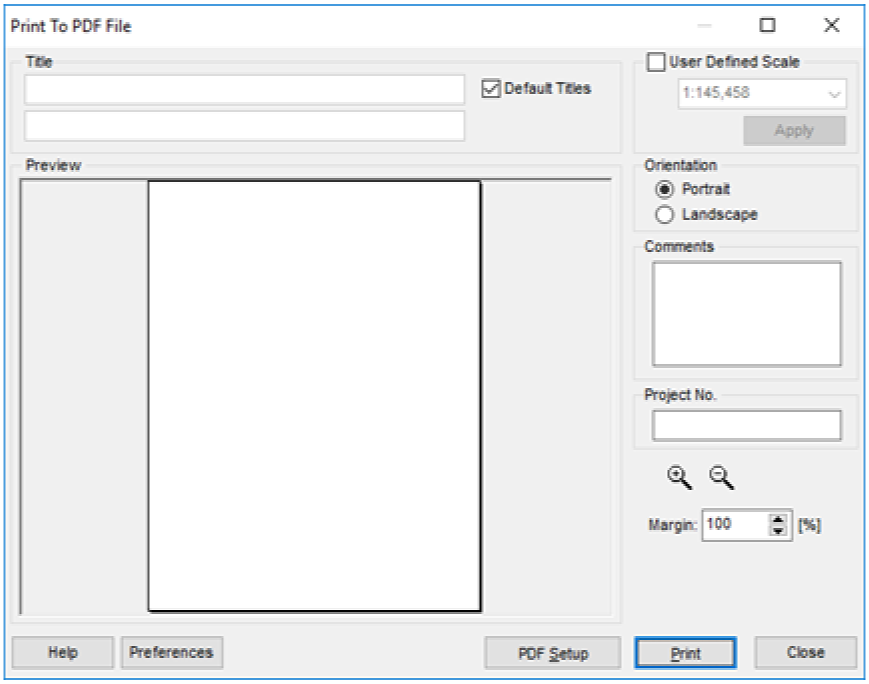
Print to PDF File

The **Print to PDF** option allows you to print the contents of the graphical area to a PDF file. You do not need to have a PDF driver installed in your computer. The PDF printing drivers are installed with the AERMOD View package.

You have access to the **Print to PDF** option in the following ways:

1. Select the menu option **File | Print to PDF**

2. Press the **Print** menu toolbar button and then select the option **To PDF...** from the floating menu.



Print To PDF File dialog

The **Print To PDF File** dialog contains the following options:

- **Title:** AERMOD View places the name of your project as the default title. However, you have the option of specifying different titles. To specify different titles, uncheck the Default Titles box and type in the titles.

- **User Defined Scale:** Here you can change the scale for your printout. A default scale is automatically calculated but if you wish to change the scale, check the User Defined Scale box which then allows you to select the scale you wish to use from the drop-down list. You can also type in a scale and click on the button to apply it.

- **Orientation:** This is the orientation for your printout. Note that the preview area can preview your results in either orientation- portrait or landscape, depending on which option is selected.

- **Comments:** In this field you can type any comments or notes you want to be printed along with your printouts.

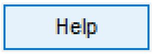
- **Project No.:** In this field you can specify your project number.

- **Margin:** Enter a percentage for the margin. The default is 100%. Specifying a percentage <100% will produce a narrower margin than the default. Specifying a percentage >100% will produce a wider margin than the default.

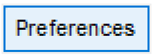
At the lower right corner of the dialog a series of buttons are available. See the function of these buttons below:

 **Zoom In:** If you select this tool, your mouse pointer changes to a magnifying glass. On the Preview area, click on the location you want to zoom in. Click as many times as necessary, until you have the right magnification.

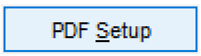
 **Zoom Out:** Select this tool to go back to the original image size.



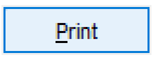
Displays the help contents for the options contained in the **Print Preview** dialog.



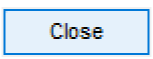
Displays the **Preferences** dialog where you can specify printing and labeling options for your printouts.



Displays the **PDF Setup** dialog where you can specify printing options such as paper format, size and resolution.



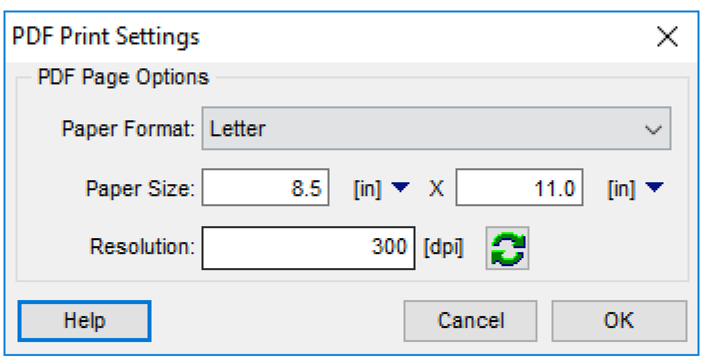
Displays the **Save to PDF** File dialog where you can specify where to save the PDF file.



Closes the **Print To PDF** File dialog.

PDF Print Settings

You have access to the PDF Print Settings dialog from the Print to PDF window.



PDF Export Settings dialog

The following options are available in the PDF Print Settings dialog:

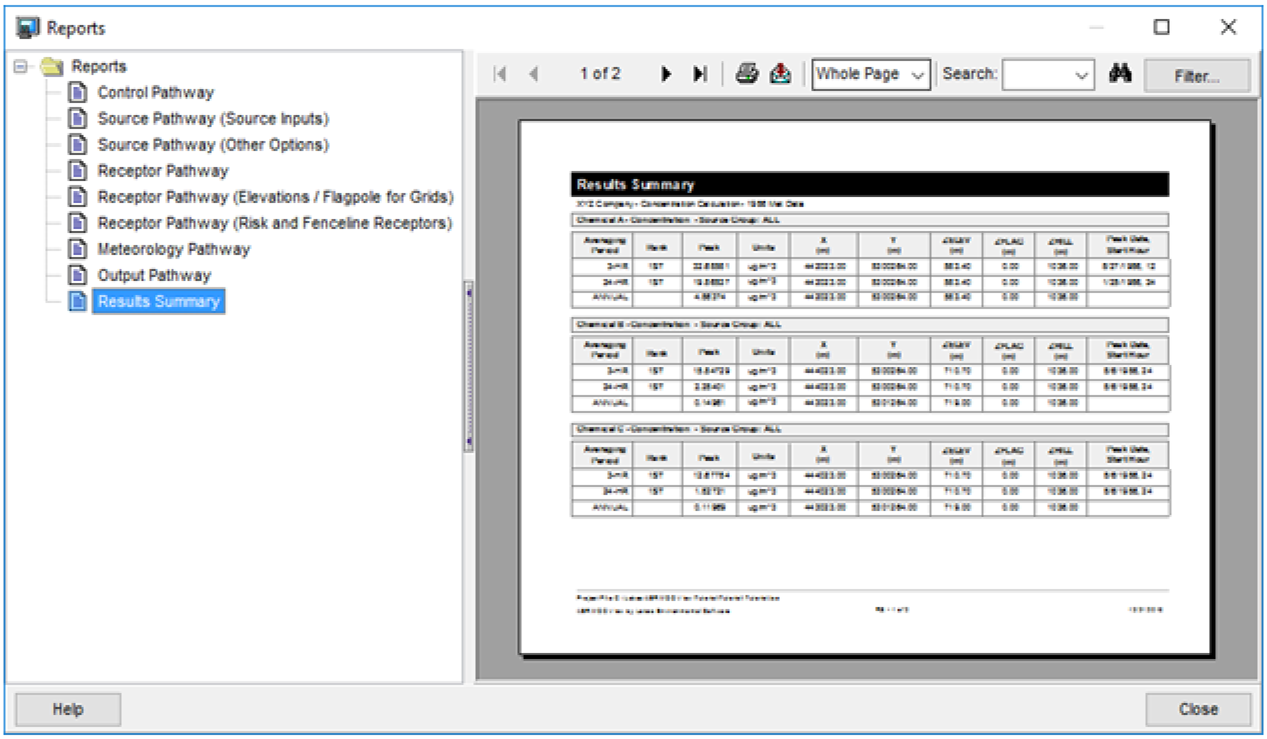
- **Paper Format:** Choose the paper format from a selection of all standard page sizes.
- **Paper Size:** Define the paper size length and width dimensions in [cm] or [in].

- **Resolution:** Define the resolution in Dots per inch (DPI) for the PDF output.

Reports

The **Reports** dialog allows you to produce reports on the input options for your project and for the output results. You have access to this dialog by selecting **File | Reports** from the menu or by clicking on the **Reports** 

menu toolbar button.



Reports dialog

The **Reports** dialog uses a two-pane view. From the tree located on the left side of the dialog, you can chose the type of report to generate.

Once you have selected a report type from the tree, it will be displayed in the preview pane of the viewer. If you select the **Results Summary** report, the Advanced Filter wizard, which allows you to specify the information to be displayed in the report, will be automatically displayed.

The following options are found in the **Reports** dialog:

- You can select the zoom percentage (size to view the report) in the **Reports** dialog by selecting a display size from the drop down list.

- You can also search the report for specific keywords of your choice. Simply enter the text in the **Search** text box and click on the button. If that text is found within that specific report, it will be highlighted by a blue hatched box. Repeated clicking of the button will locate the next occurrence of the specified text in the document, until the all items are found. Any text you enter into the Search text box will be saved, allowing you to select it from the drop down list when you wish to perform another search.

The following buttons are found in the **Reports** dialog:

- Displays the first page of the report

- Displays the previous page of the report

- Displays the next page of the report

- Displays the last page of the report

- Allows you to print the report

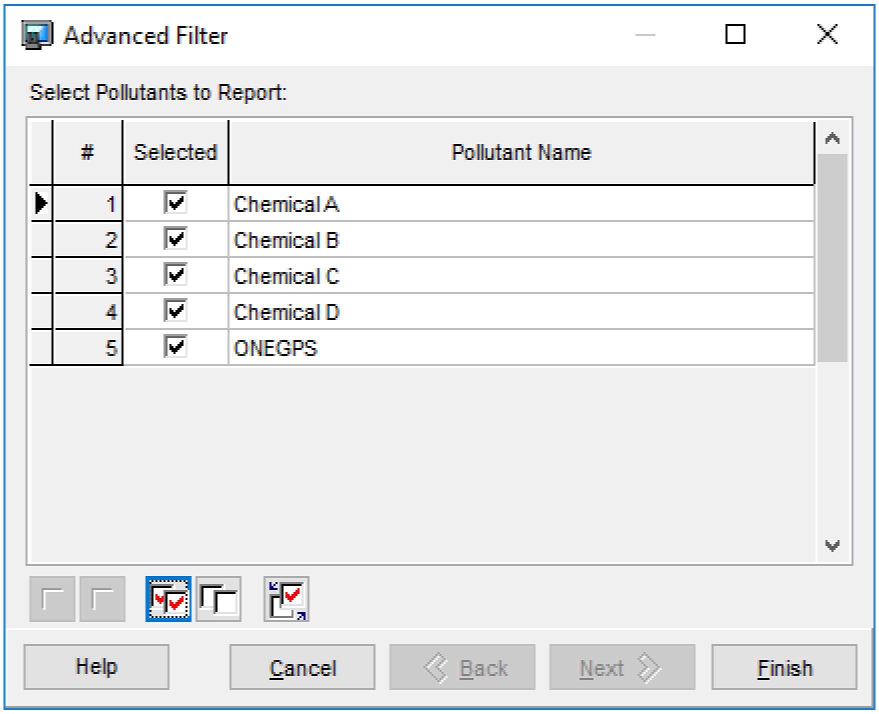
- Displays the Export dialog which allows you to select the format and destination you wish to export the report to.

Filter... - Click on this button to display the Advanced Filter wizard.

> **Advanced Filter** wizard is only available if you select the **Results Summary** report. This report contains results that can be filtered.

Advanced Filter

The **Advanced Filter** Wizard allows you to select the information you would like displayed in your reports. This wizard is displayed when you select the report type to be generated from the Reports dialog, or by clicking on the **Filter...** button in the Reports dialog. The options in this wizard allow you to generate reports that included any or all pollutants modeled in the project, including the Multi-Chemical Utility.

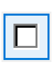


Advanced Filter dialog

How to Navigate Through the Wizard:

- Click through the Wizard using the **Next** and **Back** buttons.
- Click the **Finish** button to create the report.
- You can exit the wizard at any time without creating the report by clicking **Cancel**.
- The list of options is displayed in tabular format. In the **Selected** column, check the boxes of the options you wish to report. You can also use the following buttons to make your selections:

 Checks the selected item

 Unchecks the selected item

 Checks all the items in the list

 Unchecks all the items in the list

 Switches the selection of items

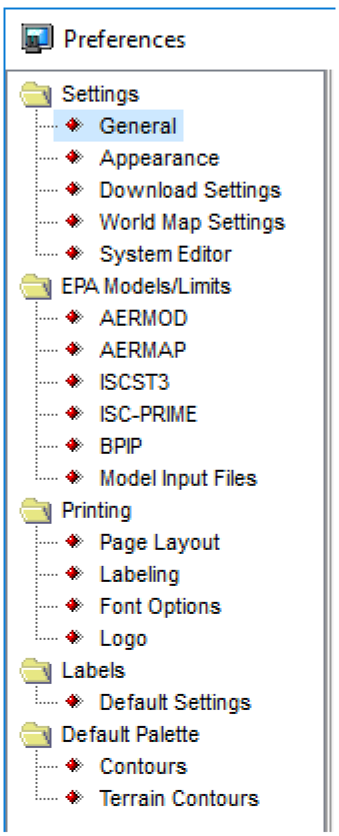
The wizard will display the appropriate screens for the type of report you are creating; not all screens will display for each report type.

Preferences

In the **Preferences** dialog you can define default options for your printouts and various other preferences for your project.

> In the **Preferences** dialog you define your global preferences. This means that the options you have selected in this dialog will be used as the default for every printout, for your current project or any other project you open.

You have access to the **Preferences** dialog by selecting **File | Preferences** from the menu or clicking on the **Preferences** button located in the Print Preview dialog.



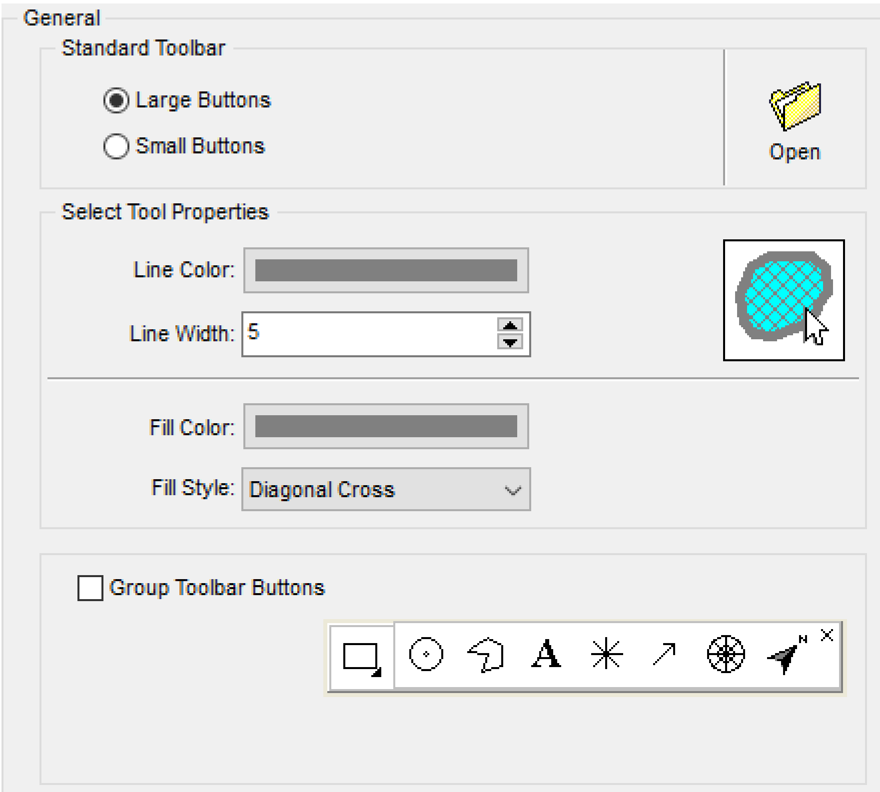
You have the following options in the **Preferences** dialog:

- General
- Appearance

- Download Settings
- World Map Settings
- System Editor
- EPA Models/Limits - AERMOD
- EPA Models/Limits - AERMAP
- EPA Models/Limits - ISCST3
- EPA Models/Limits - ISC-PRIME
- EPA Models/Limits - BPIP
- EPA Models/Limits - Model Input Files
- Page Layout
- Labeling
- Font Options
- Logo
- Default Settings
- Default Palette

General

The **General** page, of the Preferences dialog, allows you to select the display size of the **Menu Toolbar Buttons** and the **Select** tool properties.



Preferences - General

Standard Toolbar

Here you can select whether you wish to display large or small buttons on the **Menu Toolbar**.

Select Tool Properties

When an object in the drawing area is selected using the Select Tool it is outlined and filled for easy identification. In this panel you can select the following selection properties:

- **Line Color:** Click on the color bar to display the Color dialog from where you can select a color for the outline of the selected object.

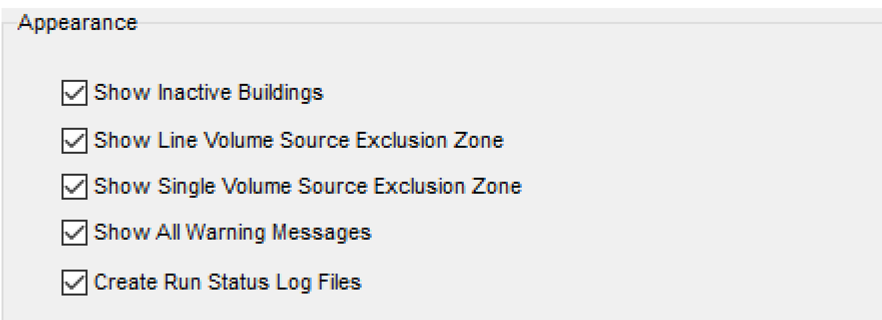
- **Line Width:** Here you can specify the width of the outline.
- **Fill Color:** Click on the color bar to display the Color dialog from where you can select a fill color for the selected object.
- **Fill Style:** From the drop-down list, select a fill style for the selected object.

Group Toolbar Buttons

Check this option to allow toolbar buttons to be grouped together based on common functionality. This allows for simplification of the Annotation Toolbar.

Appearance

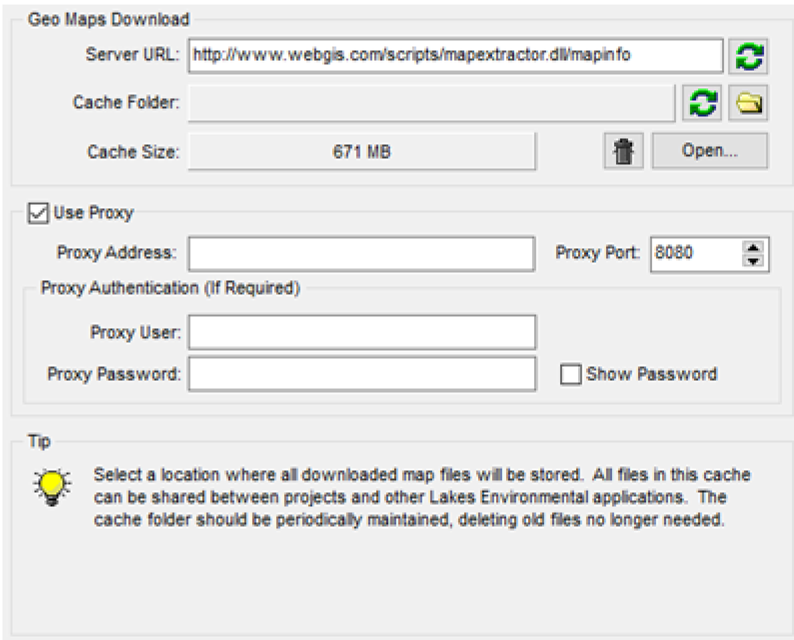
In the **Appearance** page, of the Preferences dialog, you have the option of hiding inactive settings so they will not be displayed on your drawing area and on your printouts. Simply check the appropriate box to hide.



Preferences dialog - Appearance

Download Settings

The **Download Settings** page of the Preferences dialog allows you to specify the location to where you wish to save downloaded maps.



Preferences dialog

Geo Maps Download

The **Geo Maps Download** panel displays:

- the map **Server URL** address from which USGS DEM and SRTM terrain data files are downloaded when the **WebGIS** (
) button in the Terrain Processor dialog is clicked
- the location of the **Map Cache Folder** where downloaded files are saved, and the size of the map cache

In the **Geo Maps Download** panel, you can:

- Change the location of the **Map Cache** folder by clicking the  button.

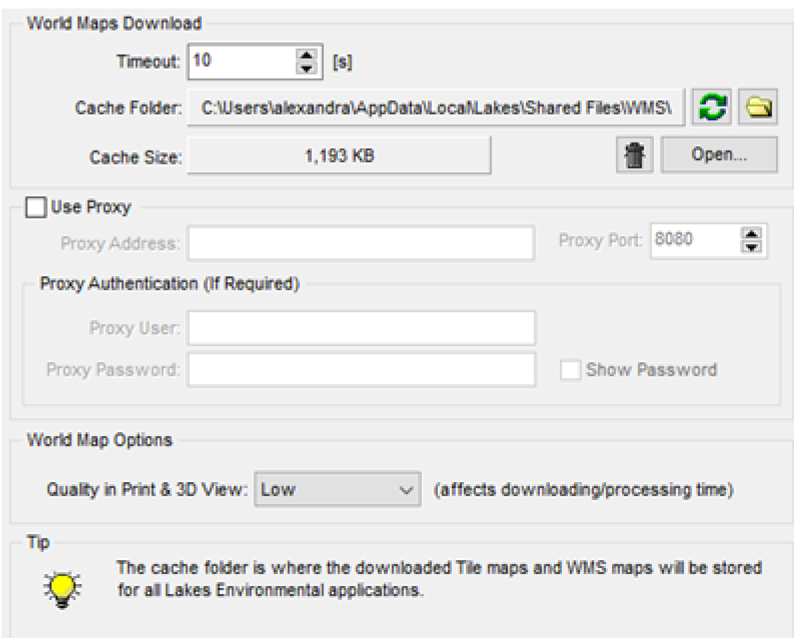
- Reset the location of the **Map Cache** folder to the default by clicking the button
- Open the cache folder to view or delete specific files by clicking the **Open** button.
- Clear all files from the cache by clicking the button.

Use Proxy

This option can be used if you have a proxy server and wish to specify the proxy address and port. Simply check the **Use Proxy** box to make this panel active and specify the required information.

World Map Settings

The **World Map Settings** page allows you to define the settings for the tile map download.



Preferences - World Map Settings

World Map Download

- **Time Out:** Time after which the program will stop attempting to download the map.

- **Cache Folder:** Folder in which the world maps will be stored.

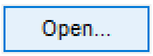
- **Cache Size:** Current size of the cache folder.

The following actions are available to you:

 Click this button to reset the cache folder to the default location.

 Click this button to select a different folder as cache.

 Click this button to delete the cache folder contents.



Click this button to open the cache folder in a browser.

World Map Options

- **Quality in Print & 3D View:** You can select the quality used to display **Tile Maps** in 3D View or any printed material. Default is set to be **Low** in order to minimize processing time. Note that higher map quality would result in the increase in processing time and the amount of data to be downloaded.

Use Proxy

This option can be used if you have a proxy server and wish to specify the proxy address and port. Simply check the **Use Proxy** box to make this panel active and specify the required information.

System Editor

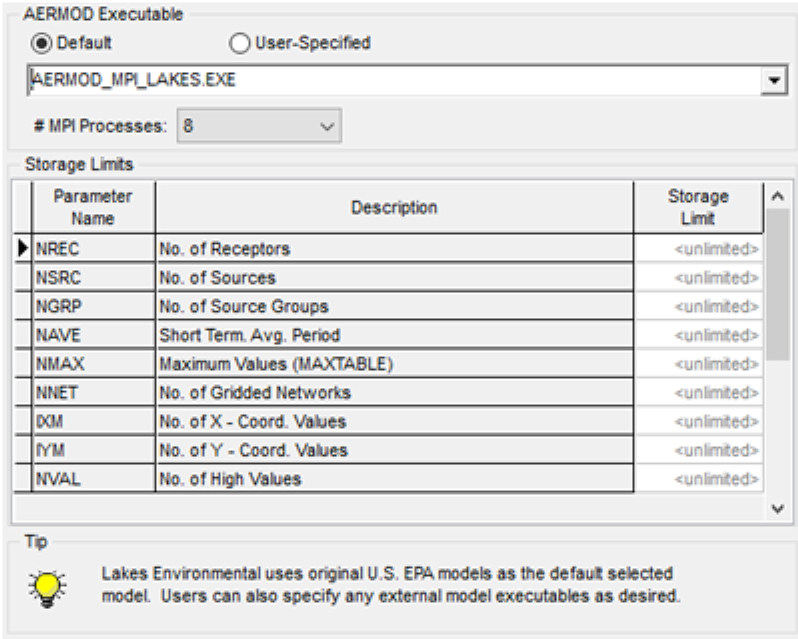
The **System Editor** page allows you to select text editor that will be used to view the generated output files.

- **MS WordPad:** selecting this option will display output files in MS WordPad.
- **MS Notepad:** Selecting this option will display output files in MS NotePad.
- **User-Specified Editor:** You may use this option to select any text editor that is installed on your machine.

EPA Models/Limits - AERMOD

The **EPA Models/Limits - AERMOD** screen of the Preferences dialog allows you to specify which AERMOD executable model should be used to run your project and displays the storage limits of the selected executable. The latest version of the U.S. EPA model AERMOD is supplied with the AERMOD View interface. You can also download the latest model from the U.S. EPA Website or from the Lakes Environmental Website.

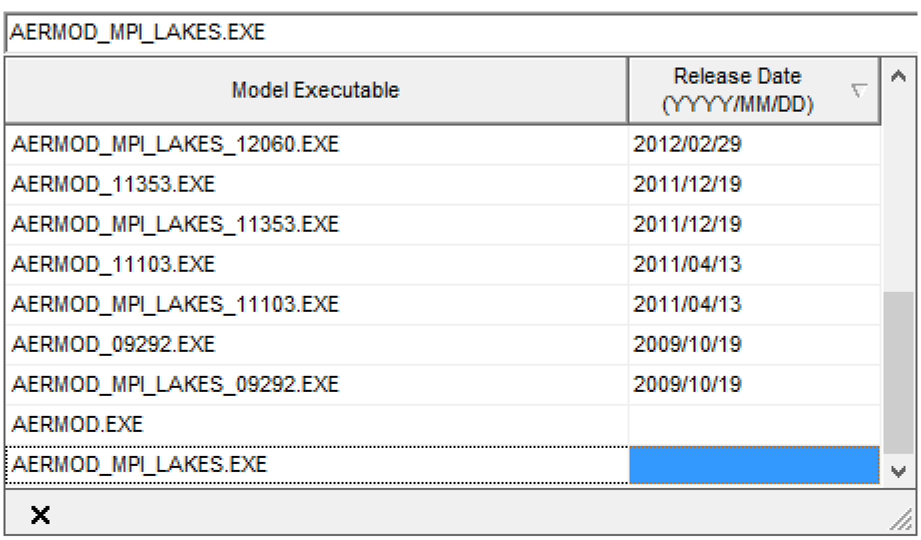
The U.S. EPA AERMOD model allocates data storage as needed based on the number of sources, receptors, source groups, and other input requirements, up to the maximum amount of memory available on a particular computer.



Preferences dialog - EPA Models/Limits - AERMOD

By default the original executable is specified for the model. The default executables are located in the **Models** folder of your AERMOD View installation directory.

> Selecting AERMOD.EXE or AERMOD_MPI_LAKES.EXE will ensure that you are running the latest executable. If you prefer to ensure that you are running the same model when you upgrade to the next version of AERMOD View, select the model that contains the version number (e.g. AERMOD_15181.EXE).



AERMOD Versions

The versionless executable is always the most current one. If you wish to select an older version you may do so, however this setting will be saved and if you wish to change to the most recent version after an upgrade you will have to do so manually. If you have the versionless executable selected, the latest version of the model will always be used after upgrades.

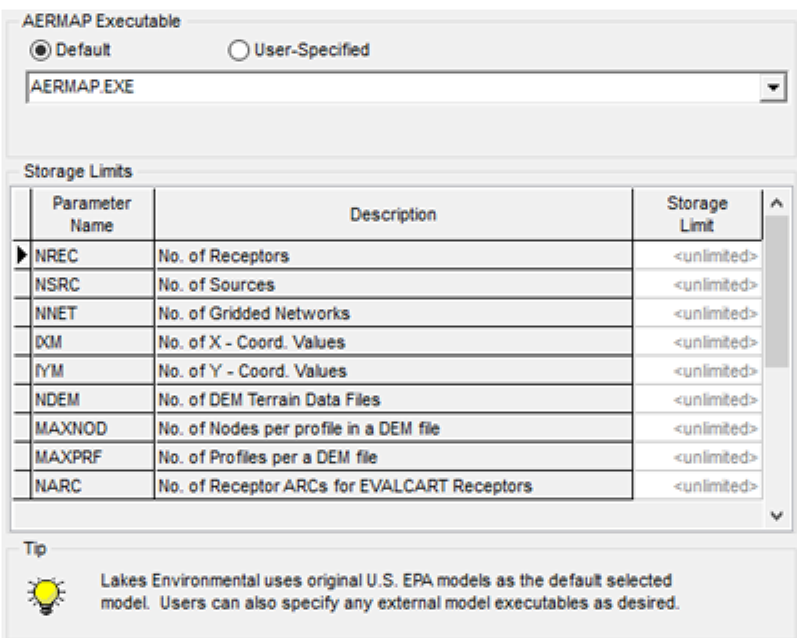
> See Also: About AERMOD MPI, which is an available default executable.

To select an alternative model executable, select the **User-Specified** option click on the button to specify the name and location of the model to be used. Please make sure to specify the new storage limits for all the parameters listed on the table. The storage limits for all user-specified model executables will be the same as the original by default, unless these limits are changed.

> All AERMOD View data validation is based on the storage limit values. Therefore, it is important that the correct limits be specified for the user-specified model executables.

EPA Models/Limits - AERMAP

The **EPA Models/Limits - AERMAP** screen of the Preferences dialog allows you to specify which AERMAP executable model should be used for your project and displays the storage limits of the selected executable. The latest version of the U.S. EPA model AERMAP is supplied with the AERMOD View interface. You can also download the latest executable from the U.S. EPA Website or from the Lakes Environmental Website.



Preferences dialog - EPA Models/Limits - AERMAP

By default the AERMAP.EXE executable is specified. The default executable is located in the **Models** folder of your AERMOD View installation directory.

AERMAP.EXE

Model Executable	Release Date (YYYY/MM/DD)
AERMAP_11103.EXE	2011/04/13
AERMAP_MPI_LAKES_11103.EXE	2011/04/13
AERMAP.EXE	
AERMAP_MPI_LAKES.EXE	

AERMAP versions

The versionless executable is always the most current one. If you wish to select an older version you may do so, however this setting will be saved and if you wish to change to the most recent version after an upgrade you will have to do so manually. If you have the versionless executable selected, the latest version of AERMAP will always be used after upgrades.

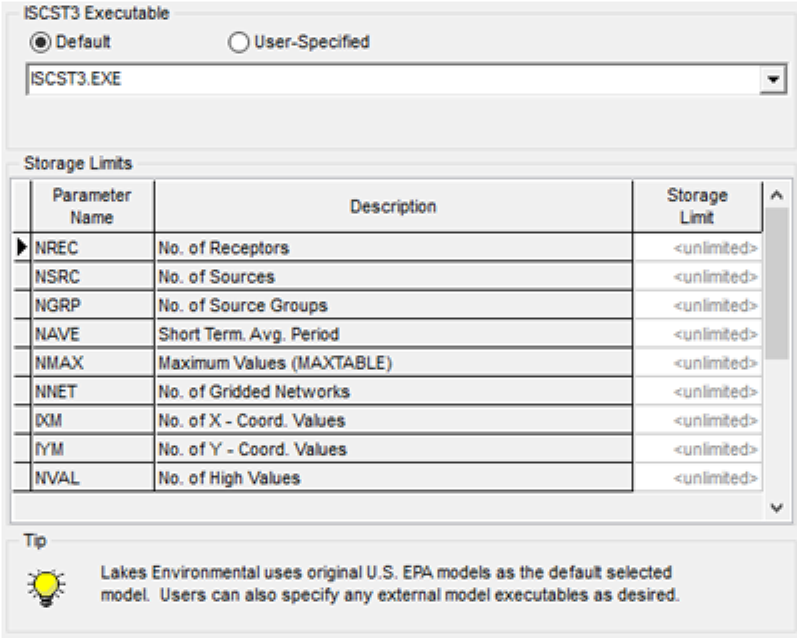
See Also: The AERMAP Model

To select an alternative model executable, select the **User-Specified** option click on the ... button to specify the name and location of the executable to be used. Please make sure to specify the new storage limits for all the parameters listed on the table. The storage limits for all user-specified model executables will be the same as the original by default, unless these limits are changed.

EPA Models/Limits - ISCST3

The EPA Models/Limits - ISCST3 screen of the Preferences dialog allows you to specify which ISCST3 executable model should be used to run your project and displays the storage limits of the selected executable. The latest version of the U.S. EPA model ISCST3 is supplied with the AERMOD View interface. You can also download the latest model from the U.S. EPA Website or from the Lakes Environmental Website.

The U.S. EPA ISCST3 model allocates data storage as needed based on the number of sources, receptors, source groups, and other input requirements, up to the maximum amount of memory available on a particular computer.



Preferences dialog - EPA Models/Limits - ISCST3

By default the original EPA executable is specified for the model. The default executable is located in the **Models** folder of your AERMOD View installation directory.

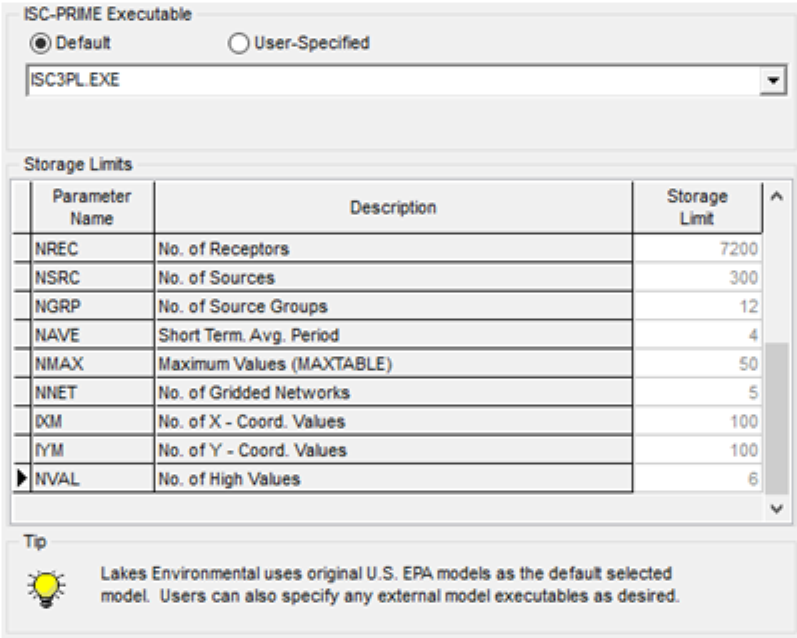
To select an alternative model executable, select the User-Specified option click on the button to specify the name and location of the model to be used. Please make sure to specify the new storage limits for all the parameters listed on the table. The storage limits for all user-specified model executables will be the same as the original by default, unless these limits are changed.

> All AERMOD View data validation is based on the storage limit values. Therefore, it is important that the correct limits be specified for the user-specified model executables.

EPA Models/Limits - ISC-PRIME

The **EPA Models/Limits - ISC-PRIME** screen of the Preferences dialog allows you to specify which ISC-PRIME executable model should be used to run your project and displays the storage limits of the selected executable. The latest version of the U.S. EPA model ISC-PRIME is supplied with the AERMOD View interface. You can also download the latest model from the U.S. EPA Website or from the Lakes Environmental Website.

Unlike the ISCST3 model, the original U.S. EPA model for ISC-PRIME has some limitations on some of the parameters such as the number of sources, receptors, and source groups. Due to these restrictions, a project containing more than 1200 receptors can not be run using the original U.S. EPA ISC-PRIME executable model (ISCP3.EXE).



Preferences dialog - EPA Models/Limits - ISC-PRIME

However, the U.S. EPA ISC-PRIME models make use of a static storage allocation design and the FORTRAN code for these models can be modified and recompiled to accept higher storage limits for many parameters. Lakes Environmental Software recompiled ISC3P.EXE to accept higher storage limits and the new executable was renamed to PRIMEL.EXE which is used as the default model. The storage limits for the recompiled code (PRIMEL.EXE) are defined in the Storage Limits tab of the Preferences dialog.

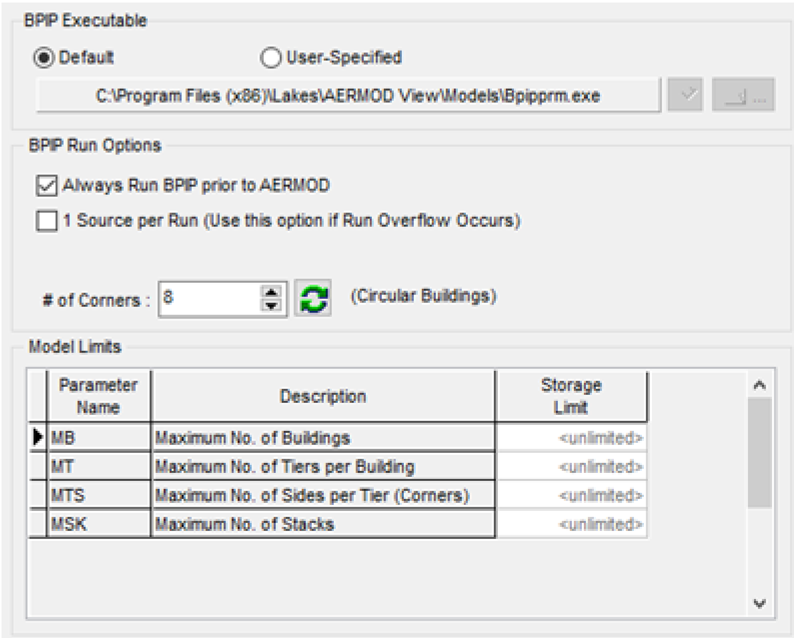
The original U.S. EPA model is located in the **Models** folder of your AERMOD View installation directory.

To select an alternative model executable, select the **User-Specified** option click on the  button to specify the name and location of the model to be used. Please make sure to specify the new storage limits for all the parameters listed on the table.

> All AERMOD View data validation is based on the storage limit values. Therefore, it is important that the correct limits be specified for the user-specified model executables.

EPA Models/Limits - BPIP

The **EPA Models/Limits** screen of the Preferences dialog allows you to specify which BPIP model executable should be used to run your project and displays the storage limits of the selected executable. The latest version of the U.S. EPA model BPIP-PRIME (BPIPPRM 04274) is supplied with the AERMOD View interface. You can also download the latest model from the U.S. EPA Website or the [Lakes Environmental Website](#).



Preferences dialog - EPA Models/Limits

BPIP Executable

> The BPIPPRM 04274 model runs BPIP and BPIP-PRIME.

By default the BPIP-PRIME EPA executable (04274) is specified for the model. The default executable is located in the **Models** folder of your AERMOD View installation directory. To select an alternative model executable, select the **User-Specified** option click on the  button to specify the name and location of the model to be used.

BPIP Run Options

You can specify the following options for BPIP:

- **Always Run BPIP prior to :** Check this option if you wish BPIP to run automatically every time model is run if buildings are present. This option provides a means to avoid using out of date building downwash data.
- **1 Source per Run (Use this option if Run Overflow Occurs):** Check this option if the number of buildings (e.g., >200) being processed causes the BPIP run to fail.

- **# of Corners (Circular Buildings):** The default number of vertices in a polygon used to define a circular building when writing the BPIP Input File.

Model Limits

The **Model Limits** panel displays the maximum number of buildings, tiers, corners and stacks that a particular model executable supports. The U.S. EPA BPIP-PRIME model allocates data storage as needed, based on the number of buildings, tiers, corners and stacks, up to the maximum amount of memory available on a particular computer.

Parameter Name	Parameter	Storage Limit BPIP-PRIME Version 04274
MB	Maximum No. of Buildings	Unlimited
MT	Maximum No. of Tiers per Building	Unlimited
MTS	Maximum No. of Sides per Tier	Unlimited
MSK	Maximum No. of Stacks	Unlimited

If specifying a **User-Specified** model, please make sure to specify the new storage limits for all the parameters listed on the table. The storage limits for all user-specified model executables will be the same as the original by default, unless these limits are changed.

> All AERMOD View data validation is based on the storage limit values. Therefore, it is important that the correct limits be specified for the user-specified model executables.

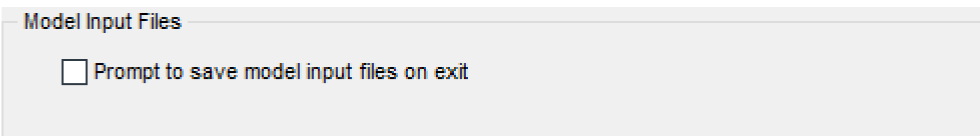
With the release of the BPIPPRM 04274 model, new input file flags were introduced. If you try to use older BPIP or BPIP-PRIME models, the new flags will not be recognized and you will not be able to run your project. The **Advanced User Only** panel circumvents this problem by recognizing that an older U.S. EPA BPIP model is being used and the appropriate input file flags utilized.

See Also: BPIP Input File Format

EPA Models/Limits - Model Input Files

The **EPA Models/Limits - Model Input Files** screen of the Preferences dialog allows you to specify whether the model input file should be updated automatically on exit, or should be updated only when you view the input file or run the model from within AERMOD View.

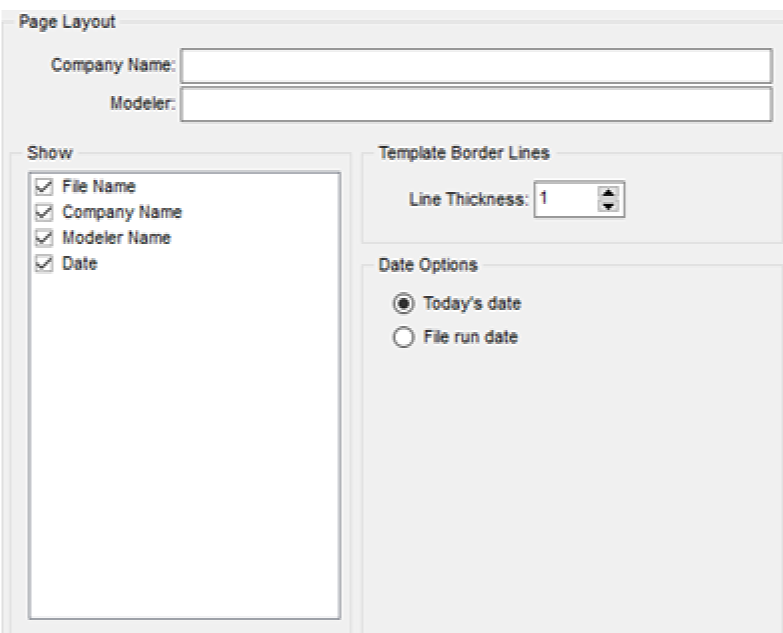
If you are using AERMOD View to prepare the model input file, but running the model outside of AERMOD View (e.g. using Batcher), check this box. If you are running the model from AERMOD View, you may leave the box unchecked.



Preferences dialog - EPA Models/Limits - Model Input Files

Page Layout

The **Page Layout** page, of the Preferences dialog, allows you to specify the information that will appear on your printouts.



Preferences dialog - Page Layout

Page Layout

- **Company Name:** Specify the name of your company to be printed on the printout template.
- **Modeler:** Type the name of the modeler to be printed on the printout template.

Show

From the list box you can select what information you would like displayed on your printout. Check the appropriate box to display the desired information on the printout.

Template Border Lines

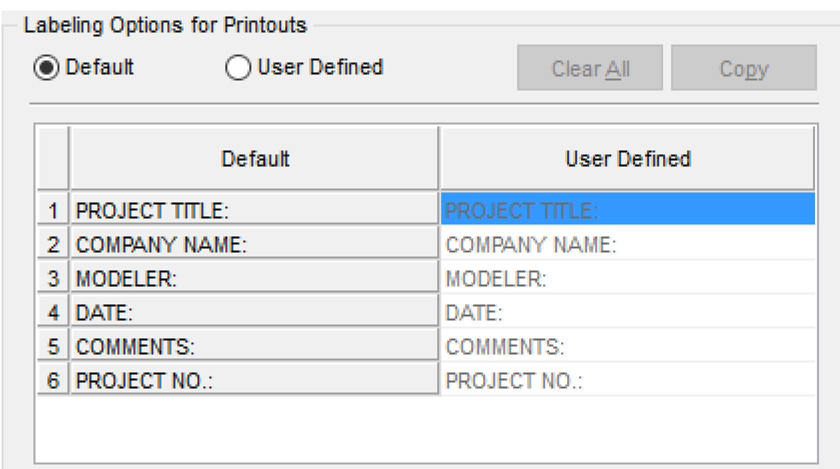
- **Line Thickness:** This is the thickness of all the boarder lines for the printout templates.

Date Options

- **Today's Date:** The current date.
- **File Run Date:** The date the project was run.

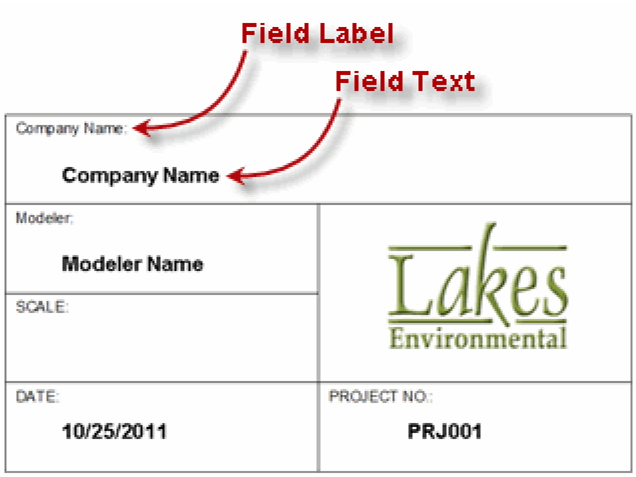
Labeling

In **Labeling** page, of the Preferences dialog, you can specify the labeling options for the printout templates.



Preferences dialog - Labeling

- **Labeling Options for Printouts:** Allows you to select the **Default** or **User Defined** options. If you select **User Defined**, you can change the default label titles for the various sections of the printout layout.

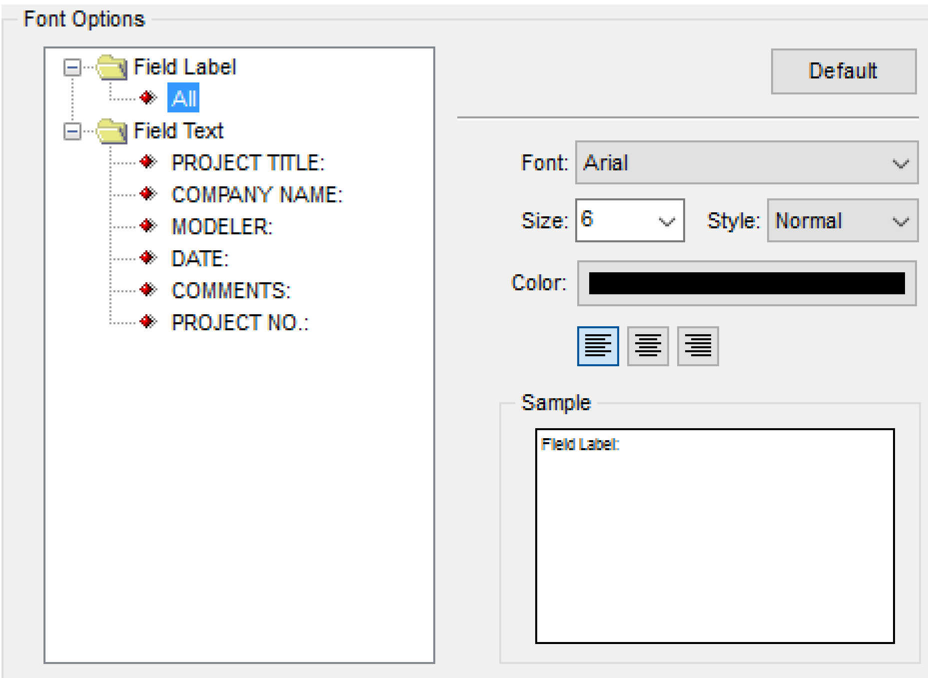


Clear All Deletes the labels in the fields of the **User Defined** column

Copy Copies the default labels into the fields of the **User Defined** column

Font Options

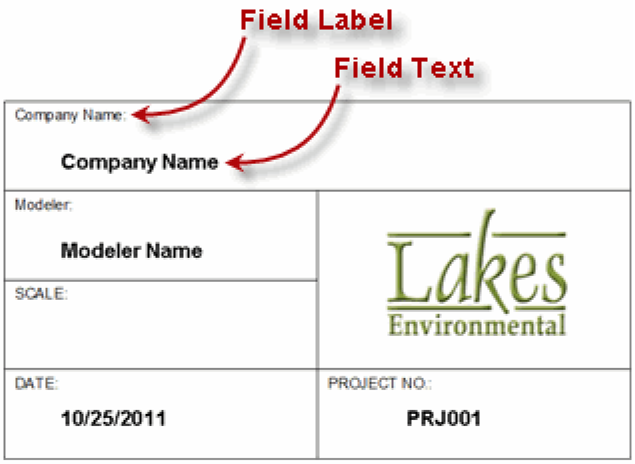
The **Font Options** page, of the Preferences dialog, allows you to specify the font, size, style, color and position of the field labels and field text for your printout.



Preferences dialog - Font Options

The following options are available:

- **Font Options - Tree View:** The tree view displays the following options -
    - **Field Label:** These are the labels that describe the field content in the printout template boxes. Font Options will be applied to all field labels.
    - **Field Text:** These are the labels used for the content of the printout template box. Under the Field Text tree item you can select the field text for which you would like to change font options.



- **Default:** The **Default** button returns all your font settings to the default. Click on this button to display a floating menu from where you can select **All** or **Selected**. If you choose **All**, then default settings will be applied to the Field Labels and all the Field Text. If you choose **Selected**, only the fields you have selected from the tree list will be returned to the default. You can select items from the tree list by pressing down the Shift key while you select multiple items in sequence or the Ctrl key to make disjoint selections.

- **Font:** Select a font from the drop-down list.

- **Size:** Select a font size from the drop-down list.

- **Style:** Select a font style from the drop-down list.

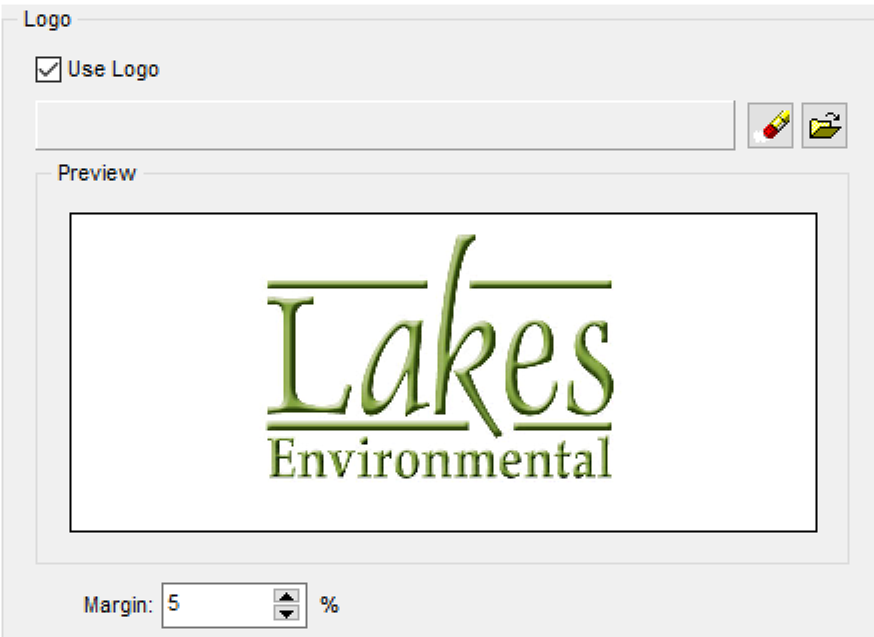
- **Color:** Click on the color bar to open the Color dialog from where you can select a color for your text.

- **Position:** Click on one of the three buttons to select the alignment of your text. The text can be either left, center or right aligned.

- **Sample:** As you make selections for color, font, size, and style for your text, the sample is updated to reflect the changes.

Logo

The **Logo** page, of the Preferences dialog, allows you to import an graphic to be used as your logo. This logo will be displayed on the bottom right side of your printout.



Preferences dialog - Logo

Check the **Use Logo** box to make the **Specify File** panel active. Click on the **Specify File** () button and specify the name and location of the graphic file you will be using.

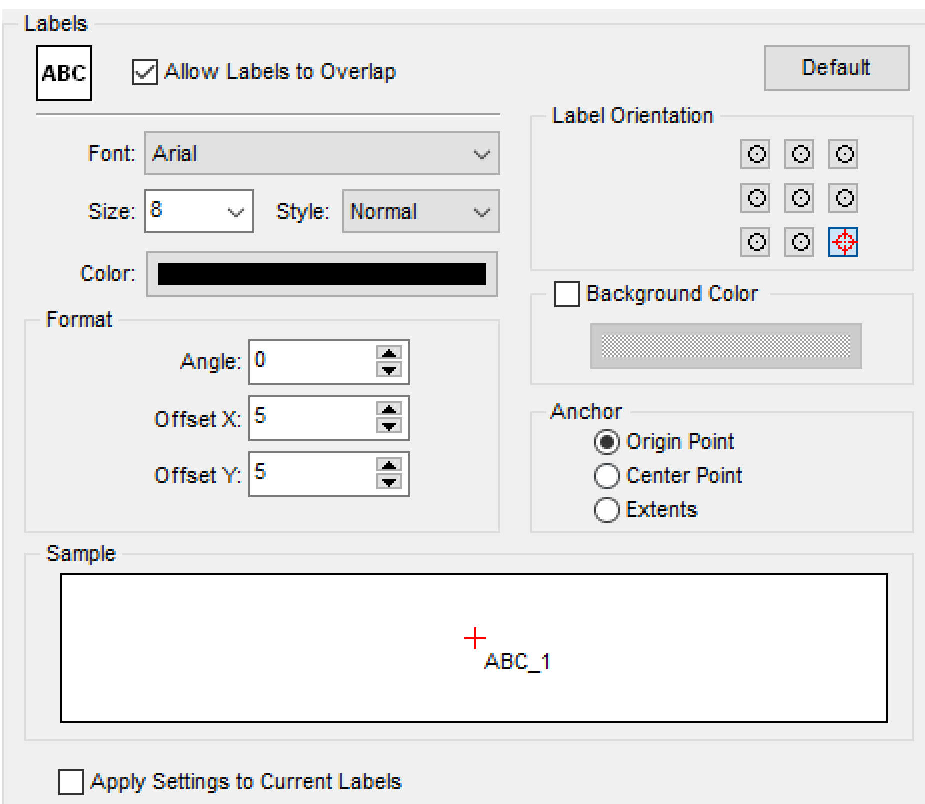
> The path displayed above the **Preview** panel indicates the folder where the image will be stored for AERMOD View.

The imported graphic will be displayed in the **Preview** panel. Several file formats are supported: gif, jpg, jpeg, bmp, emf, and wmf.

The **Margin** option lets you specify how much white space you would like surrounding your logo. The default margin is 5%, you can increase or decrease this at any time.

Default Settings

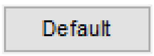
In the **Default Settings** page, of the [Preferences](#) dialog, you control the font type, font size, style, and color of the labels for all the objects in your drawing area such as your sources or buildings.



Preferences dialog - Default Settings

> If, during your project, you would like to change the labels for a particular object you can do so in the Labels page of the Graphical Options dialog. Here you can, for example, change the label settings of the buildings, while leaving the labels for the other objects at the default settings you have defined here.

The following options are available:

- **Allows Labels to Overlap:** Check this box if you want to allow for overlapping labels.
- 

: Click on this button to return all your font settings to the default.

Font Settings

- **Font:** Select a font from the drop-down list.
- **Size:** Select a font size from the drop-down list.
- **Style:** Select a font style from the drop-down list.
- **Color:** Click on the color bar to display the Color dialog from where you can select a color for the labels.

Label Orientation

- **Label Orientation:** Specify the location of the labels relative to the object marker by clicking on the button that best corresponds to the desired orientation.

Format

- **Angle:** If you would like your label placed at an angle relative to the marker, enter the angle value here.
- **Offset X:** This is the X distance between the marker and the label. The default is 5, but you can change this anytime.
- **Offset Y:** This is the Y distance between the marker and the label. The default is 5, but you can change this anytime.

Background Color

- **Background Color:** Check the **Background Color** box if you would like to change the background color just around the label. Click on the color bar to open the Color dialog from where you can select a background color.

Anchor

- **Origin Point:** Select this option if you would like the label anchored around the origin point. The origin point is the SW corner of the object, such as an area source.

- **Center Point**: Select this option if you would like the label anchored around the center point of the object.

- **Extents:** Select this option if you would like the label to be anchored in any one of the corners of the extents of the object. In this case, you can select the label orientation to specify which corners the label should be positioned (SW, NW, NE, or SE).

Sample

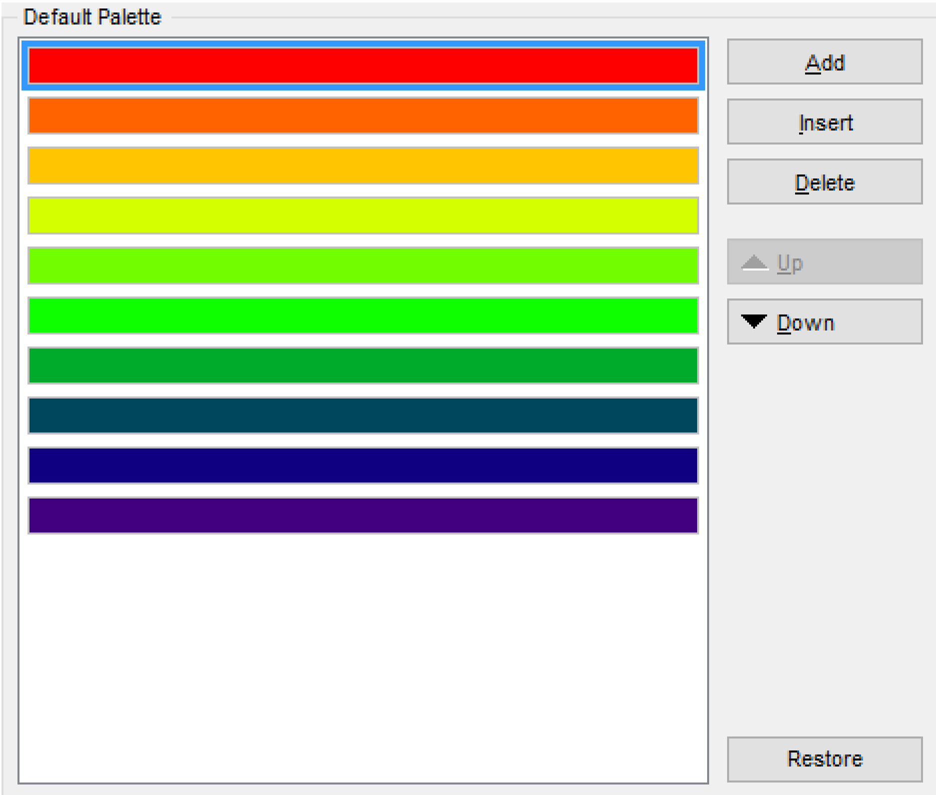
As you make selections for color, font, size, and style for your labels, the sample is updated to reflect the changes.

Checking the **Apply Settings to Current Labels** applies the changes you have made to your default settings for the labels in your current project, otherwise your settings will be applied only to future projects.

Default Palette

The **Default Palette** page, of the Preferences dialog, allows you to specify the colors you would like to use for the default palettes for your projects.

The selections available in the **Default Palette** page are the same, regardless of which palette you are changing the colors for. Changes to your colors are palettes specific. For example, if you change the colors for your Terrain Contours palette, you should note that the color settings of the other palettes will not be changed.

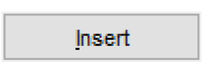


Preferences dialog - Default Palette

> If, during your project, you would like to change the colors for a particular default palette you can do so in the Levels page of the Graphical Options dialog.

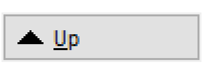
The **Default Palette** window displays the default colors for the selected palette. You can double click on a color bar to open the Color dialog where you can select an alternate color, or you can use the following buttons to customize the palette:

Press this button to display the Color dialog where you can select the desired color. The new color will be added at the end of the table.



Press this button to insert a color above the selected color.

Deletes the selected color from the custom palette.



Moves the selected color up one level in the color order.

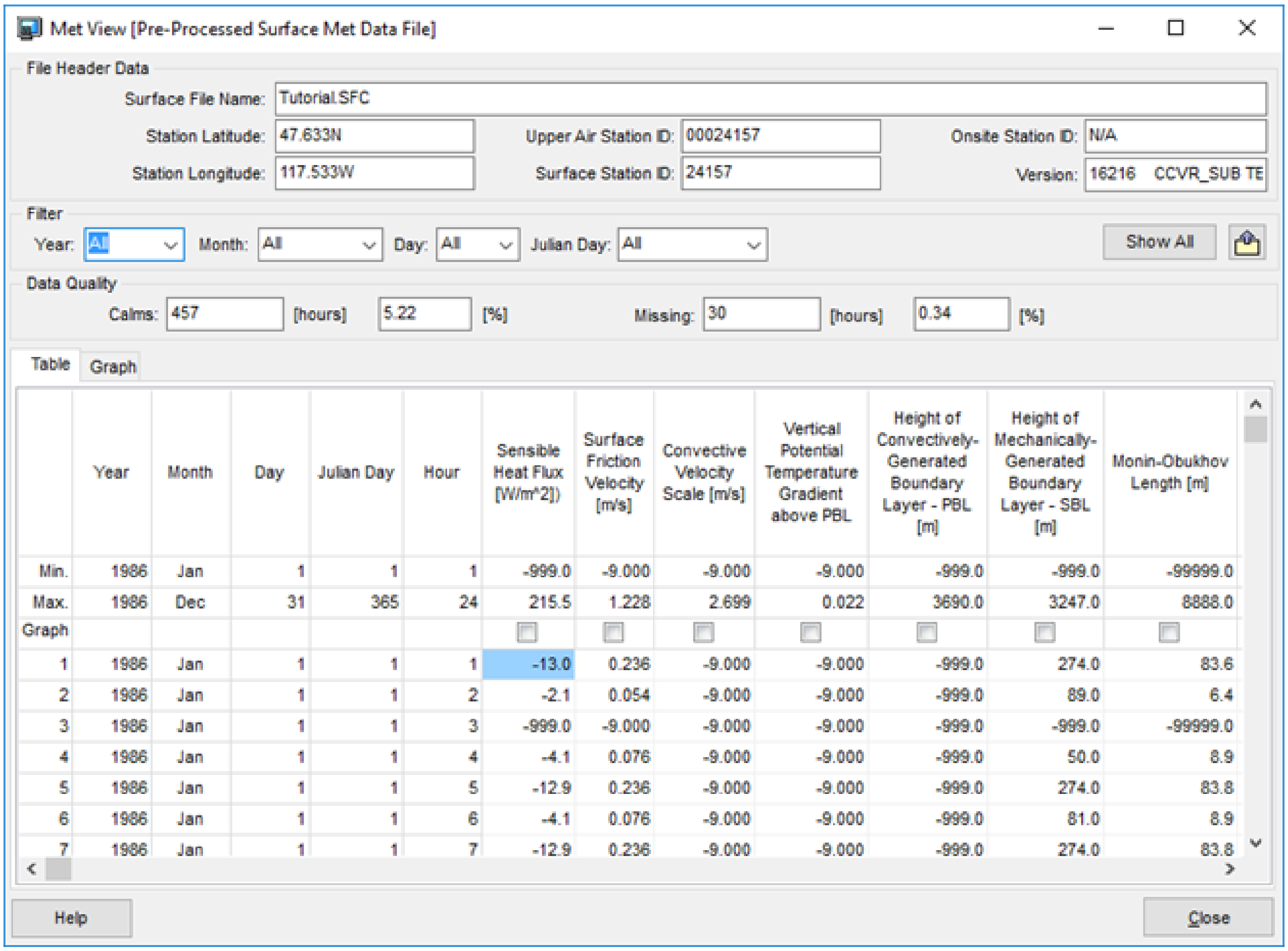
Moves the selected color down one level in the color order.

Click on this button to restore the program default palette.

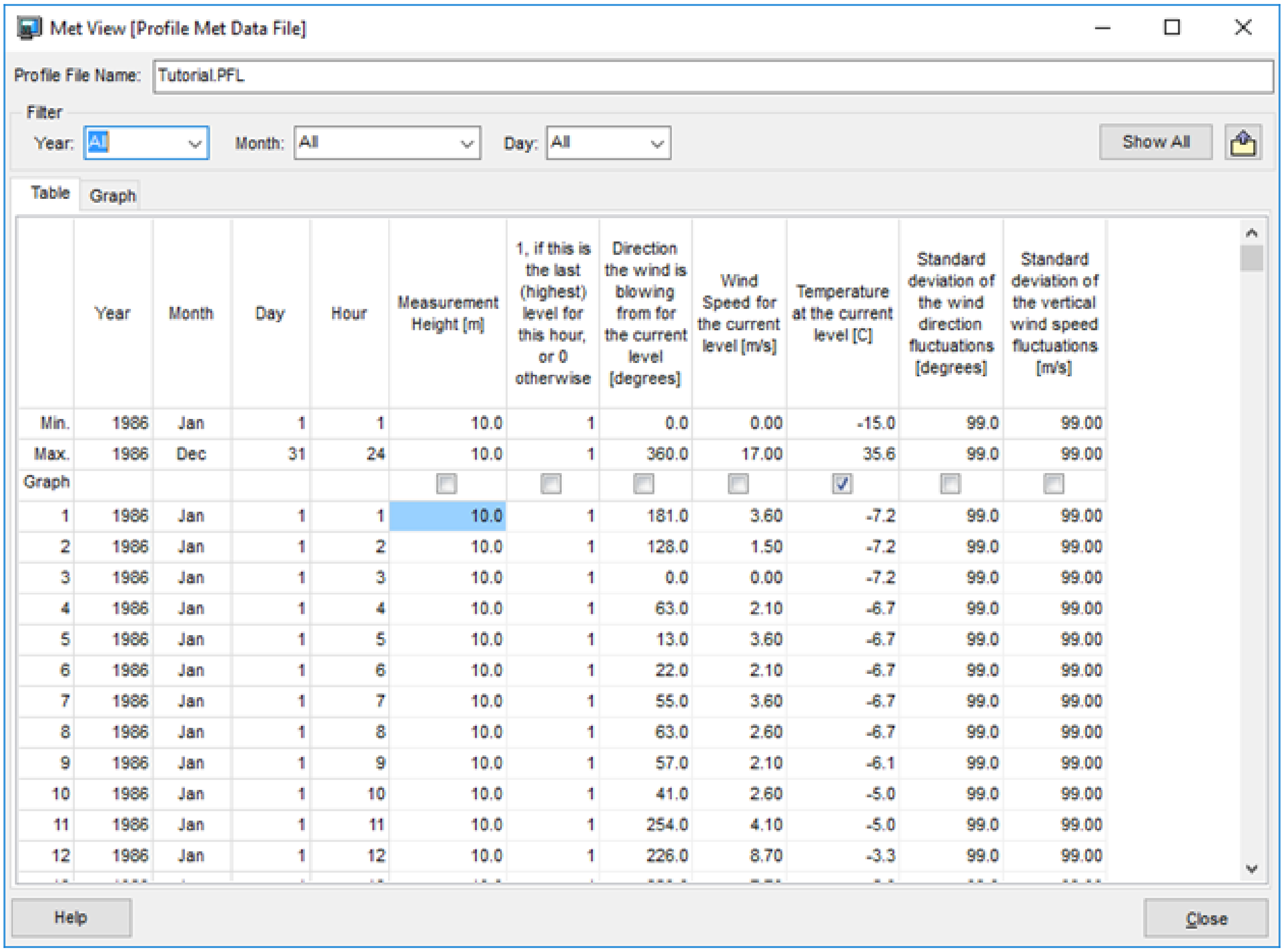
Accessory Information

Met View - Table

Met View is a utility that allows you to view the Surface (*.sfc) and Profile (*.pfl) files in a tabular or graphical format. You have access to **Met View** by pressing the **Preview** ( ) to the right of the meteorological file you wish to view. Select **Grid** option to load the **Met View** utility in tabular format.



Met View for Surface Met Data File - Table



Met View for Profile Met Data File - Table

File Header Data (For Surface Met Data File Only)

The **File Header Data** panel displays the name of your output file and the information contained in the header of the file.

Filter

Met View also allows you to refine the information displayed. You can do so using the selection boxes in the **Filter** panel. Available filters include:

- **Year**
- **Month**
- **Day**
- **Julian Day**

To show all the data available, press the **Show All** button.

You can export the data table to either CSV or Excel format by clicking on the **Export** ( ) button.

Data Quality (For Surface Met Data File Only)

This panel shows you the number of calm hours and missing hours along with their respective percentages in the data set.

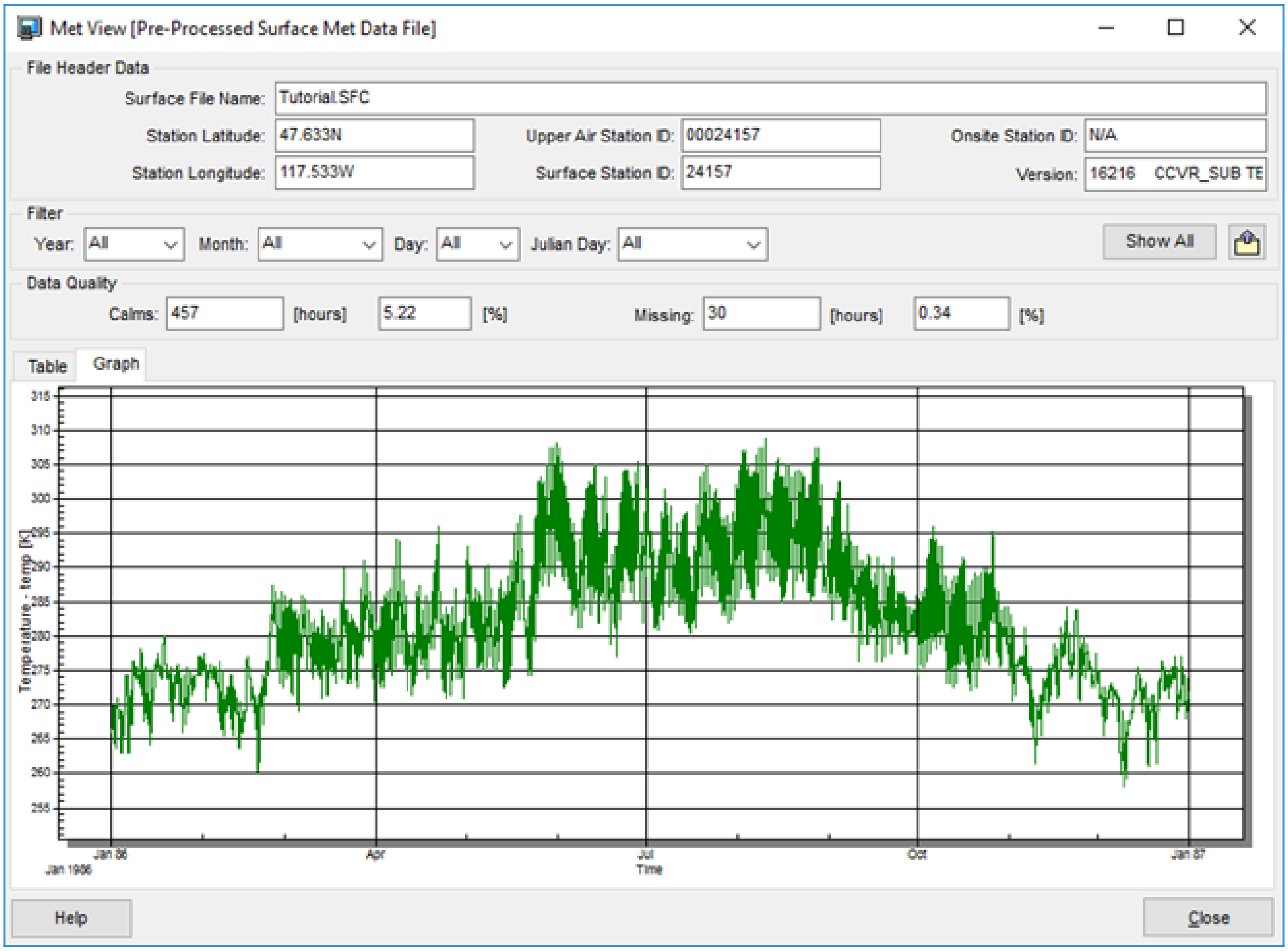
Table

Met View allows you view the data from a meteorological data file in a tabular format.

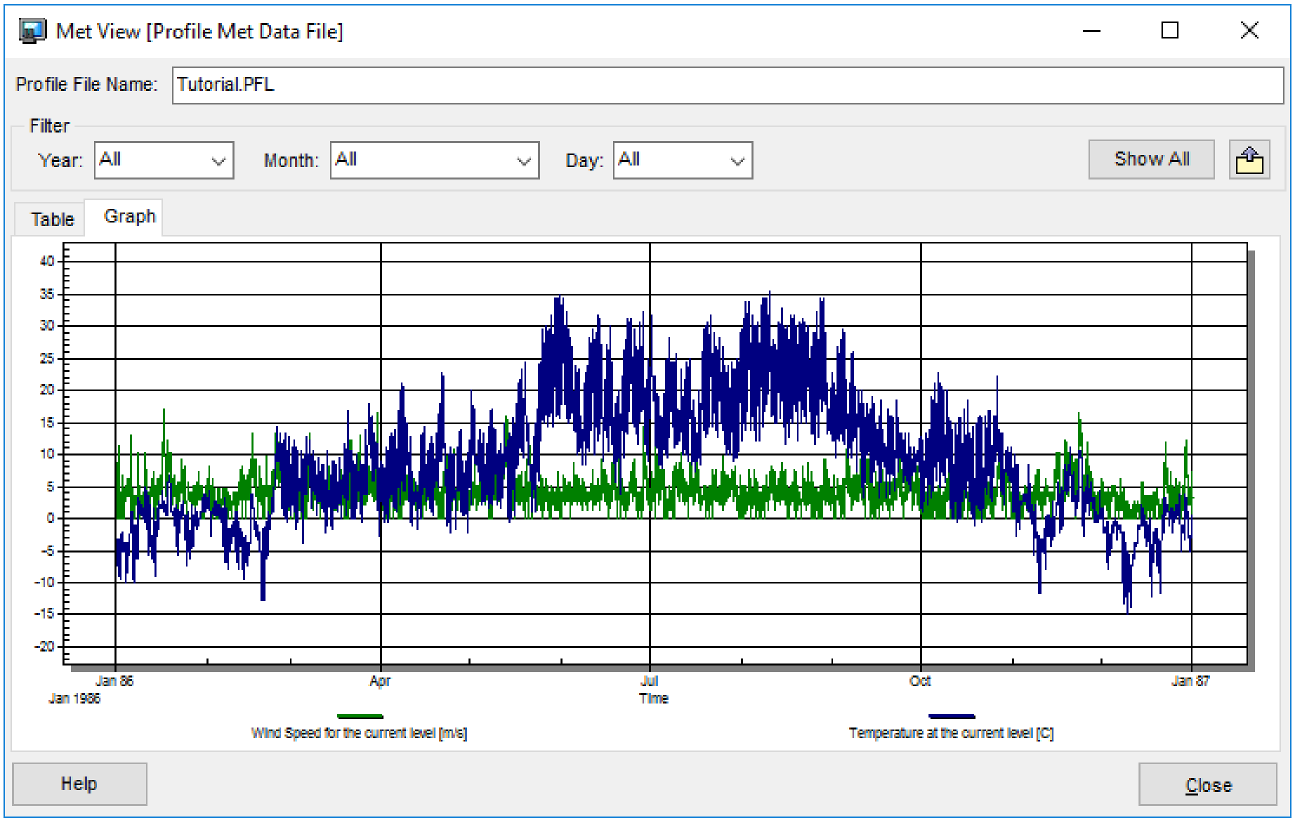
Note the third row down, labeled **Graph**. Checking the box in this row for any of the columns (parameters) will add data for that parameter to the graph. To view this data in graphical format, click the Graph tab.

Met View - Graph

Met View is a utility that allows you to view the Surface (*.sfc) and Profile (*.pfl) files in a graphical or tabular format. You have access to **Met View** by pressing the **Preview** ( ) to the right of the meteorological file you wish to view. Select **Grid** option to load the **Met View** utility in tabular format and then the **Graph** tab.



Met View for Surface Met Data File - Graph



Met View for Profile Met Data File - Graph

File Header Data (For Surface Met Data File Only)

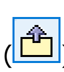
The **File Header Data** panel displays the name of your output file and the information contained in the header of the file.

Filter

Met View also allows you to refine the information displayed. You can do so using the selection boxes in the **Filter** panel. Available filters include:

- **Year**
- **Month**
- **Day**
- **Julian Day**

To show all the data available, press the **Show All** button.

You can export the data table to either CSV or Excel format by clicking on the **Export** () button.

Data Quality (For Surface Met Data File Only)

This panel shows you the number of calm hours and missing hours along with their respective percentages in the data set.

Graph

Met View allows you view the data from a meteorological data file in a graphical format. The X-axis represents the timeline of the data file, whereas the Y-axis represents the meteorological parameters which correspond to the selected column(s) from the tabular format.

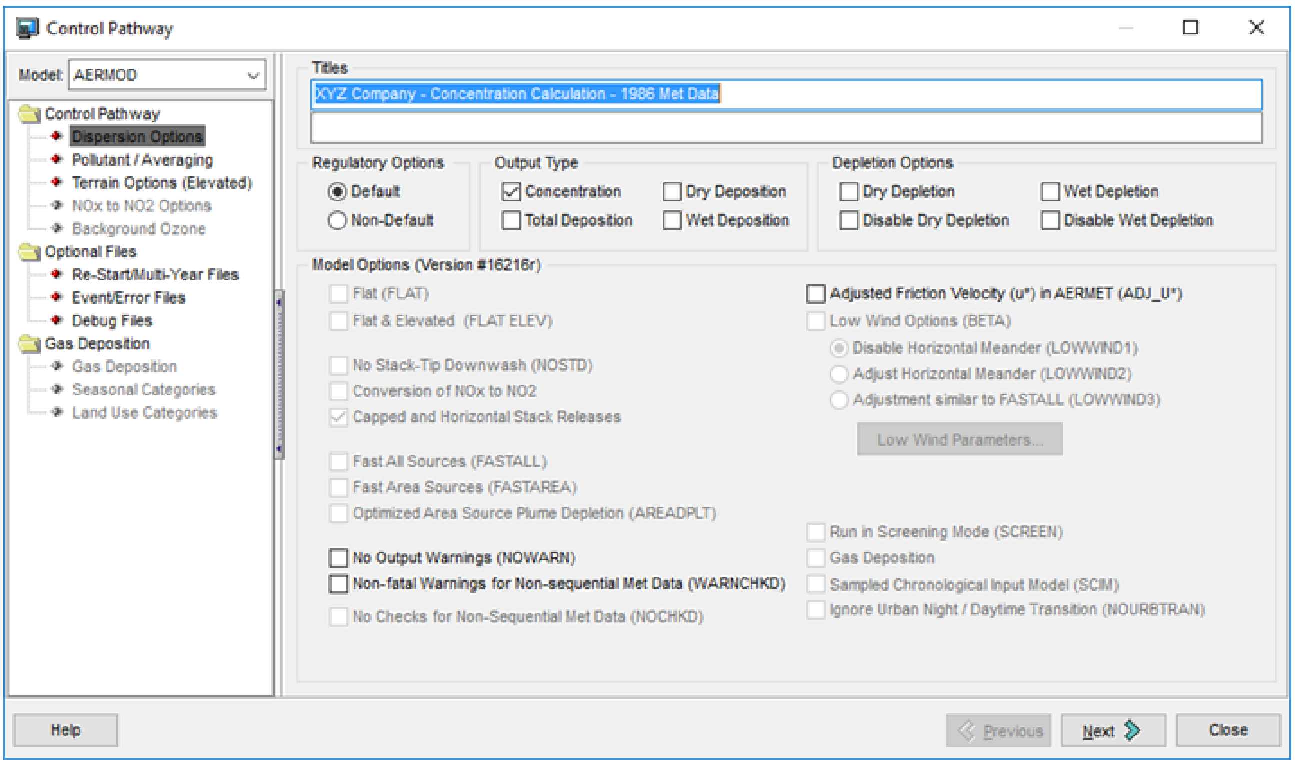
Control Pathway

The **Control Pathway** allows you to specify the overall job control options such as dispersion options, pollutant, and averaging times.

You have access to the **Control Pathway** dialog by select **Data | Control Pathway...** from the menu or by clicking on the **Control** 

menu toolbar button.

The **Control Pathway** dialog uses a two-pane view. The tree located on the left side of the dialog is used for navigation and item selection. Select an item (marked as ◆) in the tree to display the available options on the right panel.



Control Pathway dialog

As a rule, you should start inputting data first on this window, since most of the information requested here will be needed to complete other windows.

In the **Control Pathway** you define the following:

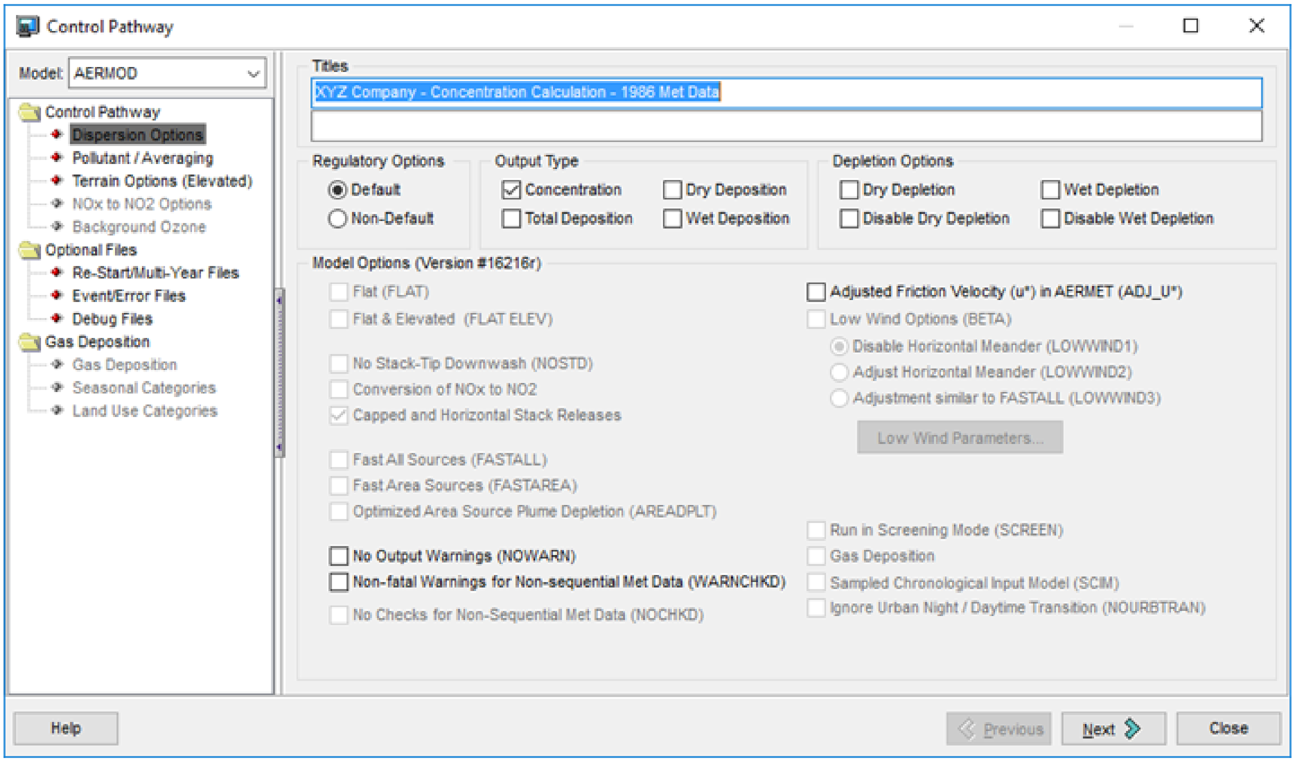
- Dispersion Options
- Pollutant/Averaging
- Terrain Options

- NOx to NO2 Options
- Background Ozone
- Re-Start/Multi-Year Files
- Event/Error Files
- Debug Files
- Gas Dry Deposition
- Seasonal Categories
- Land Use Categories

Dispersion Options

In the **Dispersion Options** page, you can specify project titles and dispersion options such as output type, plume depletion, and non-default options.

Select **Dispersion Options** from the tree located on the left side of the Control Pathway dialog, to display the available dispersion options.



Control Pathway dialog - Dispersion Options

The following options must be specified:

Titles

Here you can specify your project title. You may specify up to two lines of title information, each line accepts 68 characters including spaces. AERMOD View places the name and location of your project as the default title in the first title text box but you may change this at any time. The first title line is mandatory, you must specify a title here but the second title line is optional.

AERMOD

Regulatory Options

This section allows you to switch between **Default** and **Non-Default** model options. Different settings will enable different selections in the Model Options section.

Output Type

The Output Type section allows you to select which output type you want the model to produce.

Depletion Options

The Depletion Options allow you to select whether you wish to enable or disable wet/dry plume depletion.

Model Options

The Model Options section allows you to select individual options you wish to be included in the model run. The available options vary with selected model version and whether **Default** or **Non-Default** option is selected in **Regulatory Options** section.

ISCST3 and ISC-PRIME

Regulatory Default Options

When **Regulatory Default** is selected, only Output Type and Plume Depletion options are available.

Non-Default Options

The following options are available for both **ISCST3** and **ISC-PRIME**:

No Stack-Tip Downwash (NOSTD)

Checking this option will disable the stack tip downwash effect for point and flare sources.

Gradual Plume Rise

Check this option to model gradual plume rise throughout the project.

No buoyancy-induced dispersion

Check to disable modeling of plume dispersion caused by ambient turbulence.

Missing Data Processing Routine

Check to allow the model to bypass the fatal error for missing data contained in the meteorological data file. With this option checked, the model treats missing meteorological data in the same way as the calm processing routine.

Bypass the Calms Processing Routine

This option will incorporate zero concentrations associated with calm meteorological data hours into short-term averages.

The following options are available only for **ISCST3**:

Vertical term adjustment if HE > ZI

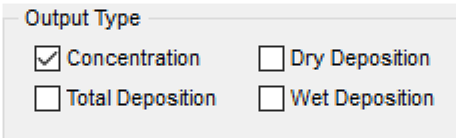
Check this option to enable vertical term adjustment if the plume (or effective stack) height is greater than mixing height.

TOXICS

Check this option to enable TOXICS dispersion modeling.

Output Type

For each model, as you specify your Dispersion Options you must specify the output type.



You may select any or all of the following output types:

- **Concentration:** This specifies that concentration values will be calculated. This is the default option and is automatically selected by AERMOD View.

- **Total Deposition:** Check this option to calculate total deposition (dry and wet) flux values.

- **Dry Deposition:** This option specifies that dry deposition flux values only will be calculated.

- **Wet Deposition:** Select this option to calculate wet deposition flux values.

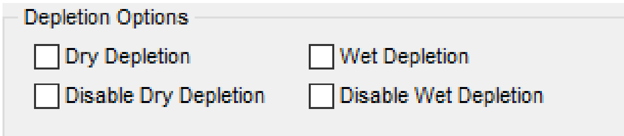
If you select the Total Deposition option or the Wet Deposition option, then you MUST define scavenging coefficients for particulate sources or gaseous sources on the Gas & Particle Data window.

When modeling with the Total Deposition option or the Dry Deposition option, you MUST include particle information for each source on the Gas & Particle Data window.

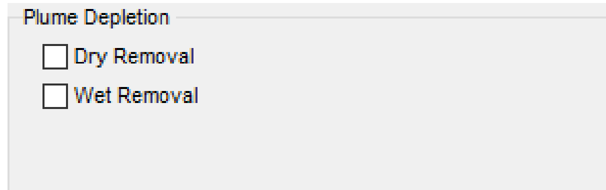
Depletion Options

When particles are modeled, a settling velocity and a deposition velocity are calculated for each size category. The settling velocity causes the plume to "tilt" towards the surface (if the plume is elevated) as it travels downwind, while the deposition velocity is used to calculate the flux of the matter deposited at the surface. When the plume depletion option is included, particle mass is removed from the plume as it is deposited on the surface, thereby conserving mass.

If plume depletion (either dry or wet removal) is not included, then the mass of particles deposited on the surface from gravitational settling and/or precipitation scavenging is not removed from the plume. The no depletion option may be acceptable if deposition is weak, and it will result in an overestimate of both concentrations and deposition.



Plume Depletion Options - AERMOD



Plume Depletion Options - ISCST3/ISC-PRIME

> The US EPA AERMOD model includes dry and/or wet depletion mechanisms (DRYDPLT and WETDPLT) in the calculated concentration or deposition flux values if the dry and/or wet deposition processes are considered.

You can select either one or none of the following plume depletion options -

- **Dry Depletion (Dry Removal):** Specifies that plume depletion due to dry removal mechanisms will be included in the calculations. When checked, specifically directs the model to model dry depletion.

> If using the **Default** regulatory option, this models particle dry depletion only. To model gaseous dry depletion as well you must select the **Non-Default** regulatory option **Gas Deposition**.

- **Disable Dry Depletion:** When checked, disables the dry depletion processes associated with the dry deposition algorithms. This will result in a more conservative estimate of concentrations and/or depostion fluxes invloving deposition processes.

- **Wet Depletion (Wet Removal):** Specifies that plume depletion due to wet removal mechanisms will be included in the calculations. When checked, specifically directs model to model wet depletion.

- **Disable Wet Depletion:** When checked, disables the wet depletion processes associated with the wet deposition algorithms. This will result in a more conservative estimate of concentrations and/or depostion fluxes invloving deposition processes.

> If the **Wet Depletion (Wet Removal)** option is selected, than you MUST specify scavenging coefficients in the Gas & Particle Data window. If the **Dry Depletion (Dry Removal)** option is selected, than you MUST specify particle information in the Gas & Particle Data window.

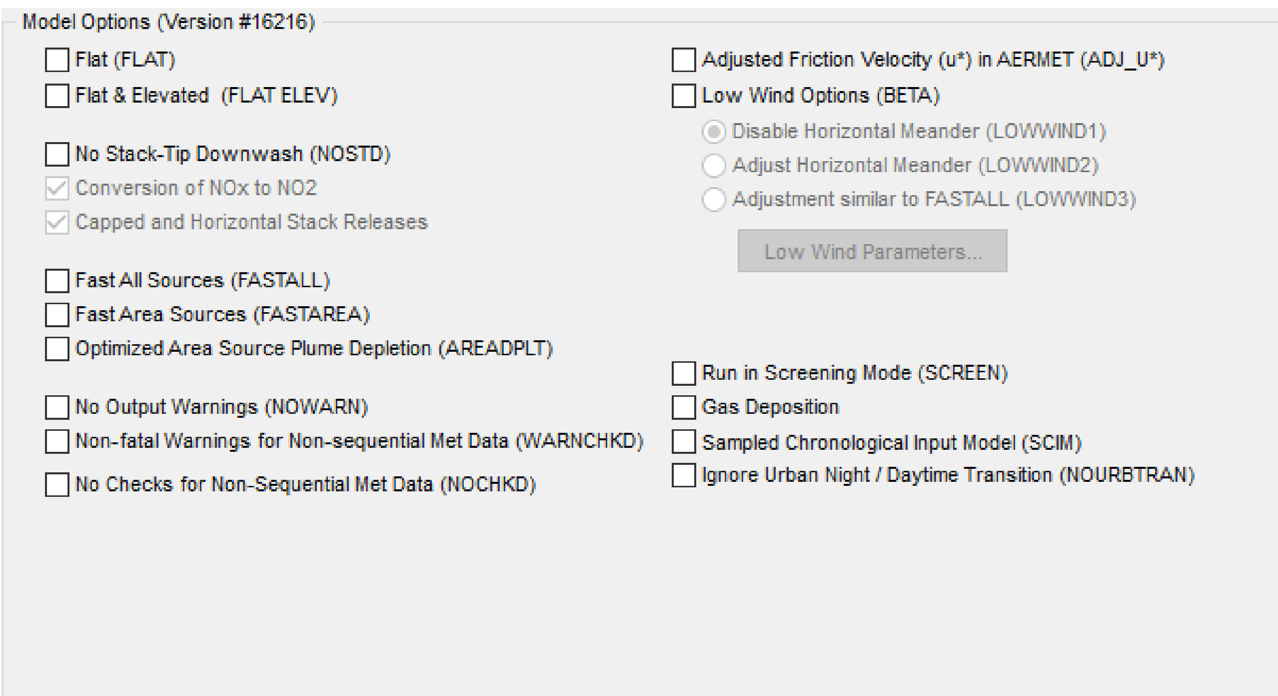
> Plume depletion calculations will result in significantly longer execution times for the model, since the model must integrate along the plume path between the

source and receptor. The amount of increase in execution time will vary depending on source characteristics and the terrain.

Model Options

AERMOD Only

The options displayed in this section are the same, regardless of which AERMOD version is selected in Preferences. Selecting a different AERMOD version and switching between **Default** and **Non-Default** in **Regulatory Options** section of Dispersion Options will determine which selections are enabled and which selections are listed as "BETA".



Control Pathway - Dispersion Options - Model Options

Flat (FLAT)

Checking this option will select the **Flat** option in Terrain Options. If it was previously selected, checking this option will deselect the **Flat & Elevated (FLAT ELEV)** option.

Flat & Elevated (FLAT ELEV)

Checking this option will select **Flat & Elevated** option in Terrain Options. If it was previously selected, checking this option will deselect the **Flat (FLAT)** option.

No Stack-Tip Downwash (NOSTD)

Checking this option will disable the stack tip downwash effect for point and flare sources.

Conversion of NOx to NO2

Checking this option will enable conversion of NOx species to NO_2.

> In model version 15181 and earlier, this is a non-default model option. For version 16216r, this option is regulatory default and is automatically checked when the pollutant ID is set to NO2.

Capped and Horizontal Stack Releases

Checking this option will enable simulation of plume rise for capped sources.

> In model version 15181 and earlier, this is a non-default BETA model option. For version 16216r, this is a regulatory default option and is automatically checked.

Fast All Sources (FASTALL)

Checking this option will optimize model runtime for POINT and VOLUME sources by skipping receptors that are more than a certain distance from the plume center line.

Fast Area Sources (FASTAREA)

Checking this option will optimize model runtime for AREA (all types) and OPENPIT sources.

Optimized Area Source Plume Depletion (AREADPLT)

Checking this option will optimize model runtime for AREA sources plume depletion due to dry removal.

No Output Warnings (NOWARN)

Checking this option will disable the printing of warning messages to the main output file.

Non-Fatal Warnings for Non-Sequential Met Data (WARNCHKD)

Checking this option will generate warning messages instead of fatal errors due to non-sequential meteorological data.

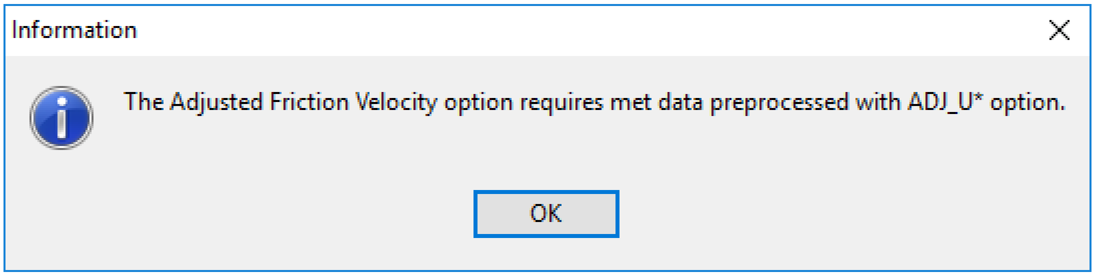
No Checks for non-Sequential Met Data (NOCHKD)

Checking this option will disable date checking for non-sequential meteorological data.

Adjusted Friction Velocity (u*) in AERMET (ADJ_U*)

Check this option if the AERMET met files were pre-processed using the **ADJ_U*** (adjusted friction velocity for low wind conditions) option.

If you do not have a met file currently selected, or if the file doe snot have the ADJ_U* flag, the following message will be shown:

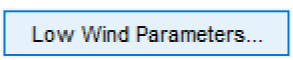


In model version 15181 and earlier, this is a non-default BETA model option. For version 16216r, this is a regulatory default option if the processed meteorological data does not include measured turbulence parameters (sigma-theta, sigma-w) in the profile file (.PFL).

Low Wind Options

Check this option to be able to enable the following modeling options:

- Disable Horizontal Meander (LOWWIND1)
- Adjust Horizontal Meander (LOWWIND2)
- Adjustment similar to FASTALL (LOWWIND3)

Click the 

button to load the Low Wind Parameters dialog to specify parameters for the currently selected LOWWIND condition.

Run in Screening Mode

This option forces the model calculations to represent values for the plume centerline regardless of the source-receptor-wind orientation. This option is included to estimate worst case impacts. Averaging periods are restricted to 1-hour when this option is selected.

Gas Deposition

Checking this option will enable you to specify the parameters for Gas Deposition.

Sampled Chronological Input Model (SCIM)

Check this option to run using only the annual averaging option to reduce model runtime. Additional parameters must be specified in Meteorology Pathway.

Ignore Urban Night/Daytime Transition (NOURBTRAN)

Check this option to disable the switch from urban boundary layer during nighttime hours to convective boundary layer during daytime hours.

> For more information about any of the options described above refer to the US EPA AERMOD User Guide.

Default Vertical Potential Temperature Gradients

The following table displays the Default Vertical Potential Temperature Gradients (K/m) -

Pasquill Stability Category	Rural	Urban
A	0.000	0.000
B	0.000	0.000
C	0.000	0.000
D	0.000	0.000
E	0.020	0.020
F	0.035	0.035

Default Wind Profile Exponents

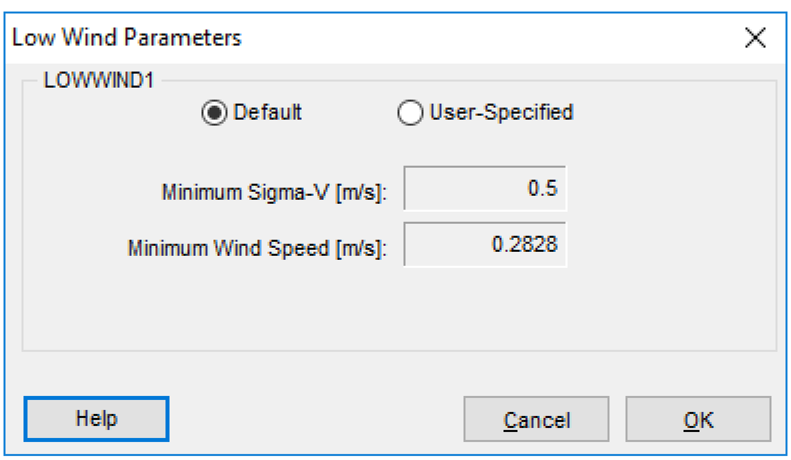
The following table displays the Default Wind Profile Exponents -

Pasquill Stability Category	Rural	Urban
A	0.07	0.15
B	0.07	0.15
C	0.10	0.20
D	0.15	0.25
E	0.35	0.30
F	0.55	0.30

Low Wind Parameters

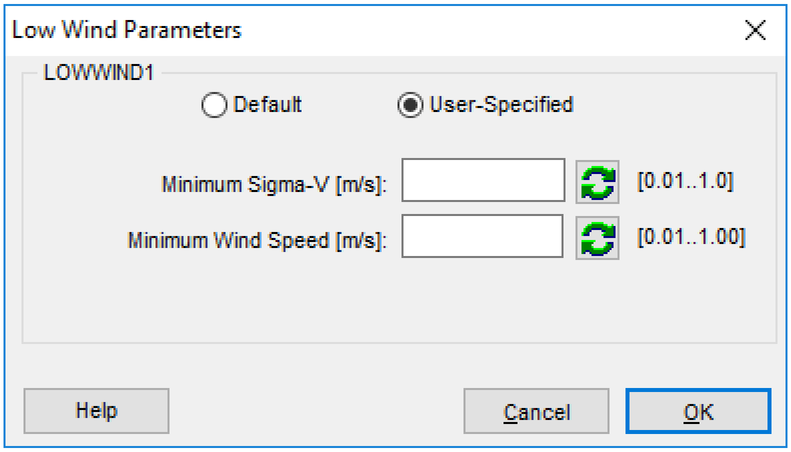
The **Low Wind Parameters** dialog allows you to set the **Low Wind Options** in Control Pathway.

Disable Horizontal Meander (LOWWIND1)



Low Wind Parameters - Disable Horizontal Meander - Default

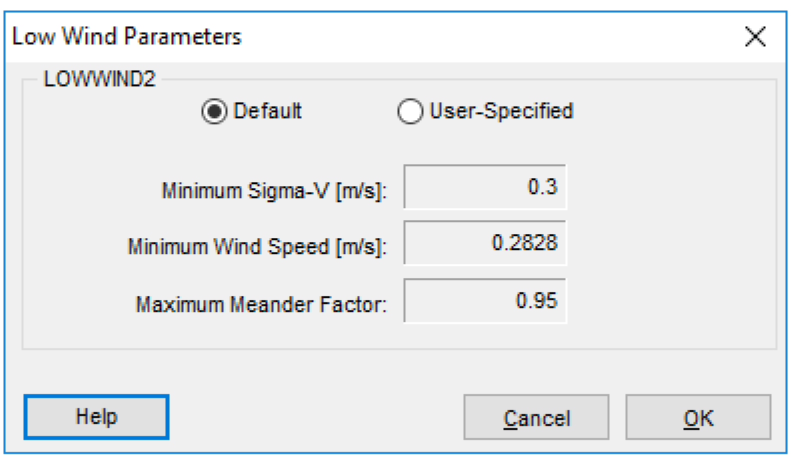
If **Disable Horizontal Meander (LOWWIND1)** is selected in the **Low Wind Options** of the [Control Pathway](), the default value for **Sigma-V** is increased from 0.2 to 0.5, **Minimum Wind Speed** is set to 0.2828 m/s, and **Maximum Meander Factor** (horizontal meander) is disabled. If the **Default** option is selected, data entry is disabled and the values cannot be changed. You may choose to enter custom values by selecting **User Specified** option. This will activate the fields to allow data entry.



Low Wind Parameters - Disable Horizontal Meander - User Specified

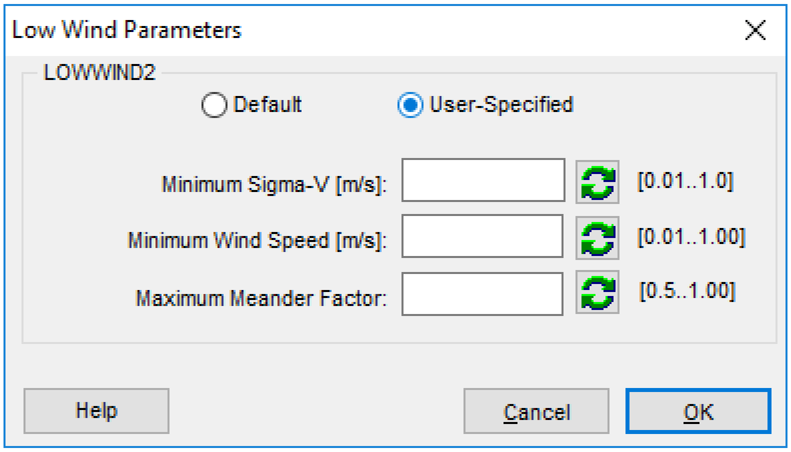
Enter values of **Minimum Sigma-V** and **Minimum Wind Speed**. Both values must be between 0.01 and 1.0. Click the  button to enter default parameters in the fields.

Adjust Horizontal Meander (LOWWIND2)



Low Wind Parameters - Adjust Horizontal Meander - Default

If **Adjust Horizontal Meander (LOWWIND2)** is selected in the **Low Wind Options** of the [Control Pathway](), the default value for **Sigma-V** is increased from 0.2 to 0.3, **Minimum Wind Speed** is set to 0.2828 m/s, and **Maximum Meander Factor** is activated. If the **Default** option is selected, data entry is disabled and the values cannot be changed. You may choose to enter custom values by selecting **User Specified** option. This will activate the fields to allow data entry.

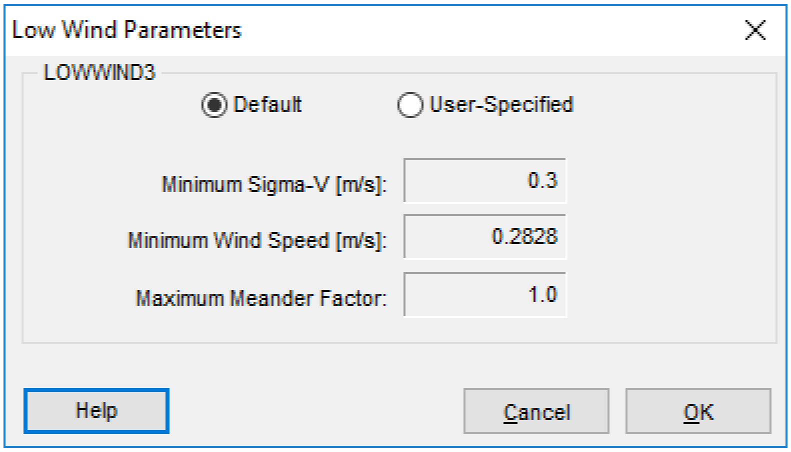


Low Wind Parameters - Adjust Horizontal Meander - User Specified

Enter values of **Minimum Sigma-V**, **Minimum Wind Speed**, and **Maximum Meander Factor**. **Minimum Sigma-V** and **Minimum Wind Speed** values must be between 0.01 and 1.0 and **Maximum Meander Factor** must be between 0.5 and 1.0. Click the  button to enter default parameters in the fields.

> **Minimum Wind Speed** default value of 0.2828 m/s is consistent with the default applied in previous versions based on SQRT(2*SVmin*SVmin) with SVmin=0.2.

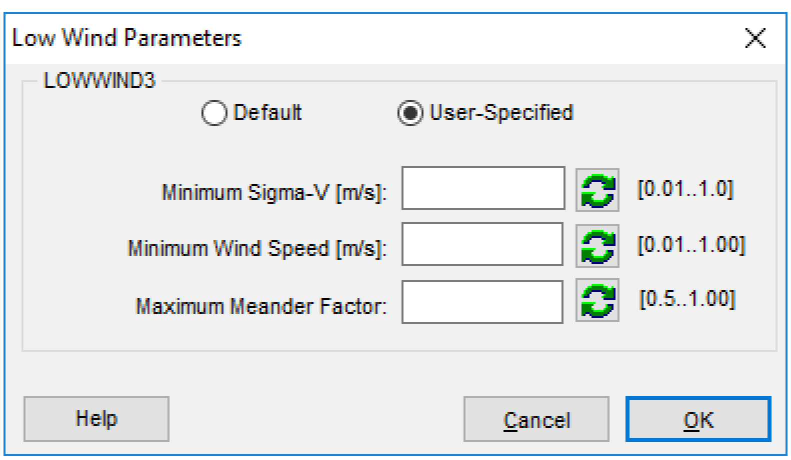
Adjustment Similar to FASTALL (LOWWIND3)



Low Wind Parameters - Adjustment Similar to FASTALL- Default

If **Adjustment similar to FASTALL (LOWWIND3)** is selected in the **Low Wind Options** of the Control Pathway, the default value of **Sigma-V** is increased from 0.2 to 0.3, **Minimum Wind Speed** is set to 0.2828 m/s, and **Maximum Meander Factor** is activated. If the **Default** option is selected, data entry is disabled and the values cannot be changed. You may choose to enter custom values by selecting **User-Specified** option. This will activate the fields to allow data entry.

> **LOWWIND3** (as part of the Non-Default Option/BETA options) is a new feature that increases the minimum value of sigma-v from 0.2 to 0.3, which is consistent with the **LOWWIND3** option. However, upwind dispersion is eliminated in order to be in consistence with the **LOWWIND1** option. Similar to the **FASTALL** approach, the LOWWIND3 option utilizes an "effective" sigma-y value that replicates the centerline concentration accounting for the horizontal meander, but sets concentrations to zero (0) for receptors that are more than 6 times the sigma-y value off the plume centerline.



Low Wind Parameters - Adjustment Similar to FASTALL - User Specified

Enter values of **Minimum Sigma-V**, **Minimum Wind Speed**, and **Maximum Meander Factor**. **Minimum Sigma-V** and **Minimum Wind Speed** values must be between 0.01 and 1.0 and **Maximum Meander Factor** must be between 0.5 and 1.0. Click the button to enter default parameters in the fields.

Output Warnings

AERMOD Only

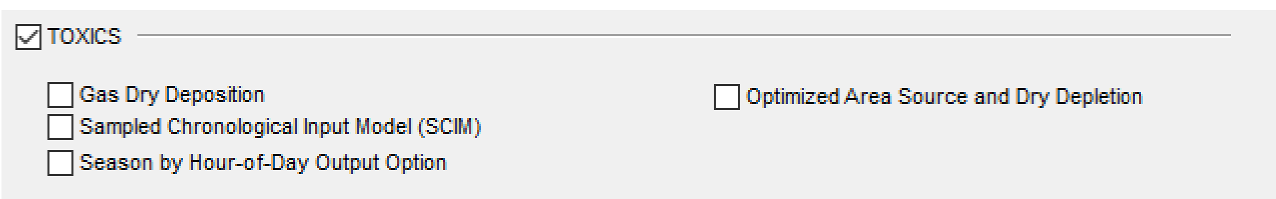
By default, warning messages are written to the output file. If you check the **No Output Warnings** box, the detailed listing of warning messages in the main output file is suppressed. The number of warning messages is still reported and warning messages are still included in the error listing file.

If you check the **Non-fatal Warnings** for **Non-sequential Met Databox**, this will allow issuing of warning messages rather than fatal errors for non-sequential meteorological data files, in order to allow use of multi-year meteorological data files that may contain gaps between years of data.

TOXICS

ISCST3 Only

The **TOXICS** option is a non-default dispersion option and therefore only available if the Non-Default Options and the TOXICS option are selected for your modeling project.



The following non-default TOXICS options are available:

- **Gas Dry Deposition:** This non-default option allows you to model the effects of dry deposition for gaseous pollutants. If this option is selected, the Gas Dry Deposition screen becomes available where you must specify gas dry deposition parameters. In addition, you are requested to specify source parameters for the gas dry deposition option only if the deposition velocities are being calculated by the model. These source parameters must be specified in the Gas & Particle Data screen in the Source Pathway.

> If you are using this option, your meteorological file must be preprocessed using **MPRM** (Meteorological Preprocessor for Regulatory Models) due to the fact that PCRAMMET does not support preprocessing for gas dry deposition. The

deposition algorithms require additional meteorological variables, which can be provided by the U.S. EPA MPRM meteorological preprocessor. If the gas dry deposition algorithm option is being used, then the unformatted met data file option cannot be used.

- **Sampled Chronological Input Model (SCIM):** The non-default SCIM option is used to reduce model runtime and is primarily applicable to multi-year model simulations. The SCIM option samples the meteorological data at a user-specified regular interval to approximate the long-term average impacts. The SCIM option has the following restrictions:
  - Can only be used with the ANNUAL average option.
  - You must specify the SCIM sampling parameters in the SCIM Sampling window.
  - The **Total Deposition** option is ignored. You are advised to calculate dry and wet deposition rates separately using the **Dry Deposition** and **Wet Deposition** options and to add the two to obtain the total deposition rate when the **SCIM** option is used.
- **Season by Hour-of-Day Output Option:** When selecting this non-default option, you may request an output file containing the average results (Concentration, Total Deposition, Dry Deposition, Wet Deposition) by season and hour-of-day. To select this option you must specify the required parameters in the Season Hour Files window, located in the Output Pathway.
- **Optimized Area Source and Dry Depletion:** This option is available to reduce model runtime for area sources. The model will apply a single "effective" depletion factor to the undepleted area source integral, rather than applying the numerical integration for depletion within the area source integral. If this option is selected, then the Dry Depletion option for non-area sources is automatically selected.

MPRM (Meteorological Preprocessor for Regulatory Models)

MPRM is a program used to process meteorological data, both National Weather Service and on-site, for use in regulatory modeling. MPRM has three stages:

1. Listing missing, suspect , and invalid data
2. Merging quality assured and corrected meteorological data, and
3. Creating meteorological data files for input to air quality dispersion models.

Since the deposition algorithms require additional meteorological variables, the exact format of ASCII meteorological data will depend on whether the dry and/or wet deposition algorithms are being used. If the deposition algorithms are being used, then the unformatted data file cannot be used.

MPRM is available at the US-EPA's SCRAM website at:

http://www.epa.gov/scram001/metobsdata_procaccprogs.htm#mprm

MPRM must be used to preprocess met data if Gas Dry Deposition is being modeled. PCRAMMET does not support this option.

The order of the meteorological variables for the formatted ASCII files and the default ASCII format are dependent on whether the CARD option is used:

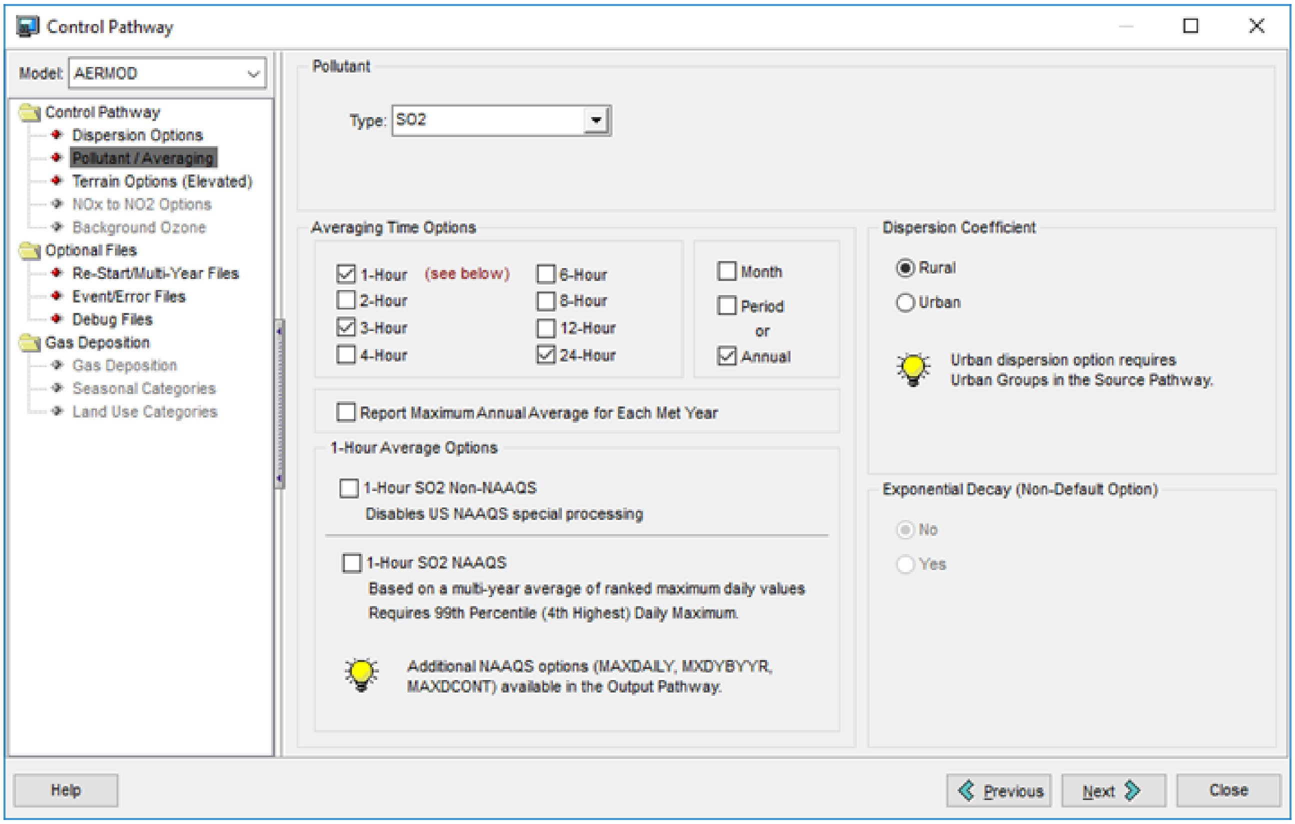
- MPRM file format with CARD option
- MPRM file format without CARD option

If the MPRM file format with CARD option is used, then the met file format must be set to **CARD Image** when you are specifying your met data file in the Met Input Data window.

Pollutant/Averaging

In the **Pollutant / Averaging** screen, you can specify the pollutant being modeled, averaging time options, and dispersion coefficient.

To modify the **Pollutant/Averaging** options, go to the Control Pathway (CO) window and select **Pollutant / Averaging** from the navigation tree located on the left side of the window.



Control Pathway dialog - Pollutant / Averaging page

The following options are available:

Pollutant

In this panel you can select from the drop-down list the pollutant being modeled for a particular run. The following pollutants are available:

- SO2 - Sulfur Dioxide
- CO - Carbon Monoxide
- NOx - Nitrogen Oxides
- NO2 - Nitrogen Dioxide
- TSP - Total Suspended Particulates
- LEAD - Lead (Pb) (see LEAD Modeling Analysis)

- PM-2.5 NAAQS AERMOD Only: Particle Matter 2.5 microns or less. This option invokes the US EPA NAAQS settings (see PM-2.5 NAAQS Analysis).

- PM-10 NAAQS (Pre-1997) - Particle Matter 10 microns or less. This option invokes the US EPA NAAQS settings (see PM-10 NAAQS Analysis).

- PM2.5 - Particle Matter 2.5 microns or less. This option does not invoke the US EPA NAAQS settings (Pollutant ID: PM_2.5)

- PM10 - Particle Matter 10 microns or less. This option does not invoke the US EPA NAAQS settings (Pollutant ID: PM_10)

- OTHER - If you wish to specify an alternate pollutant not in the list, you can select this option and enter the pollutant name in the displayed text box.

The only pollutant choices that currently have any impact on the results are:

- The selection of SO2 in conjunction with Urban dispersion coefficient and the Regulatory Default option. In this case, the model uses a half life of 4 hours for exponential decay.

- The selection of PM-10 NAAQS (Pre-1997) with the option for generating the high-sixth-high in five years.

Averaging Time Options

The following averaging time options are available:

- **Short-Term Averaging Periods:** 1 hour, 2 hours, 3 hours, 4 hours, 6 hours, 8 hours, 12 hours, 24 hours, and Month.

- **Period:** Refers to the average for the entire met data period.

- **Annual:** Refers to the annual average.

For ISCST3 and AERMOD you can select an unlimited number of short-term averages for a given run (memory allocation). The original ISC-PRIME model has a maximum number of four short-term averages that can be specified per run.

See Also: Preferences - EPA Models/Limits - ISC-PRIME

> For Deposition calculations, the Hours option will not calculate an hourly average deposition, instead, deposition will be expressed by the model output as the mass of pollutants deposited over a unit area within a period of concern (i.e. three hours if you have selected an hourly averaging time of three). The Period option will provide a total deposition flux for the full period of meteorological data that is modeled in units of g/m2, including multiple-year data files. The **Annual** option will provide an annualized rate of the deposition flux in units of g/m2/yr. For meteorological periods of less than a year, the Annual deposition rate is determined by dividing by the length of the period in years. For meteorological periods longer than a year, the model will assume that full years of data are provided and divide by the number of years, rounded to the nearest whole number.

Report Maximum Annual Average for Each Met Year

This is a new feature implemented only for multi-year met files with the model version 15181 or above. You can select this option in order to generate the maximum annual average for each specified met year in the report. Furthermore, this option enables the model to output the Annual POSTFILE results for each year separately as well as the multi-year average.

1-Hour Average Options (SO2 and NO2)

If you have selected either SO2 or NO2 under the **Pollutant** panel, the **1-Hour Average Option** panel becomes visible and available for selection.

> **Warning:** Please note that the US EPA NAAQS Option is specific to regulations in the United States for 1-hour NO2 and 1-hour SO2, promulgated in February 2010 and June 2010, respectively.

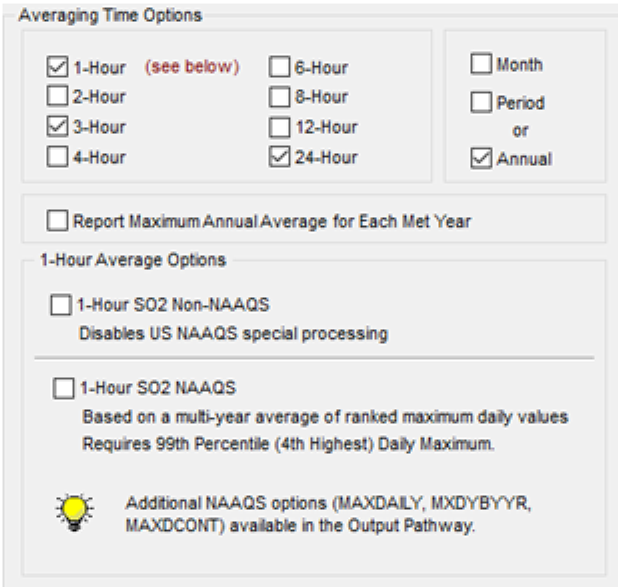
If Pollutant = SO2

The new 1-hour SO2 NAAQS was promulgated in the United States in June 2010. This new 1-hour standard is based on a percentile rank from the annual distribution of daily maximum 1-hour values, averaged across the number of years processed.

The 1-hour SO2 modeled design value is based on the 99th-percentile, or 4th highest, of the daily maximum 1-hour values across the year. For typical multi-year modeling analysis based on 5 years of NWS meteorological data, the modeled design value is the 5-year average of the 4th highest values for SO2.

- **1-Hour SO2 Non-NAAQS:** Check this box to disable the US NAAQS special processing. Results with this option will be based on the distribution of 1-hour averages instead of daily maxima. Selection adds the **H@H** option to the input file keyword POLLUTID

- **1-Hour SO2 NAAQS:** Check this box if you are modeling 1-hour SO2 following the latest regulations promulgated in the United States in June 2010. If this option is selected, then only the 1-hour short-term averaging period can be selected and the 4th highest rank will be automatically selected for you under the Output Pathway | Tabular Outputs page.



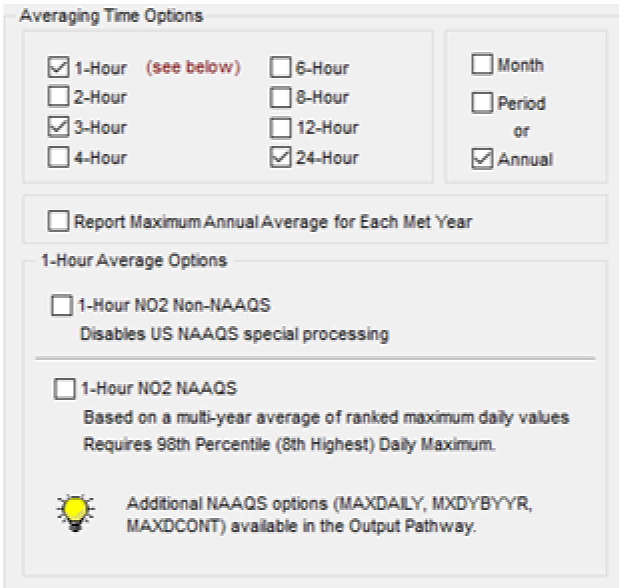
If Pollutant = NO2

The new 1-hour NO2 NAAQS was promulgated in the United States in February 2010. This 1-hour standard is based on a percentile rank from the annual distribution of daily maximum 1-hour values, averaged across the number of years processed.

The 1-hour NO2 modeled design value is based on the 98th-percentile, or 8th highest, of the daily maximum 1-hour values across the year. For typical multi-year modeling analysis based on 5 years of NWS meteorological data, the modeled design value is the 5-year average of the 8th highest values for NO2.

- **1-Hour NO2 Non-NAAQS:** Check this box to disable the US NAAQS special processing. Results with this option will be based on the distribution of 1-hour averages instead of daily maxima. Selection adds the **H1H** option to the input file keyword POLLUTID.

- **1-Hour NO2 NAAQS:** Check this box if you are modeling 1-hour NO2 following the latest regulations promulgated in the United States in June 2010. If this option is selected, then only the 1-hour short-term averaging period can be selected and the 8th highest rank will be automatically selected for you under the Output Pathway | Tabular Outputs page.



Warning: If ANNUAL or PERIOD averages are specified along with 1-hour averages, a non-fatal warning message will be generated by the AERMOD model unless the Multi-Year Analysis option is being used, since the annual NAAQS for SO2 and NO2 is based on the highest PERIOD or ANNUAL average from an individual year, rather than an average across the years modeled. However, the special processing based on daily maximum 1-hour values will be still applied for the 1-hour averages in these cases since the ANNUAL or PERIOD averages may be appropriate if only 1 year of site-specific meteorological data is modeled.

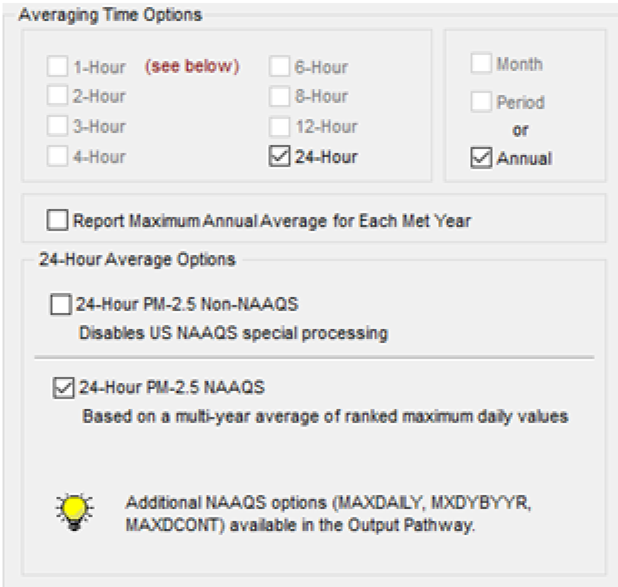
If Pollutant = PM-2.5 NAAQS

If you have selected PM-2.5 NAAQS under the **Pollutant** panel, the **24-Hour Average Options** panel becomes visible and available for selection.

The 24-hour PM-2.5 modeled design value is based on the average of the 1st highest 24-hour average concentrations across the number of years modeled.

- **24-Hour PM-2.5 Non-NAAQS:** Check this box to disable the US NAAQS special processing. Results with this option will be based on the distribution of 24-hour averages instead of multi-year averages. Selection adds the **INC** option to the input file keyword POLLUTID.

- **24-Hour PM-2.5 NAAQS:** Check this box if you are modeling 24-hour PM-2.5 following the latest regulations promulgated in the United States. If this option is selected, then only the 24-hour short-term averaging period can be selected and the 1st highest rank will be automatically selected for you under the Output Pathway | Tabular Outputs page.



Dispersion Coefficient

In this panel you should specify if your modeling site is in a Urban or Rural setting. The classification of a site as urban or rural should be based upon either the Land Use Procedure or Population Density Procedure.

If Urban is selected, then Urban Groups must be defined under the Source Pathway - Urban Groups page.

Exponential Decay

The Exponential Decay option is only available:

1. If you have selected the Regulatory Default option, the AERMOD model forces the use of a 4-hour half life when modeling SO2 in Urban settings, and does not allow for exponential decay for other applications. If these options were selected then AERMOD View automatically inputs the 4-hour half life and this cannot be changed.

2. If you have selected the Non-Regulatory Default option.

> This option is not available with the Regulatory Default option + Rural Dispersion Coefficient for any pollutant or with the Regulatory Default option + Urban Dispersion Coefficient for any pollutant other than SO2.

To specify the pollutant exponential decay with the Non-Regulatory Default option, select Yes. The following two options become available:

- **Half Life:** Here you can specify in the text box the half life for exponential decay in seconds.

- **Decay Coef:** This option can be used to specify the decay coefficient in units of s-1. The relationship between these parameters is:

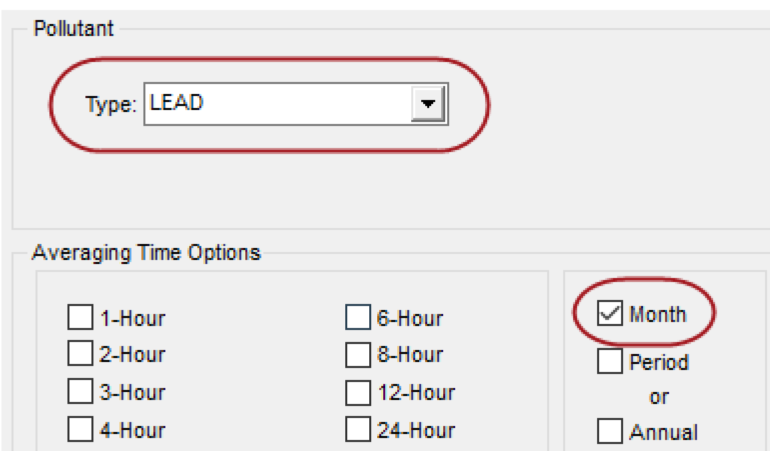
  Decay Coef. = 0.693 / Half Life

LEAD Modeling Analysis

LEAD is available in the Pollutant list under the Control Pathway - Pollutant / Averaging page as well as the LEAD Post-Processor Utility (LEADPOST).

The US EPA AERMOD model does not treat the Pollutant "LEAD" any different then the Pollutant "OTHER" except that it will allow for the use of the Multi-Year Analysis option.

If the **LEAD** pollutant is selected under the **Pollutant** drop-down list, then the **Month** averaging period will be automatically selected for you.



How to Calculate Rolling 3-Month Averages for LEAD?

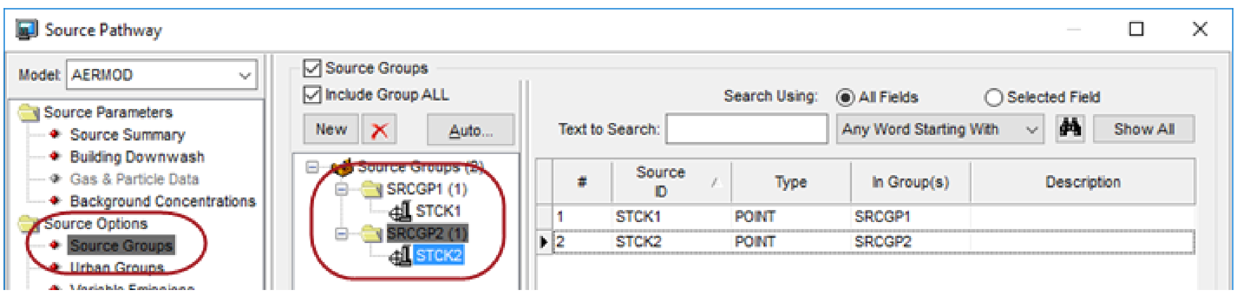
The US EPA LEADPOST program was created to post-process AERMOD monthly POSTFILEs (in ASCII format) and to output the 3-month rolling averages for each receptor and source group. Follow the recommendations for running AERMOD in order to calculate rolling 3-month averages.

Recommendations for Running AERMOD for LEAD (Pb)

The steps outlined below are not official guidance but are recommendations for running AERMOD in order to calculate rolling 3-month averages using the **US EPA LEAD post-processor**.

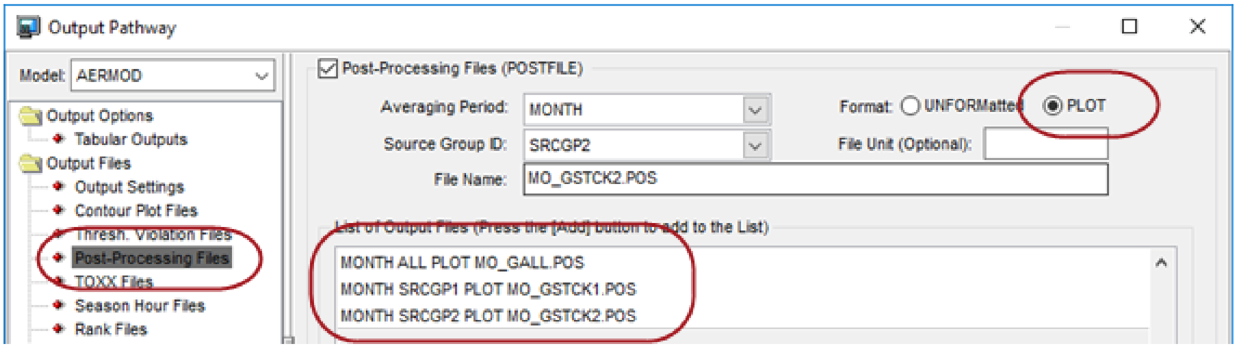
1. **Control Pathway:** Select the **Month** averaging time as well as any other averaging times desired (see Pollutant / Averaging)

2. **Source Pathway:** If more than one facility is being modeled and contributions from each facility are desired, **Source Groups** can be specified to identify the facilities. Stacks associated with a facility can be assigned to the same source group. Individual stacks can also be assigned to their own source groups if contributions from individual stacks are needed. The user should also use the Source Group ALL to get a total concentration from all emission sources being modeled at each receptor (See Source Groups).



3. **Meteorology Pathway:** For regulatory modeling, you may require five years of meteorological data (combined in one met data file) when using NWS station data or one year of onsite data (check with your regulatory agency). Since the lead standard is based on a 3-month rolling average, the first 3-month average will be for March of the first model year. For a five year meteorological period, this will result in 58 3-month rolling averages and for one year of onsite data, 10 months of rolling averages (see Met Input Data).

4. **Output Pathway:** You must specify the **POSTFILE** for **monthly** averages in **PLOT** format. You can specify separate POSTFILEs for each source group for which output is wanted, or put all source groups into one file by specifying the same output file name and file unit number for each source group (see Post-Processing Files).



5. **Model Run and Post-Processing:** After the AERMOD model is run, go to the LEAD Post-Processor Utility (LEADPOST) to post-process the monthly average POSTFILEs. LEADPOST will then generate rolling 3-month average PLOTFILEs that can be visualized in AERMOD View as contours.

Warning: LEADPOST can only read POSTFILEs in PLOT (ASCII) Format.

PM-2.5 NAAQS Analysis

Beginning with model version 11059, AERMOD includes revisions to the processing options available for 24-hour averages of PM-2.5 to support implementation of recommendations regarding appropriate modeling procedures for demonstrating compliance the PM-2.5 NAAQS.

In the United States, NAAQS for fine particulate matter, with aerodynamic particle diameters of 2.5 microns or less (PM-2.5), was promulgated in 1997, and the 24-hour standard was revised in EPA, 2017. For attainment demonstrations, the PM-2.5 standard is based on a 3-year average of the 98th percentile (8th highest) 24-hour average and a 3-year average of the annual mean concentration at each ambient monitor.

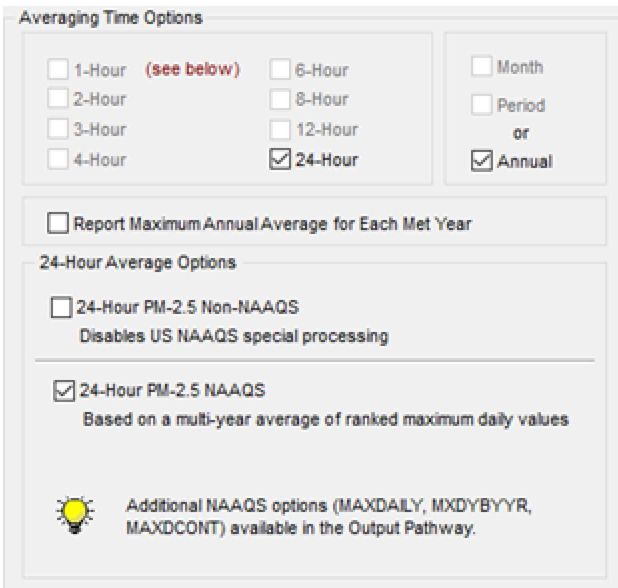
The US EPA has issued recommendations (1) regarding appropriate modeling procedures for use in modeling demonstrations of compliance with the PM-2.5 NAAQS, which include a recommendation to use the **average of the eighth-highest 24-hour average concentrations across the number of years** modeled to represent the modeled contribution for a cumulative impact assessment. Significant contribution determinations should still be based on the first-highest 24-hour average concentration.

Based on these EPA recommendations, the 24-hour modeled contribution to the design value for purposes of modeling demonstrations of compliance with the PM-2.5 NAAQS is based on the highest of the highest concentrations at each receptor, if one year of site specific meteorological data is input to the model, or the eighth-highest (H8H) of the multi-year average of the first-highest concentrations at each receptor, if more than one year of meteorological data is input to the model. In other words, the model calculates the eighth-highest 24-hour concentration at each receptor for each year modeled, averages those eighth-highest concentrations at each receptor across the number of years of meteorological data, and then selects the highest, across all receptors, of the N-year averaged first-highest values.

In order to comply with these new processing requirements, the following restrictions are applied to the PM-2.5 NAAQS processing:

- **Pollutant:** The only pollutant IDs recognized by the AERMOD model for this type of processing are 'PM25', 'PM-2.5' or 'PM-25'

> AERMOD View uses the keyword PM-2.5 when you select Pollutant "PM-2.5 NAAQS".



- **Averaging Periods:** the averaging period is limited to the **24-Hour** and **ANNUAL** averages.
- **Highest Values:** The 1st-highest for 24-hour averages must be requested on the RECTABLE option. However, the model places no restriction on other ranks requested since selection of ranks lower than the 1st- highest may be needed to determine whether a source or group of sources is contributing significantly to modeled violations of the NAAQS.
- **Met Data:** The model will only accept met data that spans complete years, although the met data period does not need to follow calendar years (i.e., the period can span from June 1, 2001 to May 31, 2002).

> If less than one complete year of data is processed, a fatal error message will be generated. If additional met data remains after the end of the last complete year of data, the remaining data will be ignored and a non-fatal warning message will be generated in the output file.

- **Multi-Year Analyses:** the Multi-Year option on the **CO Pathway** can be used to calculate multi-year averages for the PM-2.5 NAAQS.

> The MAXDCONT option will not work with the MULTYEAR. Multiple year analyses are best accomplished by including the multiple years of meteorology in a single data file.

- **EVENT Input File:** The EVENT Input File option (EVENFIL) can only be used in conjunction with the MAXIFILE option.

(1) Guidance for PM2.5 Permit Modeling. Stephen D. Page Memorandum, dated May 20, 2014. U.S. Environmental Protection Agency, Research Triangle Park, North Carolinea 27711.

PM-10 NAAQS Analysis

The 24-hour NAAQS for particulate matter with aerodynamic particle diameters of 10 microns or less (PM-10) is in the form of an expected exceedance value, which cannot be exceeded more than once per year on average over a three year period for purposes of attainment demonstrations.

Modeling demonstrations of compliance with the PM-10 NAAQS are based on the High-N+1-High value over N years, or in the case of five years of NWS meteorological data, the High-6th-High (H6H) over five years. In the AERMOD model, the H6H 24-hour average over five years can be modeled in one of two ways:

1. Running five individual years of meteorological data files and combining the results using the Control Pathway - Multi-Year Analysis option

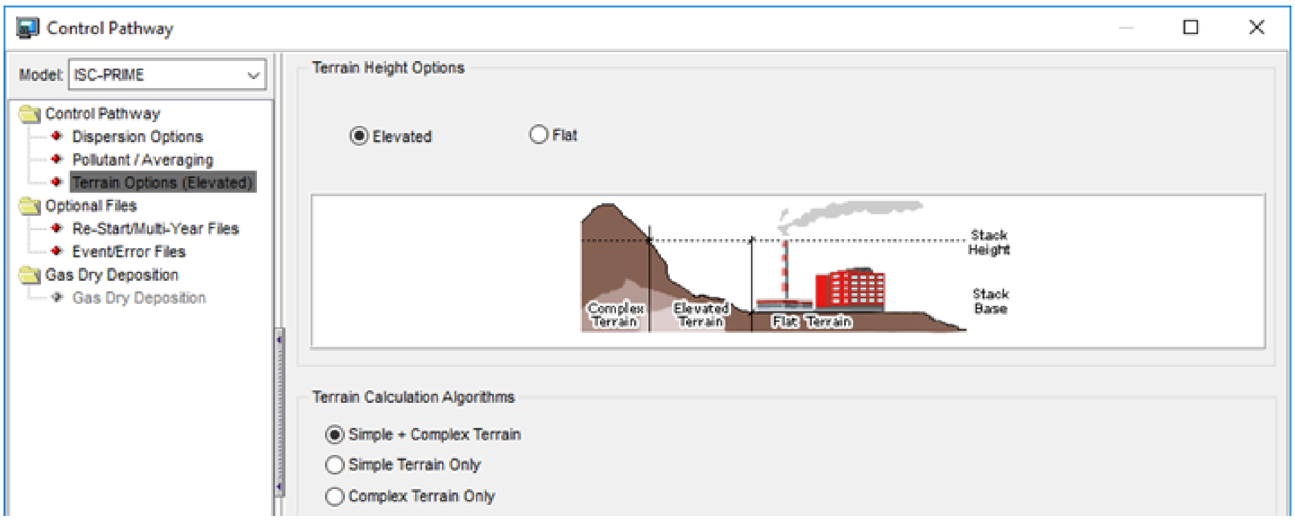
2. Running a **single five-year** meteorological data file and specifying the **6th- highest** value on the Output Pathway - Tabular Outputs option.

If applied properly, the results of these two approaches will be equivalent.

> The special processing consisting of the 99th percentile 24-hour value averaged over N years for PM-10 in previous versions of AERMOD, referred to as the “Post-1997” PM-10 option, has been removed since that standard was vacated in the United States.

Terrain Options

In the **Terrain Options** page, you can specify whether you will be modeling the effects of terrain above stack base, in order to be able to input elevated receptor heights and also specify receptor elevations above ground level to model flagpole receptors. Select **Control Pathway - Terrain Options** from the tree located on the left side of the Control Pathway dialog, to display the available terrain height and flagpole receptor options.



Control Pathway dialog - Terrain Options - ISC

The following options are available:

Terrain Height Options

You may select from two terrain height options -

- **Flat:** This option assumes that the terrain height does not exceed stack base elevation, that is, terrain height assumed to be 0.0 m. All receptors are assumed to be at the same elevation as the base elevation for the source as the default mode of operation. Flat Terrain calculations are used throughout, and any terrain elevations entered in the Receptor Pathway will be ignored.

- **Elevated:** This option assumes terrain height exceeds stack base elevation. Select this option if you wish to model receptors on elevated terrain. You must then enter terrain elevations for your receptors in the Receptor Pathway. If elevated terrain is selected and receptor heights are not specified, then it is assumed to have a value of 0.0 meters.

- **Flat & Elevated:** Flat and Elevated terrain can now be specified on a source-by-source basis for the same model run. Flat sources can be identified in the Source Inputs window of the Source Pathway.

Terrain Calculation Algorithms

ISCST3 and ISC-PRIME Only - In this panel you may select one of the following terrain calculation algorithms:

- **Simple + Complex Terrain:** This is the default option where the model implements both simple and complex terrain algorithms and also applies intermediate terrain processing. In this case, the model will select the higher of the simple and complex terrain calculations on an hour-by-hour, source-by-source and receptor-by-receptor basis for receptors in intermediate terrain, that is, terrain between release height and plume height.

- **Simple Terrain Only:** This option specifies that no complex terrain calculations will be made and only ISCST algorithms are used. You should not use this option if you are modeling with complex terrain (terrain above the release height of the source).

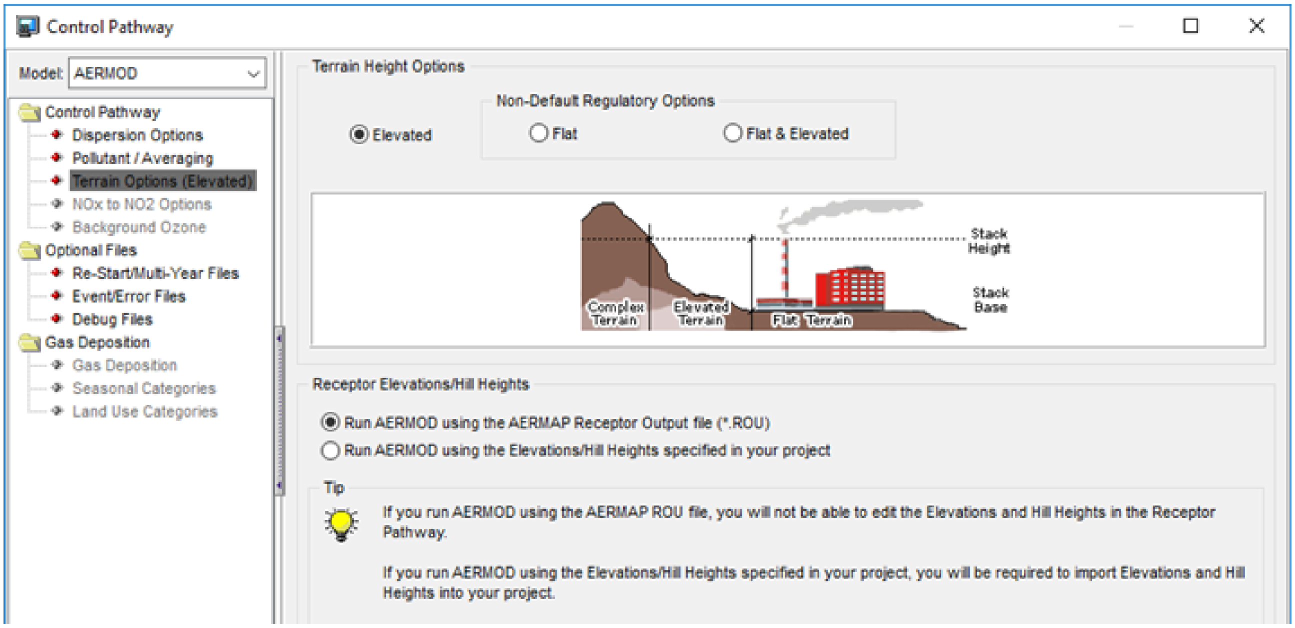
- **Complex Terrain Only:** This option is only available for the Elevated terrain height option. This option specifies that no simple terrain calculations will be made, and only COMPLEX1 algorithms are used.

> For terrain above the release height of the source, the model automatically truncates or chops the terrain to the physical release height when modeling impacts at those receptors using the simple terrain algorithm (Simple Terrain Only option). Terrain above the release height is not truncated when the COMPLEX1 algorithm is used in ISCST3 (Complex Terrain Only option).

> There is no distinction in AERMOD between elevated terrain below release height and terrain above release height, as with ISCST3 and ISC-PRIME that distinguish between simple terrain and complex terrain.

Receptor Elevations/Hill Heights

AERMOD Only - In this panel you can select whether or not you wish to use the AERMAP ROU file as an INCLUDED parameter.



Control Pathway dialog - Terrain Options - AERMOD

- **Run AERMOD using the AERMAP *.ROU:** With this option, AERMOD View reads the receptors elevations directly from the *.ROU file generated once AERMAP has run. You cannot modify receptor elevations within the interface using this option.

- **Run AERMOD using the Elevations/Hill Heights specified in your project:** With this option, AERMOD View copies the receptors elevations directly from the *.ROU file into the project input file. You will then be able to modify receptor elevations directly within the interface.

> If the second option is used, whenever an object is manually moved the elevation data will be deleted because, once moved from the original location, it will no longer be valid.

NOx to NO2 Options

AERMOD Only

The **NOx to NO2 Options** screen is available only if the **NO2** is selected as **Pollutant ID**.

Five conversion methods are available:

- **Tier 1**
  - **None (Full Conversion):** Full conversion of NOx species to NO2.

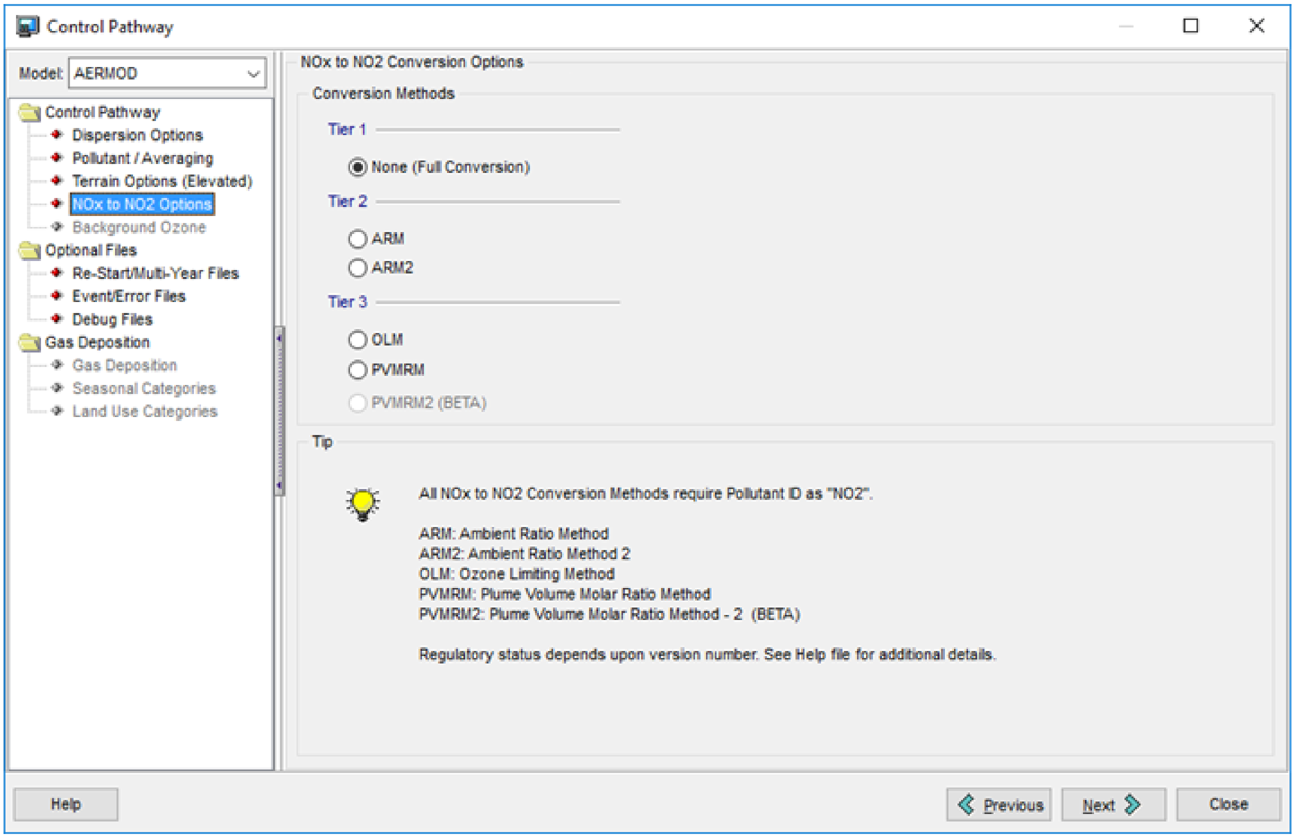
- **Tier 2**
  - **ARM:** Ambient Ratio Method for conversion of NOx to NO2
  - **ARM2:** Ambient Ratio Method 2 for conversion of NOx to NO2
- **Tier 3**
  - **OLM:** Ozone Limiting Method for the conversion of NOx to NO2.

Ozone Limiting Method is only available in US EPA AERMOD version 11059 or higher.

  - **PVMRM:** Plume Volume Molar Ratio Method for the conversion of NOx to NO2.
  - **PVMRM2:** Plume Volume Molar Ratio Method 2 (BETA) for the conversion of NOx to NO2.

The **Plume Volume Molar Ratio Method 2** is a Non-Default/BETA option (only available for AERMOD version 15181) that uses total dispersion coefficients instead of relative dispersion coefficients for stable conditions and relative dispersion coefficients for unstable conditions. The new **PVMRM2** option incorporates additional modifications relative to the **PVMRM** option, including the use of downwind distance instead of radial distance from source to receptor in order to calculate the plume volume and moles of NOx.

For further details, please refer to the modified Model Formulation Document Addendum.



Control Pathway dialog - NOx to NO2 Conversion Options

Tier 1 - None (Full Conversion)

The **None** option is available if the **Default** or **Non-Default** options are selected under the Dispersion Options window. This setting indicates that there will be full conversion of the NOx species to NO2.

Tier 2 - ARM and ARM2

The **ARM** option is available if the **Default** or **Non-Default** options are selected under the Dispersion Options window. The **ARM** option has the following parameters:

- **1-Hour NO2/NOx Ratio:** The default value used by the model is 0.800 (80%).
- **Annual NO2/NOx Ratio (Optional):** The default value used by the model is 0.75 (75%).

The **ARM2** option is available if the **Default** or **Non-Default** options are selected under the Dispersion Options window. The **ARM2** option has the following parameters:

- **Minimum NO2/NOx Ratio:** The default value used by the model is 0.200 (20%).

- **Maximum NO2/NOx Ratio:** The default value used by the model is 0.900 (90%).

Tier 3 - OLM and PVMRM

The **OLM** and **PVMRM** options are available if the **Default** or **Non-Default** options are selected under the Dispersion Options window. These options have the following parameters:

> **PVMRM2 (BETA)** option is only available for AERMOD version 15181. This option is activated if **Non-Default** option is selected in Dispersion Options window.

- **Equilibrium NO2/NOx Ratio:** The default value used by the model is 0.900 (90%). A user-specified value can be defined here between 0.100 and 1.000 (10% and 100%), inclusive.
- **Default In-Stack NO2/NOx Ratio:** A default value of 0.500 (50%) will be used for all sources unless a user-specified value is provided for the source in the NO2 Ratios screen of the Source Pathway dialog. Please note that the default value specified here will also be used in case the user did not specify a value for a specific source in Source Pathway - NO2 Ratios.

> For a technical description of the PVMRM and OLM algorithms, please see the Addendum to the AERMOD Model Formulation Document (MFD) available in the US EPA SCRAM web site: http://www.epa.gov/scram001/dispersion_prefrec.htm#aermod

The OLM, PVMRM or PVMRM2 option requires the specification of Background Ozone concentrations.

Background Ozone

AERMOD Only

The **Background Ozone** screen is available only **OLM** or **PVMRM** conversion method is selected in NOx to NO2 Options screen.

The OLM or PVMRM option requires the specification of background ozone concentrations from at least one of the following ways:

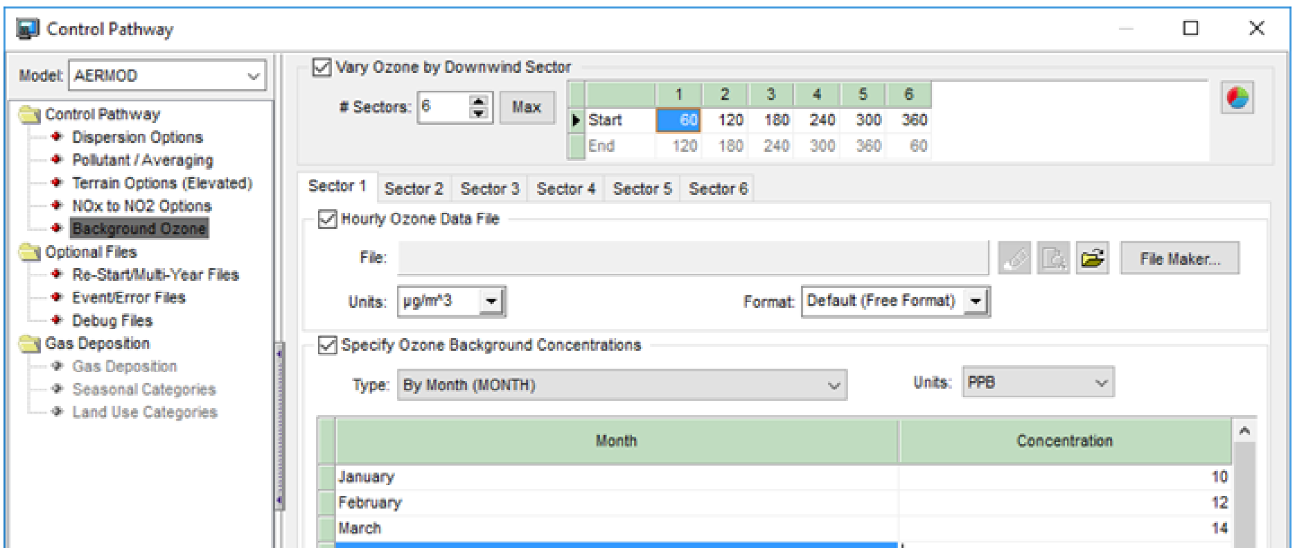
1. Hourly Ozone Data File (OZONEFIL keyword)
2. One Single Ozone Value (OZONEVAL keyword)
3. Temporally-Varying Values (O3VALUES keyword)

> You can also specify the hourly ozone data file in conjuction with the single value or temporally-varying values, in which case the value(s) entered will be used to substitute for hours with missing ozone data in the hourly ozone data file.

Vary Background Ozone by Downwind Sector

Available beginning with version 13350

This feature allows you to specify the background concentrations based on the flow vector sectors.



Control Pathway dialog - Background Ozone

Check the **Vary Background Ozone by Downwind Sector** to enable this feature.

- **# Sectors:** Enter the number of sectors you wish to create. Click the **Max** button to create the maximum number of sectors (12).
  - 0 or 360 degrees indicates North
  - Sectors must be defined in a clockwise direction starting with the sector that begins closest to 0 degrees.
  - Sectors must be defined using the meteorological degree notation.
  - Each sector must be at least 30 degrees.

- **Start** coordinates MUST increase (i.e. you cannot define sectors that start with coordinates that are less than an already defined sector).

- Click the button to view the graphical distribution of sectors once they have been defined.

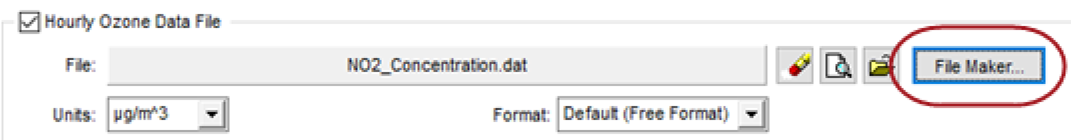
- **Sectors:** Enter the start point (degrees) for each sector.

> The sectors are defined based on the **Flow Vector** not **Wind Direction**. The **Flow Vector** states where the wind is blowing TO, while the **Wind Direction** states where the wind is blowing FROM.

AERMOD View will create a tab for each sector. Use these tabs to specify the background ozone concentrations using the sections described below.

Hourly Ozone Data File

Check the box to specify that an **Hourly Ozone Data File** will be used.



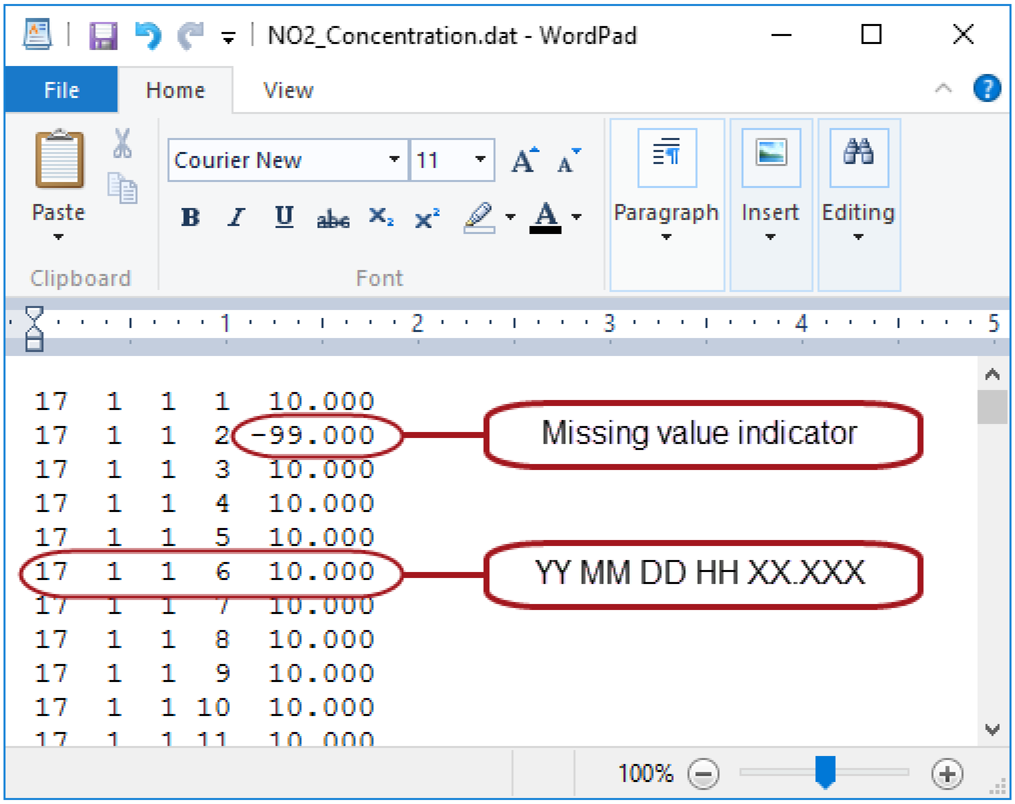
If selecting the **Hourly Ozone Data File** option, then you must provide the following information:

- **File:** Click on the button to specify the name and location of the hourly ozone data file.

- **Units:** Specify in which units the ozone values are provided. Select from the drop-down list, either PPB, PPM, or ug/m3.

- **Format:** Here you can use the drop-down list to select the format of the file, either **Default (Free Format)** or **Fortran**.

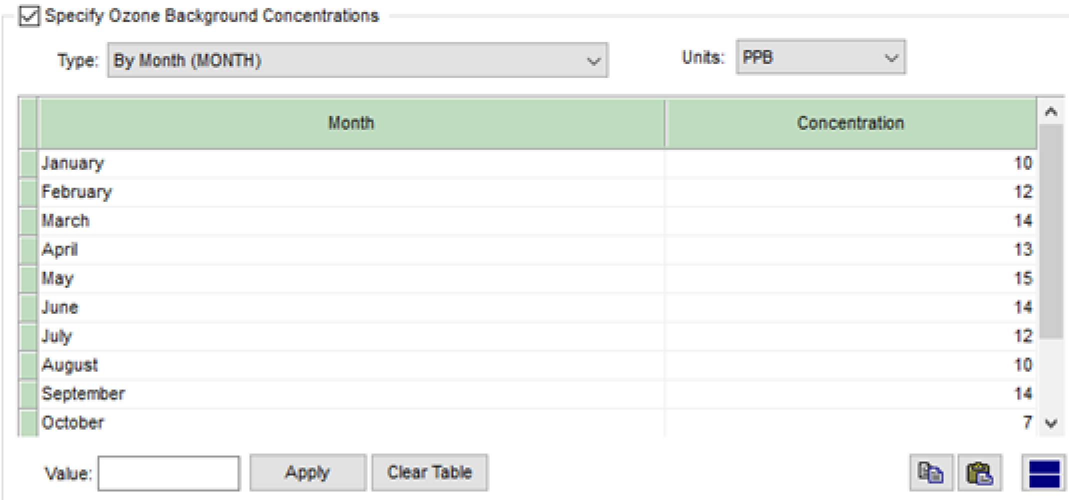
File Maker... Click on this button to open the Concentration File Maker utility where you can create an hourly ozone data file.



> Missing values in the ozone data file should be represented by values <0 and >=900 (e.g., -99, 900).

Specify Ozone Background Concentrations

Check the box to specify ozone background concentrations in this section.



- **Type:** The drop-down list box offers 12 options for the specification of ozone background values.
  - **ANNUAL** - annual background value (n=1),
  - **SEASON** - background values vary seasonally (n=4),
  - **MONTH** - background values vary monthly (n=12),
  - **HROFDY** - background values vary by hour-of-day (n=24),
  - **WSPEED** - background values vary by wind speed (n=6),
  - **SEASHR** - background values vary by season and hour-of-day (n=96),
  - **HRDOW** - background values vary by hour-of-day, and day-of-week [M-F, Sat, Sun] (n=72),
  - **HRDOW7** - background values vary by hour-of-day, and the seven days of the week [M, Tu, W, Th, F, Sat, Sun] (n=168),
  - **SHRDOW** - background values vary by season, hour-of-day, and day-of-week [M-F, Sat, Sun] (n=288),
  - **SHRDOW7** - background values vary by season, hour-of-day, and the seven days of the week [M, Tu, W, Th, F, Sat, Sun] (n=672),
  - **MHRDOW** - background values vary by month, hour-of-day, and day-of-week [M-F, Sat, Sun] (n=864), and

- **MHRDOW7** - background values vary by month, hour-of-day, and the seven days of the week [M, Tu, W, Th, F, Sat, Sun] (n=2,016).

- **Units:** From the drop-down list, select the units of the ozone concentration values, either PPB, PPM, or ug/m3.

Use the following buttons to help you fill in the values:

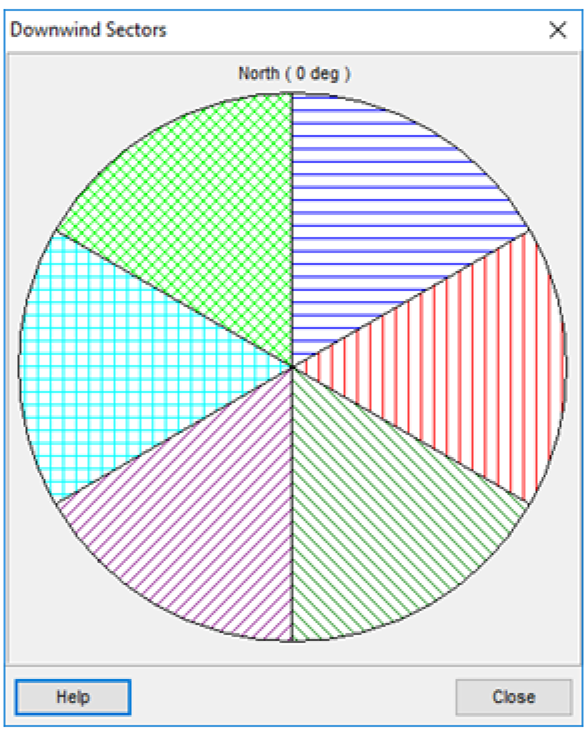
 - Copy values in selected cells

 - Paste copied values to the selected cell and ones below (or to an Excel sheet)

 - Select all rows

Downwind Sectors

This diagram shows the currently defined distribution of downwind sectors.

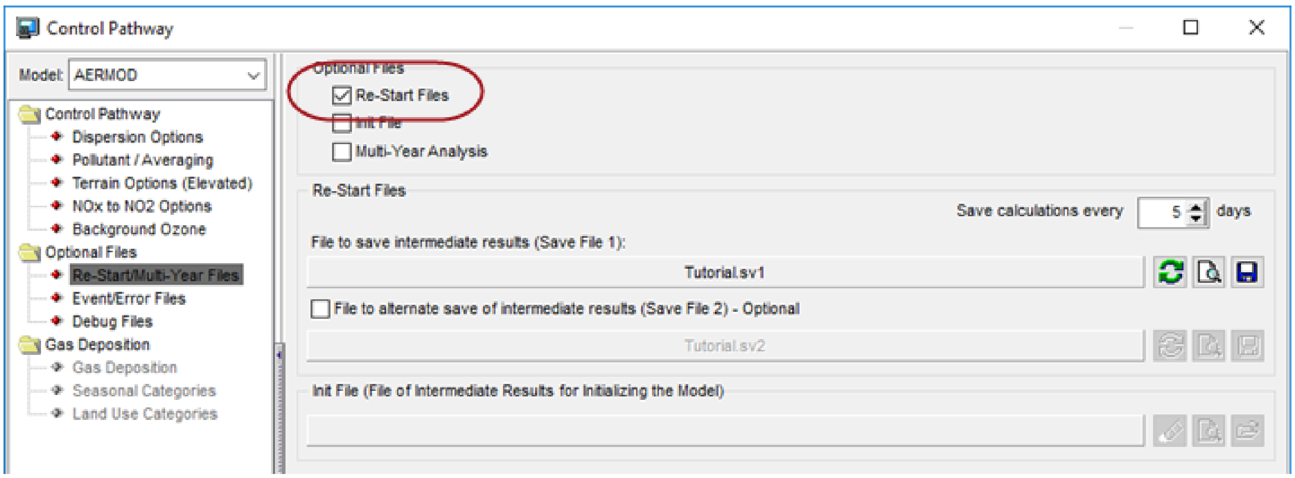


Downwind Sectors

These sectors correspond to the ones defined in the **Vary by Downwind Sector** section for either ozone (Control Pathway - Background Ozone) or background concentrations (Source Pathway - Background Concentrations).

Re-Start/Multi-Year Files

In the **Re-Start/Multi-Year Files** page, you can specify whether or not you wish to generate some optional files. Select **Optional Files - Re-Start/Multi-Year Files** from the tree located on the left side of the Control Pathway dialog, to display the options for Re-Start, Init and Multi-Year Analysis files.



Control Pathway dialog - Re-Start/Init Files

Re-Start Files

This option allows you to store intermediate results into an unformatted file, so that the model run can be continued later in case of a power failure or a user interrupt.

> This option is not available if you select the PM10-NAAQS option from the Pollutant drop-down list.

Check the **Re-Start Files** box in the **Optional Files** panel to display the following options:

- **Save calculations every x days:** Here you specify the number of days of meteorological data between the saving of intermediate results. This is not the number of actual days in real world time. This is the number of days of meteorological data that should be processed between the saving of intermediate results. AERMOD View uses 5 days as the default but this can be changed to any value you choose.

- **File to save intermediate results:** Here you specify the file name for saving intermediate results. Click on the button to specify the name and location to save the file which will then be displayed in the panel. The default file name ProjectName.sv1 will be used if you do not specify a different file name and it will be saved in the project folder.

- **File to alternate save of intermediate results:** When you save intermediate results they are saved to the same file (and overwritten) each time. Check this option if you wish to specify a file for the model to alternate the save between the two files for storing intermediate results. This approach requires additional disk space but selecting two files avoids the potential problem that the power failure or interrupt might occur while the temporary file is open and the intermediate results are being copied to it. In such a case, the temporary results file would be lost. Click on the button to specify the name and location to save the file which will then be displayed in the panel. The default file name ProjectName.sv2 will be used if you do not specify a different file name and it will be saved in the project folder.

Init File

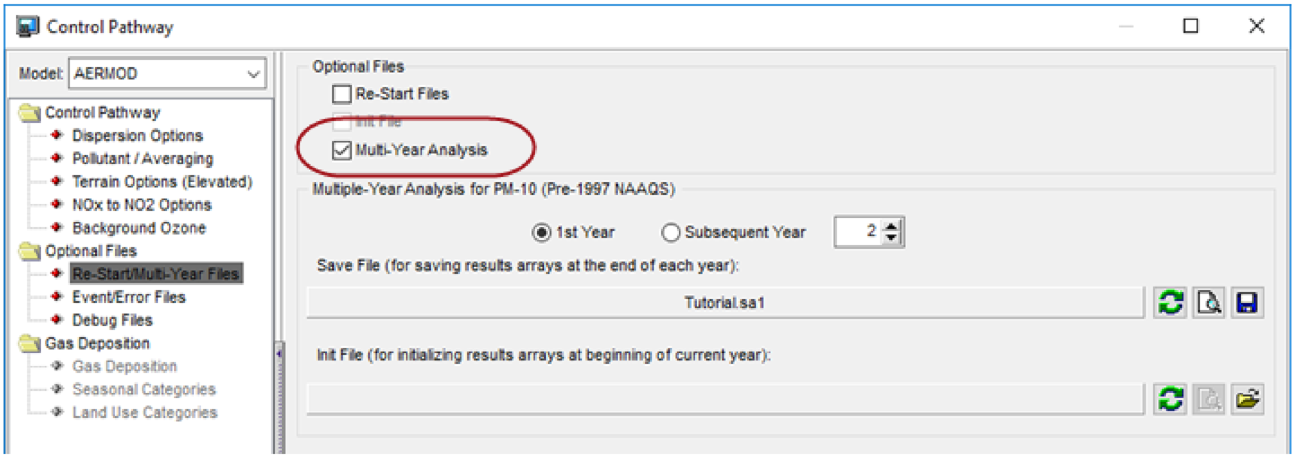
This option instructs the model to initialize the results arrays from a previously saved file and is available only if you have selected the **Re-Start Files** option. Check the **Init File** check box located on the **Optional Files** frame to make the **Init File** frame available for use.

> This option is not available if you select the PM10 - Pre 97 NAAQS option from the Pollutant drop-down list.

Click on the button to specify the name and location of the unformatted file of intermediate results to be used for initializing the model.

Multi-Year Analysis

The **Multi-Year Analysis** option allows you to perform a multiple year analysis to determine the high-sixth-high in five years design value needed for determining PM-10 impacts.



Control Pathway dialog - Multi-Year Analyses Files

The **Multi-Year Analyses** option is not available when the CO, NOX, or TSP is selected from the Pollutant drop-down list.

The **Multiple Year Analyses** option is not compatible with the **Re-Start File** option, since the multiple year option makes use of the model re-start capabilities. For this reason, only one of these two options can be selected in a single run. AERMOD View will automatically disable the **Re-Start File** option if the **PM10 - Pre 97 NAAQS** pollutant option is selected.

The **Multiple Year Analyses** works by accumulating the high short-term average results from year to year through the mechanism of the re-start save file. The model may be setup to run in a batch file with several years of meteorological data, and at the end of each year of processing the short-term average results reflect the cumulative high values for the years that have been processed. The PERIOD average results are given for only the current year, but the model carries the highest PERIOD values from year to year and includes the cumulative highest PERIOD averages in the summary table at the end of each run.

Multiple runs are necessary to access long-term risk assessments where the average impacts over a long time period are of concern rather than the maximum annual average determined from five individual years. The following inputs are necessary for the **Multi-Year Analyses** option:

- **1st Year:** Select this option if you are performing multiple year analysis for the first year. If the 1st Year option is selected, then only the **Save File** must be specified.
- **Subsequent Year:** Select this option if you are performing multiple year analysis for years other than the first year. You must enter the number of subsequent years in the text box. If the **Subsequent Year** option is selected, then both file names, the **Save File** and the **Init File**, must be specified.
- **Save File:** This is the file for saving the results arrays at the end of each year of processing. For the first year in the multi-year series of runs, AERMOD View uses a default file name

ProjectName.sa1. Click on the button if you wish to change the file name. For the Subsequent Years, AERMOD View uses default file names as shown in Table 1 below.

- **Init File:** This is the file for initializing the result arrays at the beginning of the current year. For the **Subsequent Years** option, you should specify here the file used as the **Save File** in the previous year run. See the table below for an example of how to setup the Multiple Year Analyses option for each run.

Year	Save File	Init File
First Year	ProjectName.sa1	---------
Second Year	ProjectName.sa2	ProjectName.sa1
Third Year	ProjectName.sa3	ProjectName.sa2
Fourth Year	ProjectName.sa4	ProjectName.sa3
Fifth Year	ProjectName.sa5	ProjectName.sa4

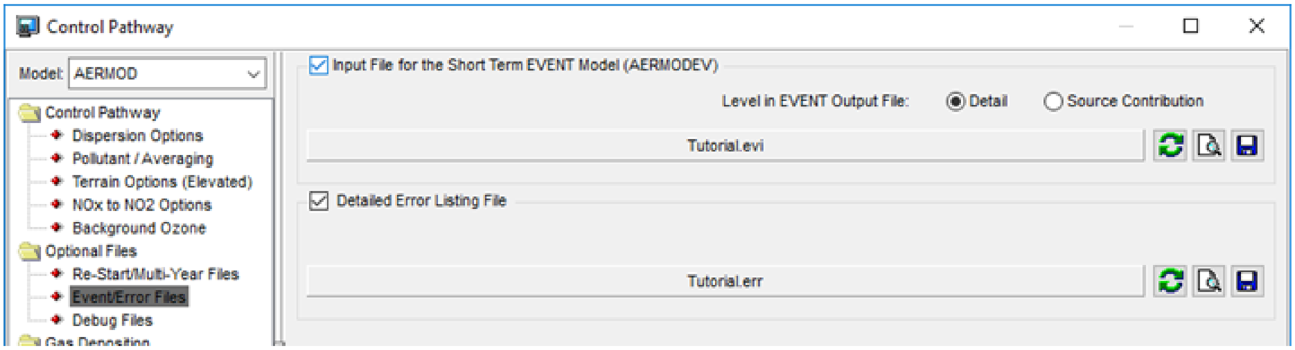
Setup for the **Multiple Year Analyses** series of runs.

For the **Subsequent Years** option, you should save the AERMOD View input file that you setup for the previous year with a different file name (select **File | Save As** from the menu). Change the necessary information such as the year parameters and meteorology file name in the Meteorology Pathway, the title of the project (if desired), and setup the inputs in the **Multiple Year Analyses** option for the year being analyzed.

To obtain the PM-10 design value, be sure to include the sixth highest value for the highest values option in the Tabular Outputs window.

Event/Error Files

In the **Event/Errors Files** page, you can specify whether or not you wish to generate an event input file or a detailed error listing file. Select **Optional Files - Event/Errors Files** from the tree located on the left side of the Control Pathway dialog, to display the options available.



Control Pathway dialog - Event/Errors Files

Input File for the Short Term EVENT Model

Here you can specify whether you wish the model to generate an input runstream file that can be used directly with the EVENT model. To select this option, check the box. The following information must then be specified:

- **Level in EVENT Output File:** You need to specify the level of detail in the EVENT output file. Two options are available:
  - **Detail:** This option produces more detailed summary in the output file. The basic source contribution information is provided along with the hourly average concentration (or total deposition) values for each source for every hour in the averaging period, and a summary of the hourly meteorological data for the event period.
  - **Source Contribution:** This option produces only the source contribution information in the output file. The average concentration or total deposition value (i.e., the contribution) from each source for the period corresponding to the event for the source group is provided.
- **File name:** AERMOD View uses the default file name ProjectName.evi for the EVENT input file but if you wish to change this click on the  button and specify an alternate name.

> The EVENT model can only process one type of output at a time. Therefore, if more than one Output Type (**Concentration**, **Total Deposition**, **Dry Deposition**, and **Wet Deposition**) is selected in the Dispersion Options window, only the events associated with the first output type, in the order stated above, will be included in the EVENT model input file.

Detailed Error Listing File

Here you can request a detailed listing file of all messages generated by the model. The detailed error listing file includes:

- The error and warning messages that are listed as part of the message summaries provided in the main output file.
- Any information messages such as occurrences of calm winds.
- Quality assurance messages.

To select this option, check the box. The following information must then be specified:

- **Extensive output results: ISCST3 and ISC-PRIME Only** - Select the Yes option to obtain detail output results including plume heights, sigmas, etc., for each hour calculated for debugging purposes. No is the default for this option and means that no detail output results will be generated.

> When using the **Extensive Output Results** option, be aware that ISCST3 model will generate very large files, in some cases several hundred megabytes or more. AERMOD View gives you a warning message every time you select this option.

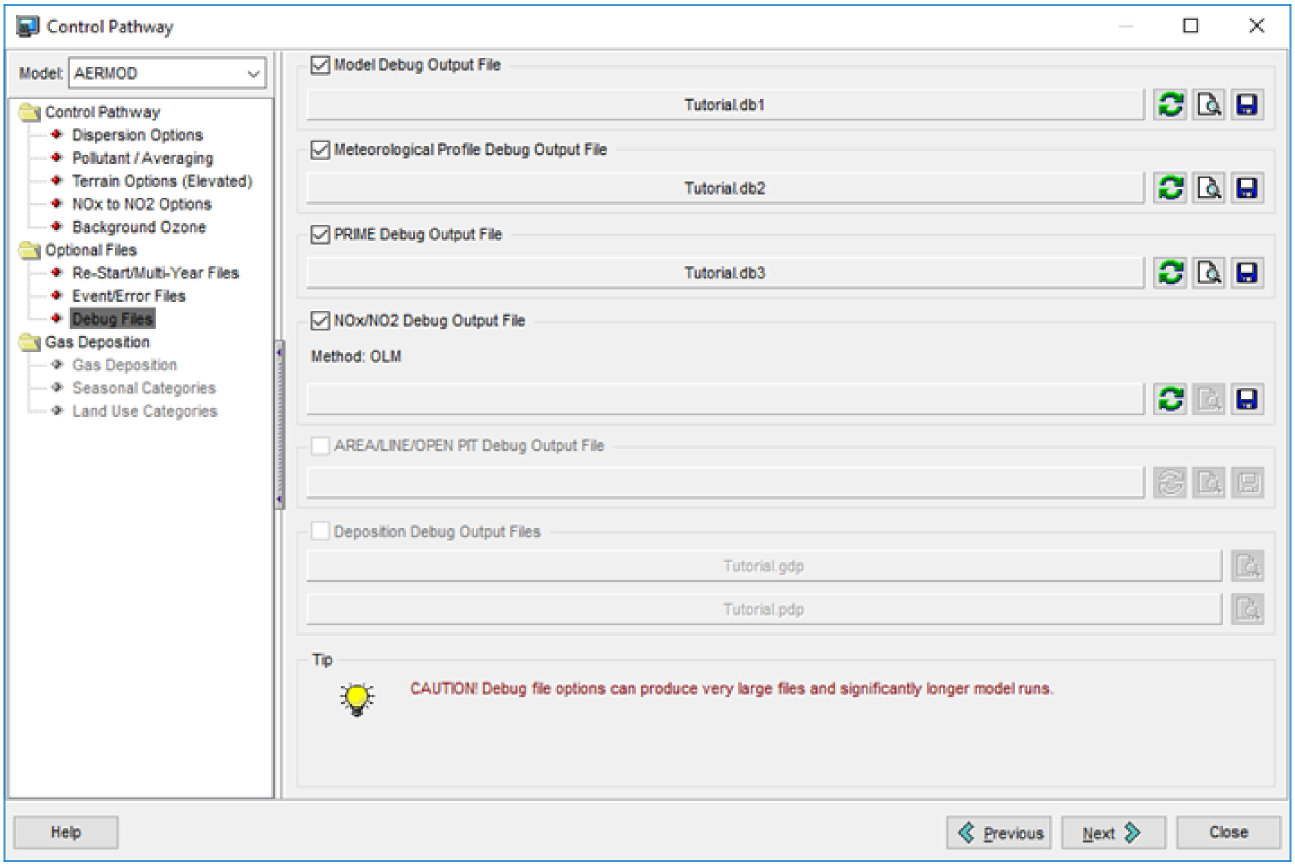
- **File name:** AERMOD View uses the default file name **ProjectName.err** for the detailed error listing file but if you wish to change this click on the button and specify an alternate name.

Debug Files

AERMOD Only

In the **Debug Files** page, you can specify whether or not you wish to generate debug files. Select **Optional Files - Debug Files** from the tree located on the left side of the Control Pathway dialog, to display the options available.

> **Warning:** Debug file options can produce very large files (gigabytes) and significantly longer model runs.

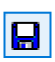


Control Pathway dialog - Debug Files

The following actions are available for the debug files:

 - reload default file name

 - view the file (once the file has been generated)

 - specify file name, other than default

Model Debug Output File

This option produces an debugging output file containing intermediate calculations related to the model results for each source and receptor, such as dispersion parameters, and plume heights. AERMOD View uses the default file name ProjectName.db1 for the model debug file but if you wish to change this click on the  button and specify an alternate name.

Meteorological Profile Debug Output File

This option allows you to request a detailed file with gridded profiles of meteorological variables for each hour of data for debugging purposes. AERMOD View uses the default file name ProjectName.db2 for the meteorological profile debug file but if you wish to change this click on the button and specify an alternate name.

PRIME Debug Output File

This option produces a file that separates the debug information associated with the PRIME downwash algorithm from the non-PRIME related information provided under the Model Debug option. AERMOD View uses the default file name ProjectName.db3 for the PRIME debug file but if you wish to change this click on the button and specify an alternate name.

NOx/NO2 Debug Options

This option produces a debugging output file for the ARM, ARM2, OLM, and PVMRM options. AERMOD View uses the default file name ProjectName.db4 for the NOx/NO2 debug file but if you wish to change this click on the button and specify an alternate name.

AREA/LINE/OPEN PIT Debug Output File

This option produces debugging output file that includes additional information regarding Area, Line, and Open Pit calculations. AERMOD View uses the default file name ProjectName.db5 for the AREA/ Line debug file but if you wish to change this click on the button and specify an alternate name.

Deposition Debug Output Files

This option produces two debugging output files - GDEP.DAT for gas deposition and PDEP.DAT for particle deposition. These file names cannot be changed.

Gas Dry Deposition

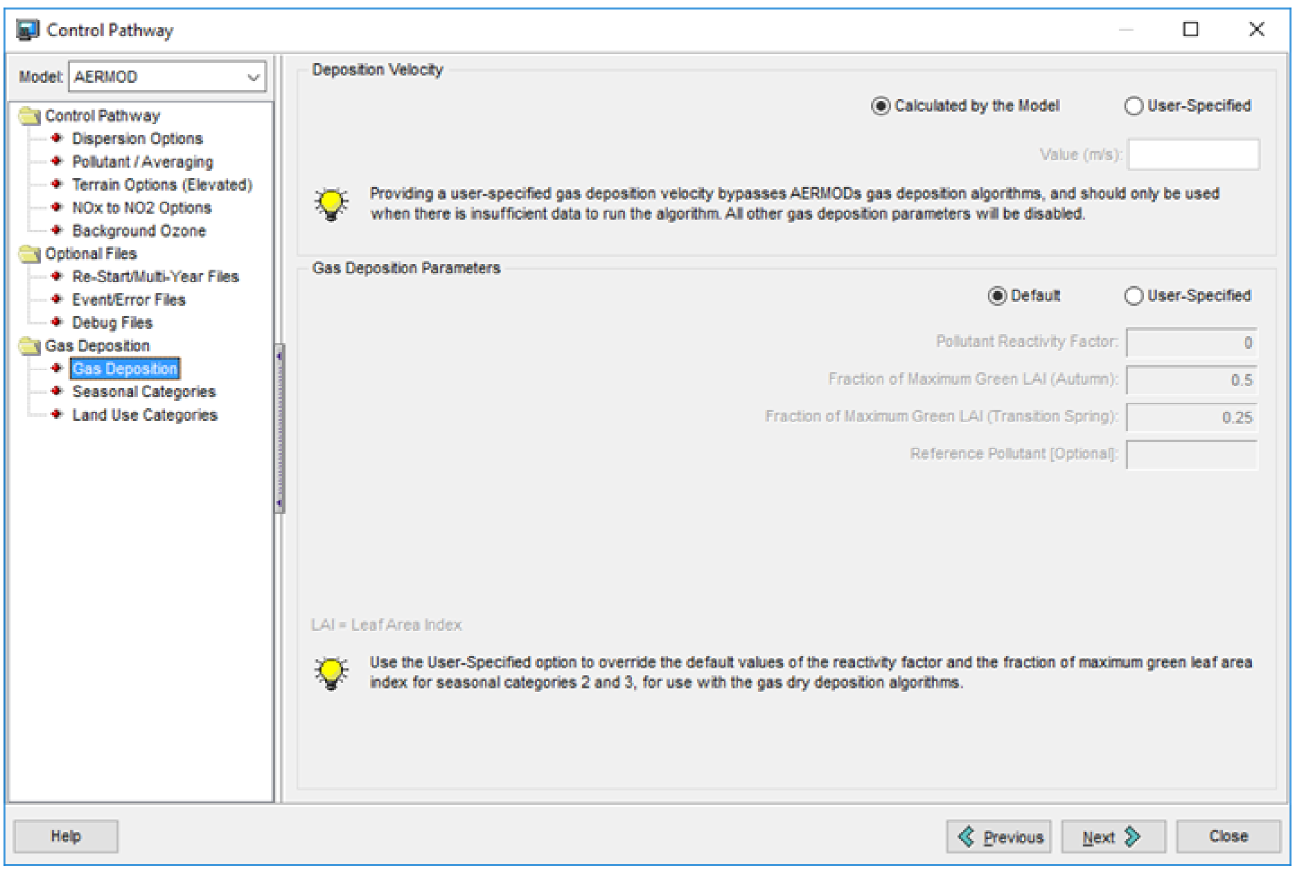
ISCST3 & AERMOD Only

In the **Gas Deposition** page, you can specify the gas dry deposition parameters. Select **Control Pathway - Gas Deposition** from the tree located on the left side of the Control Pathway dialog, to display the available options.

The gas dry deposition parameters required are different for the ISCST3 and AERMOD models. See below for a description of the parameters required for each model:

AERMOD

Gas Deposition section only becomes available if **Gas Deposition** option is selected in the Dispersion Options.



Control Pathway dialog - Gas Dry Deposition (AERMOD)

Deposition Velocity

Here you are requested to either specify a deposition velocity in meters per second or allow the model to calculate the deposition velocities.

- **Calculated by the Model:** If this option is selected the deposition velocities are calculated by the model.
- **User-Specified:** If this option is selected, then a single deposition velocity can be input for a given model run, and is used for all sources of gaseous pollutants. This option will by-pass AERMOD's gas deposition algorithms, and should only be used when sufficient data to run the algorithm are not available. All other gas deposition parameters will be disabled.

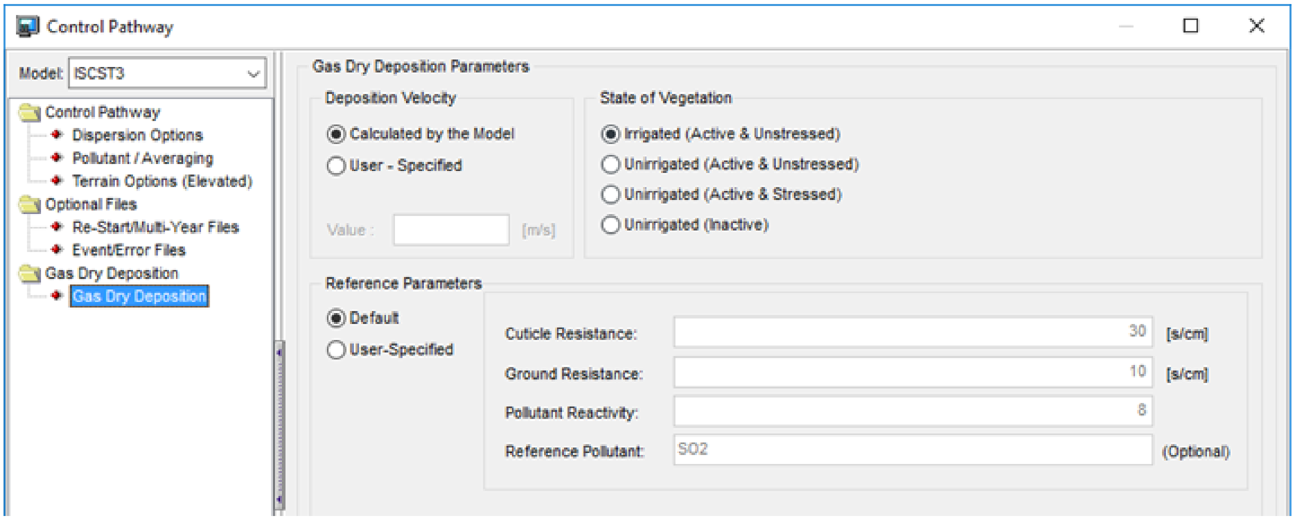
Gas Deposition Parameters

Here you can either use the default values for the gas dry deposition parameters or specify your own values to override the default. The default values used are 0 for the pollutant reactivity factor, 0.5 for the fraction of maximum green leaf area index during Autumn and 0.25 for the fraction of maximum green leaf area index during Transitional Spring. If you use the User-Specified option, you may also specify an optional name for the Reference Pollutant.

> A pollutant reactivity factor of 1 should be used for ozone (O_3), titanium tetrachloride ($TiCl_4$) and divalent mercury (Hg^{2+}) and a value of 0.1 should be input for nitrogen dioxide (NO_2).

> For midsummer, late autumn and winter, a value of 1 is used for the fraction of maximum green leaf area index.

ISCST3



Control Pathway dialog - Gas Dry Deposition (ISCST3)

> The **Gas Dry Deposition** option is available only if you have selected the Non-Default - TOXICS - **Gas Dry Deposition** option in the Dispersion Options window.

The following gas dry deposition parameters must be specified for the ISCST3 model:

Deposition Velocity

Here you are requested to either specify a deposition velocity in meters per second or allow the model to calculate the deposition velocities.

- **Calculated by the Model:** If this option is selected the deposition velocities are calculated by the model. You will then also need to specify the State of Vegetation, Reference Parameters, Source Parameters, and the Meteorological Data File with the additional meteorological variables.

- **User-Specified:** If this option is selected, then a single deposition velocity can be input for a given model run, and is used for all sources of gaseous pollutants. This option will by-pass the algorithm for computing deposition velocities for gaseous pollutants, and should only be used when sufficient data to run the algorithm are not available. Results of the ISCST3 model based on a user-specified deposition velocity should be used with extra caution. If using this option, you also need to specify the Meteorological Data File with the additional meteorological variables.

A non-fatal warning message is generated if a value greater than 0.05 m/s is input for the deposition velocity.

State of Vegetation

For the gas dry deposition algorithm option, you are requested to define the state of the vegetation. The state of vegetation is used in the model, along with ambient temperature and incoming short-wave radiation, to determine the resistance to transport through the stomatal pores. Four options are available:

- Irrigated (Active & Unstressed)
- Unirrigated (Active & Unstressed)
- Unirrigated (Active & Stressed)
- Unirrigated (Inactive)

For unirrigated vegetation, you should select the appropriate option based on existing soil moisture conditions.

Reference Parameters

You can either use the default values for the references parameters or specify your own values to override the default. The reference parameters required for the gas dry deposition algorithm are:

- Cuticle Resistance
- Ground Resistance
- Pollutant Reactivity
- Reference Pollutant (optional)

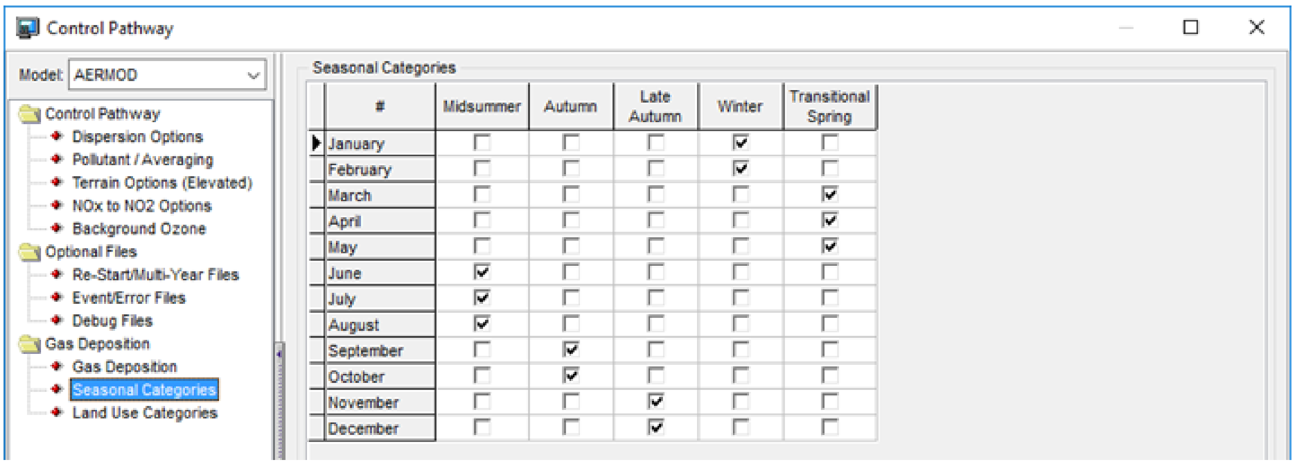
The gas dry deposition parameters for the pollutant being modeled may be found in chemical engineering handbooks and various publications, such as the Air/Superfund National Technical Guidance Study Series (EPA, 1993).

Seasonal Categories

AERMOD Only

The AERMOD deposition algorithms include land use characteristics and gas resistance terms based on five seasonal categories. You must correlate these seasonal categories to each calendar month. In the **Seasonal Categories** page, you can specify the seasonal category for each month of the year. Select **Control Pathway - Seasonal Categories** from the tree located on the left side of the Control Pathway dialog, to display the available options.

> The Seasonal Categories screen is only enabled/available if the Gas Deposition option was selected and Deposition Velocity was set to Calculated by the Model.



Control Pathway dialog - Seasonal Categories

Default seasonal categories have been applied to each month, you can change these at any time. You can either simply check the seasonal category box you wish to use instead for the corresponding month, or click on the Clear All button to clear all the boxes.

The seasonal categories available are -

- **Midsummer**, with lush green vegetation
- **Autumn**, with unharvested cropland
- **Late Autumn**, after frost and harvest, or winter with no snow
- **Winter**, with snow on ground
- **Transitional Spring**, with partial green coverage or short annuals

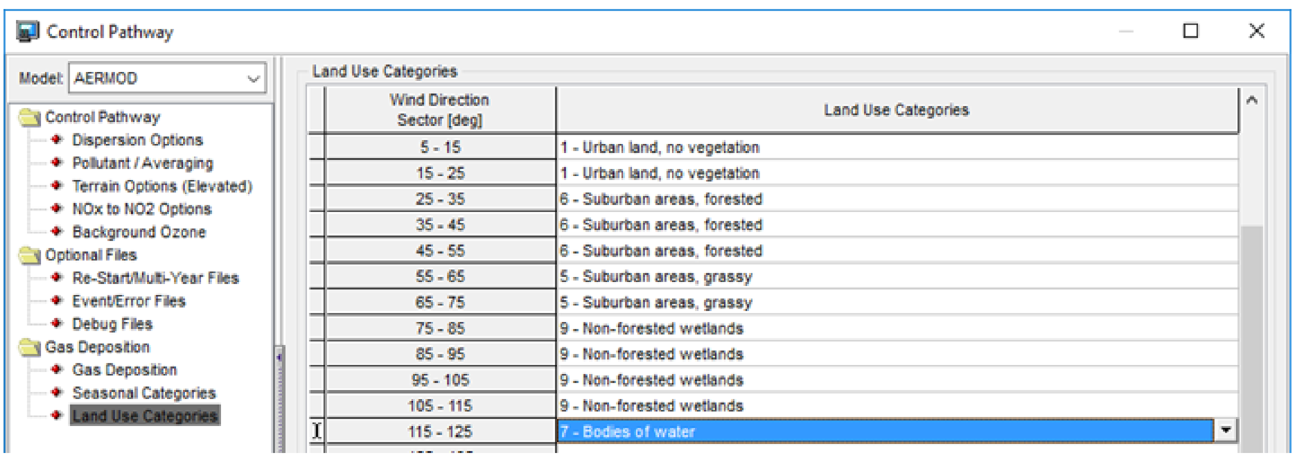
> Some of the seasonal categories defined above may not apply for certain regions, for example the Winter category for southern latitudes.

Land Use Categories

AERMOD Only

The AERMOD deposition algorithms include some gas deposition resistance terms based on five Seasonal Categories and on nine land use categories. For each wind direction sector, you must define the land use category. In the **Land Use Categories** page, you can define the land use category by wind direction. Select **Control Pathway - Land Use Categories** from the tree located on the left side of the Control Pathway dialog, to display the available options.

> The **Land Use Categories** screen is only enabled/available if the Gas Deposition option was selected and Deposition Velocity was set to Calculated by the Model.



Control Pathway dialog - Land Use Categories

There are 36 wind direction sectors, one for every 10 degrees. For example, the wind direction sector 5-15 corresponds to winds blowing toward 10 degrees plus or minus 5 degrees.

The land use categories available are -

- Urban land, no vegetation
- Agricultural land

- Rangeland
- Forest
- Suburban areas, grassy
- Suburban areas, forested
- Bodies of water
- Barren land, mostly desert
- Non-forested wetlands

For each wind direction sector, you can click in the land use category cell and from the drop-down list select one of the above land use categories. If you wish to apply the same category to several wind direction sectors you may select the cells by pressing down the Shift key while you select multiple items in sequence or the Ctrl key to make disjoint selections. Once the cells are selected, choose the category you wish to use for the selected cells from the drop-down list at the bottom of the table and click on the Apply button.

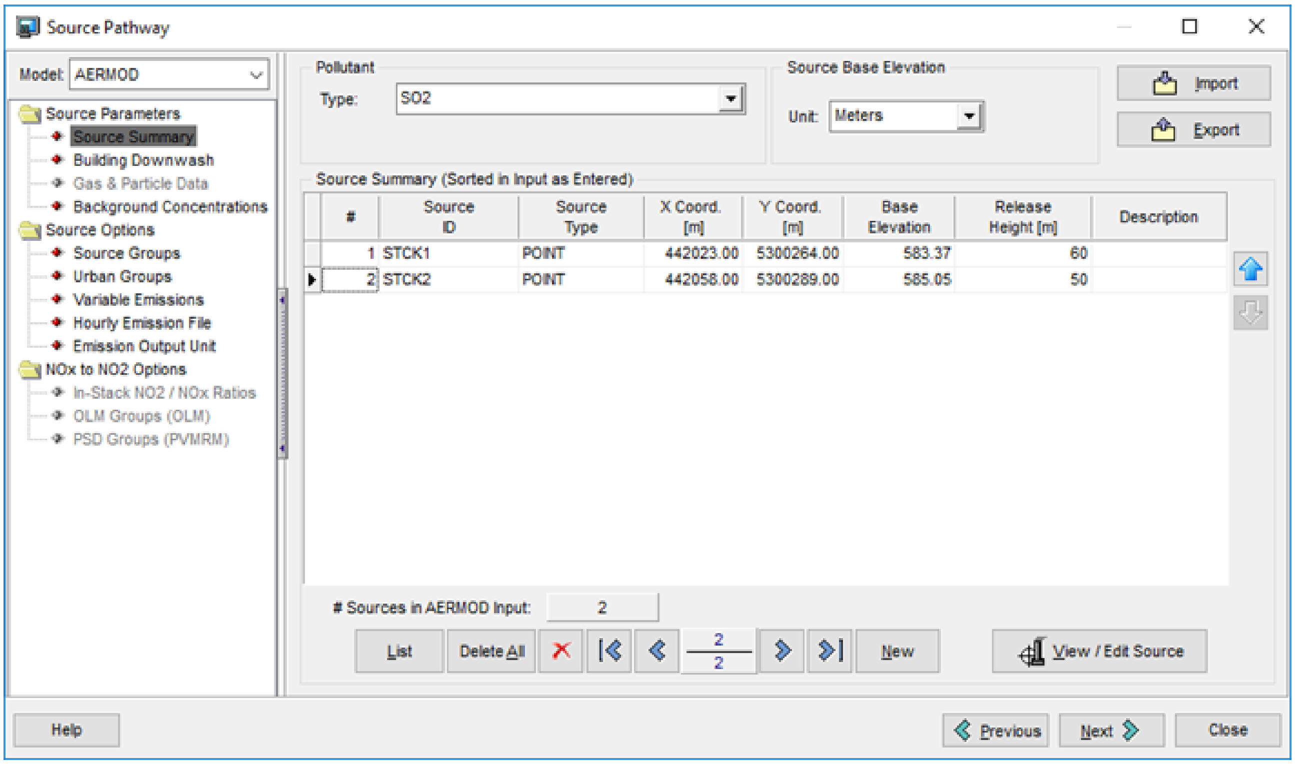
If you wish to apply the same land use category to all the wind direction sectors, click on the Select All button, select the land use category from the drop-down list at the bottom of the table and click on the Apply button.

Source Pathway

The Source Pathway allows you to specify the source input parameters and source group information such as source types, building downwash and variable emissions. You have access to the **Source Pathway** dialog by select **Data | Source Pathway...** from the menu or by clicking on the **Source** 

menu toolbar button.

The **Source Pathway** dialog uses a two-pane view. The tree located on the left side of the dialog is used for navigation and item selection. Select an item (marked as ◆) in the tree to display the available options on the right panel.



Source Pathway dialog

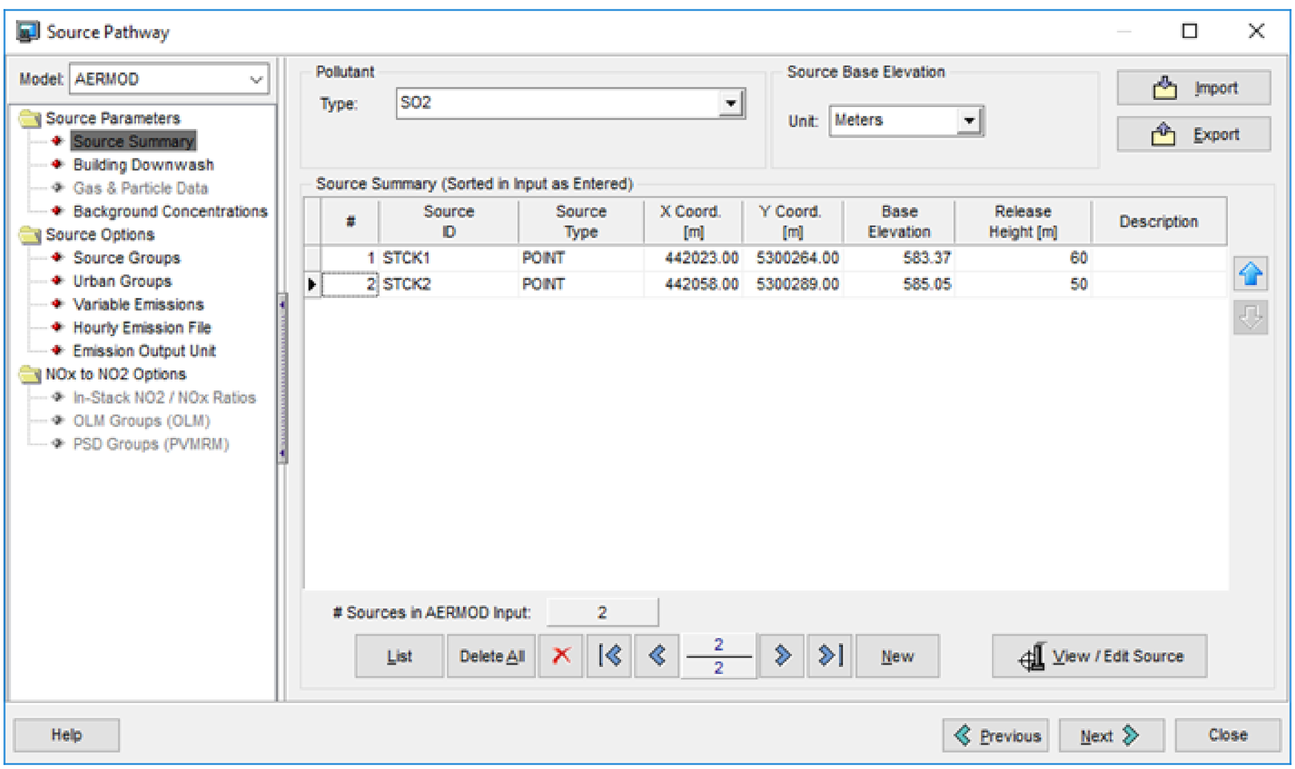
In the **Source Pathway** dialog you can define the following:

- Source Summary
- Building Downwash
- Gas & Particle Data
- Background Concentrations
- Source Groups
- Urban Groups

- Variable Emissions
- Hourly Emission File
- Emission Output Unit
- In-Stack N02 / NOx Ratios
- OLM Groups (OLM)
- PSD Groups (PVMRM)
- Source Inputs
- Import Sources
- Accessory Dialogs

Source Summary

In the **Source Summary** page, you can specify information on the number of sources specified for your current project, pollutant information and source base elevation information. Select **Source Pathway - Source Summary** from the tree located on the left side of the Source Pathway dialog, to display the available options.



Source Pathway dialog - Source Summary

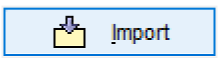
The following options are available -

Pollutant

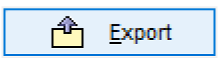
For each run, you need to specify the type of pollutant being modeled. The pollutant displayed here will be same pollutant selected in the Pollutant/Averaging screen of the **Control Pathway**. Changes in the pollutant type in the **Source Pathway** will be automatically displayed in the **Control Pathway** and vice-versa.

Source Base Elevation

Unit: Use the drop-down list to select the units in which the source base elevation is specified (meters or feet).



Click on this button to import sources into your project.



Click on this button to export your sources to an excel spreadsheet.

See Also: Import Sources.

Source Summary

The **Source Summary** displays a summary of all the sources defined in your project. The following information is displayed for each source:

- **#**: AERMOD View automatically creates an entry number for each source created.
- **Source ID:** The ID for the source is displayed here.
- **Source Type:** This column displays source type.
- **Release Type:** This column displays the release type if you have defined a point source (either vertical, capped or horizontal) when the option for capped and horizontal releases is being used.
- **X Coord:** The X Coordinate for the source is displayed here.
- **Y Coord:** The Y Coordinate for the source is displayed here.

See Also: Location of Source X and Y Coordinates for each source type.

- **Base Elevation:** This column displays the base elevation for the sources.

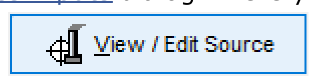
- **Description:** This column displays the description for the source.

**:** Move selected source up in the list.

**:** Move selected source down in the list.

- **# Sources in AERMOD Input:** Displays the number of sources that will be included in the input file for AERMOD.

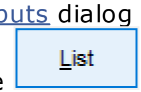
The **# Sources in AERMOD Input** number includes all individual sources generated for any Line type sources, such as Line Area Source and Line Volume Source.

You cannot directly modify the information in the **Source Summary** table but you can reorder sources by dragging and dropping. This can be useful for grouping like sources. If you wish to change source information, select the source and double click to display the Source Inputs dialog where you can modify information. You can also select the source and click on the 

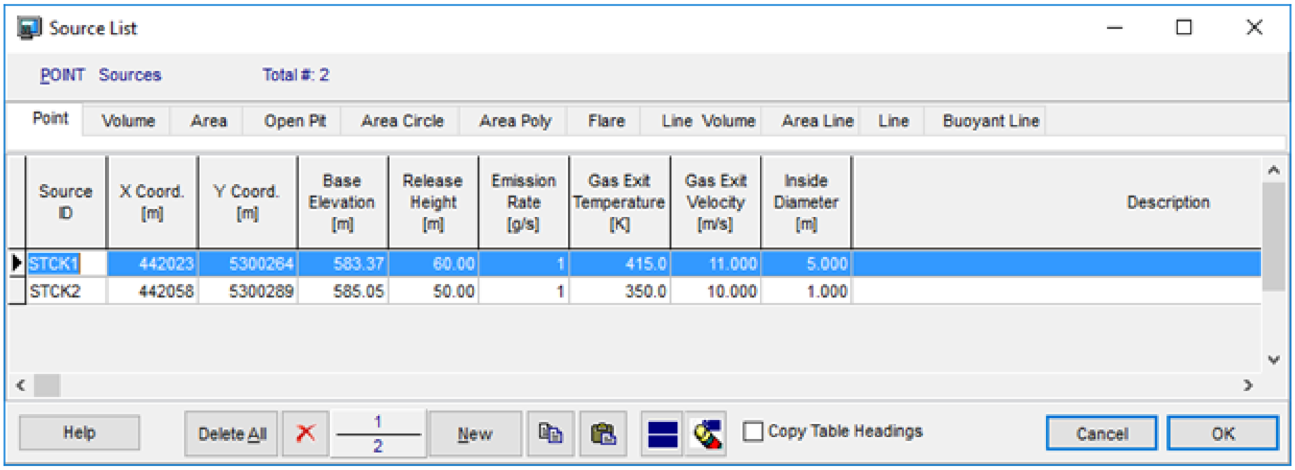
button.

The buttons at the bottom of the Source Summary window, called the Record Navigator buttons, will help you in managing the information that is defined for your sources. Click the **List** button to view the Source List.

Source List

The **Source List** dialog contains a list of all the sources already defined in the Source Inputs dialog for the current run in the order they were created. This dialog is displayed by clicking on the 

button of the Record Navigator located in the Source Pathway | Source Summary dialog.

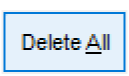


Source List dialog

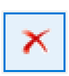
At the top right of the dialog the number of sources already created for the specified source type is displayed. There are eight tabs in this dialog, one for each source type, each displaying the source type specific information.

> To copy a record from the Source List to a Microsoft® Excel® spreadsheet, select the record's line in the table, right-click it and select **Copy** from the drop down list, and paste it in the spreadsheet.

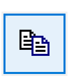
A number of buttons are found at the bottom of this dialog. Their functions are described below:



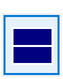
Delete All: Deletes all the records already defined in the current window.


Delete: Deletes the currently selected record.

New: Allows you to create a new record.


Copy: Allows you to copy the contents of the current record to the local clipboard.


Paste: Allows you to paste the contents of the local clipboard into the current record.


Select All: Selects all sources in the list.

 **Unselect All:** Unselects all selected sources in the list.

> To display the Source Inputs dialog for a particular source, double click with the left mouse button on the source.

Location of Source X and Y Coordinates

The location for the X and Y Coordinates for each source type is given in the table shown below. Please note that for AREA POLY and LINE sources this coordinate is for the first vertex or point input by the user.

Source Type	Location for the X & Y Coordinates
POINT	Center of the source
VOLUME	Center of the source
AREA	Southwest corner
OPEN PIT	Southwest corner
AREA CIRC	Center of the source
AREA POLY	First vertex defined for the polygon
FLARE	Center of the source
LINE	First point defined for the line source

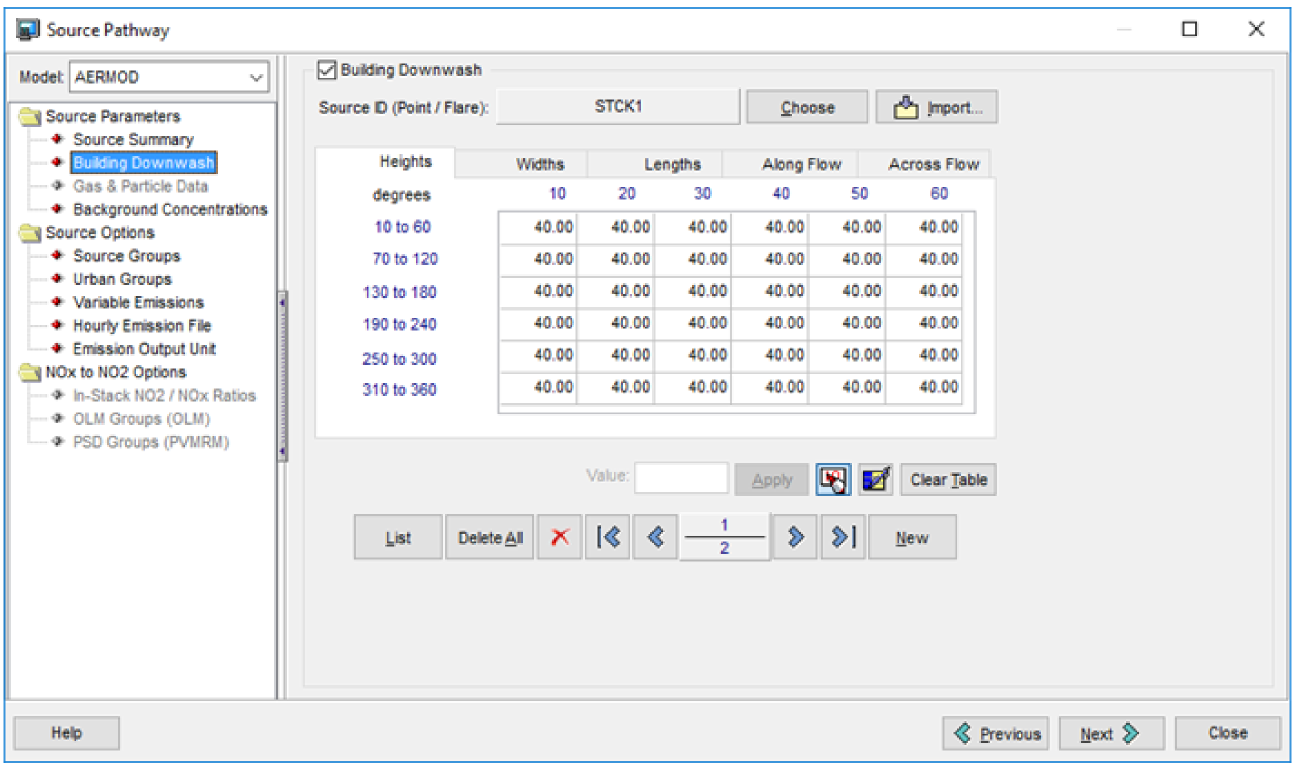
Location for the X and Y Coordinates

Building Downwash

In the **Building Downwash** page, you can specify the building downwash information for your point or flare sources. Select **Source Pathway - Building Downwash** from the tree located on the left side of the Source Pathway dialog, to display the available options.

The U.S. EPA models ISCST3, ISC-PRIME, and AERMOD include algorithms to model the effects of building downwash on emissions from nearby or adjacent point sources. The building downwash algorithms only apply to point sources however since AERMOD View implements the flare source as a point source, the flare sources can also considered for building downwash calculations.

> Refer to Volume II of U.S. EPA's ISC3 User's Guide (US EPA 1995e) for a technical description of the building downwash algorithms.



Source Pathway dialog - Building Downwash

> See Also: What is Building Downwash?

If you wish to specify building downwash for your point or flare sources you will need to check the **Building Downwash** box to enable the **Building Downwash** panel. In this panel you have the option of specifying your building downwash information in two ways -

- **Manually:** When typing in the information you can also use the **Edit/Mark** tools (described below), which allow you to quickly and easily enter data into the table. At the bottom of the **Building Downwash** panel are the Record Navigator buttons, which will help you manage information for multiple sources defined.

- **Import:** You can import the building downwash information from a BPIP output file (*.pro).

  Click on the Choose button to open the Source ID dialog where you can select the source for which you wish to import downwash data. Once the source has been selected the source ID will be displayed. You can then click on the Import... button to import the downwash information for the source.

You can assign the values in the table using the following controls:

Value: Type in the value you wish to assign to selected cells.

Apply - Click to apply the contents of the **Value** field to the selected cells.

- Click this button to be able to manually edit values in individual cells.

- Click this button to be able to highlight/select entire rows of cells. Clicking on an already selected row will clear that selection.

Clear Table - Click to clear all the values in the table.

If you have created your buildings and run BPIP from within AERMOD View, the building downwash information will be automatically displayed for each source. The BPIP Primary Output File (* .pro) contains the essential output data such as the Preliminary GEP stack height values and the BH (Building Height) and PBW (Projected Building Width) input for an ISCST3 input runstream file as well as some additional downwash information to perform enhanced downwash calculations required by the ISC-PRIME and AERMOD models. The primary difference is the inclusion of values for the additional parameters XBADJ and YBADJ in the last portion of the primary output file. AERMOD View will read this file and place all BH, PBW, PBL, XBADJ and YBADJ values for each stack. The downwash information displayed differs depending on which model you are using -

ISCST3

The following information is requested for the building downwash option for ISCST3 -

Building Heights [m] | Building Widths [m] | Lower Bound (Optional)

degrees	10	20	30	40	50	60
10 to 60	40.00	40.00	40.00	40.00	40.00	40.00
70 to 120	40.00	40.00	40.00	40.00	40.00	40.00
130 to 180	40.00	40.00	40.00	40.00	40.00	40.00
190 to 240	40.00	40.00	40.00	40.00	40.00	40.00
250 to 300	40.00	40.00	40.00	40.00	40.00	40.00
310 to 360	40.00	40.00	40.00	40.00	40.00	40.00

- **Building Heights:** This tab displays a table containing 6 columns and 6 rows with a total of 36 cells. You must input all 36 direction-specific building heights in meters, beginning with the 10 degree flow vector (wind blowing toward 10 degrees from north), and incrementing by 10 degrees in a clockwise direction. The column heading displays the directions in degrees for the selected row while the row heading displays the angle range for that specific row (e.g., 10 to 60 deg).

- **Building Widths:** This tab displays a table containing 6 columns and 6 rows with a total of 36 cells. You must input all the 36 direction-specific building widths in meters, beginning with the 10 degree flow vector (wind blowing toward 10 degrees from north), and incrementing by 10 degrees in a clockwise direction.

- **Lower Bound:** This optional tab displays a table containing 6 columns and 6 rows with a total of 36 cells corresponding to an array of 36 lower bound wake option switches, beginning with the 10 degree flow vector and incrementing by 10 degrees in a clockwise direction. The **Lower Bound** option is available only if you have selected to use the Non-Regulatory Default option of calculating "low bound" concentration or deposition values for downwash sources subject to enhanced lateral plume spread by super-squat buildings. A super-squat building is a building whose width is more than five times the height. To indicate the use of the lower bound wake option for a specific sector you should input the value 1 and for use of upper bound input 0:

- Value = 0 means to use the upper bound (regulatory default), or

- Value = 1 means to use the lower bound for that sector (applicable for the Non-Regulatory Default option only).

> If the Regulatory Default option has been selected then the **Lower Bound** tab becomes invisible and any low bound inputs will be ignored, and the model will calculate the upper bound estimates only (value = 0).

If a cell in the **Lower Bound** table is double-clicked, the upper bound value (0) is switched to a lower bound value (1) and vice-versa.

AERMOD & ISC-PRIME

The following information is requested for the building downwash option for AERMOD and ISC-PRIME

Heights | Widths | Lengths | Along Flow | Across Flow

degrees	10	20	30	40	50	60
10 to 60	40.00	40.00	40.00	40.00	40.00	40.00
70 to 120	40.00	40.00	40.00	40.00	40.00	40.00
130 to 180	40.00	40.00	40.00	40.00	40.00	40.00
190 to 240	40.00	40.00	40.00	40.00	40.00	40.00
250 to 300	40.00	40.00	40.00	40.00	40.00	40.00
310 to 360	40.00	40.00	40.00	40.00	40.00	40.00

- **Heights:** The Heights tab displays a table containing 6 columns and 6 rows with a total of 36 cells. The table must contain all 36 direction-specific building heights in meters, beginning with the 10 degree flow vector (wind blowing toward 10 degrees from north), and incrementing by 10 degrees in a clockwise direction. The column heading displays the directions in degrees for the selected row while the row heading displays the angle range for that specific row (e.g., 10 to 60 deg).

- **Widths:** The Widths tab displays a table containing 6 columns and 6 rows with a total of 36 cells. The table must contain all the 36 direction-specific building widths in meters, beginning with the 10 degree flow vector (wind blowing toward 10 degrees from north), and incrementing by 10 degrees in a clockwise direction.

- **Lengths:** The Lengths tab displays a table containing 6 columns and 6 rows with a total of 36 cells. The table must contain all the 36 projected length of the building along the flow in meters, beginning with the 10 degree flow vector (wind blowing toward 10 degrees from north), and incrementing by 10 degrees in a clockwise direction.

- **Along Flow:** The Along Flow tab displays a table containing 6 columns and 6 rows with a total of 36 cells. The table must contain all the 36 along-flow distance, in meters, from the stack to the center of the upwind face of the projected building, beginning with the 10 degree flow vector (wind blowing toward 10 degrees from north), and incrementing by 10 degrees in a clockwise direction.

- **Across Flow:** The Across Flow tab displays a table containing 6 columns and 6 rows with a total of 36 cells. The table must contain all the 36 across-flow distance, in meters, from the stack to the center of the upwind face of the projected building, beginning with the 10 degree flow vector (wind blowing toward 10 degrees from north), and incrementing by 10 degrees in a clockwise direction.

What is Building Downwash?

Building Downwash occurs when the aerodynamic turbulence induced by nearby buildings cause a pollutant emitted from an elevated source to be mixed rapidly toward the ground (downwash), resulting in higher ground-level concentrations.

"If stacks for new or existing major sources are found to be less than the height defined by EPA's refined formula for determining GEP height, then air quality impacts associated with cavity or wake effects due to the nearby building structures should be determined." (EPA 1986).

GEP Stack Height = H + 1.5L

(EPA's refined formula for determining GEP stack height)

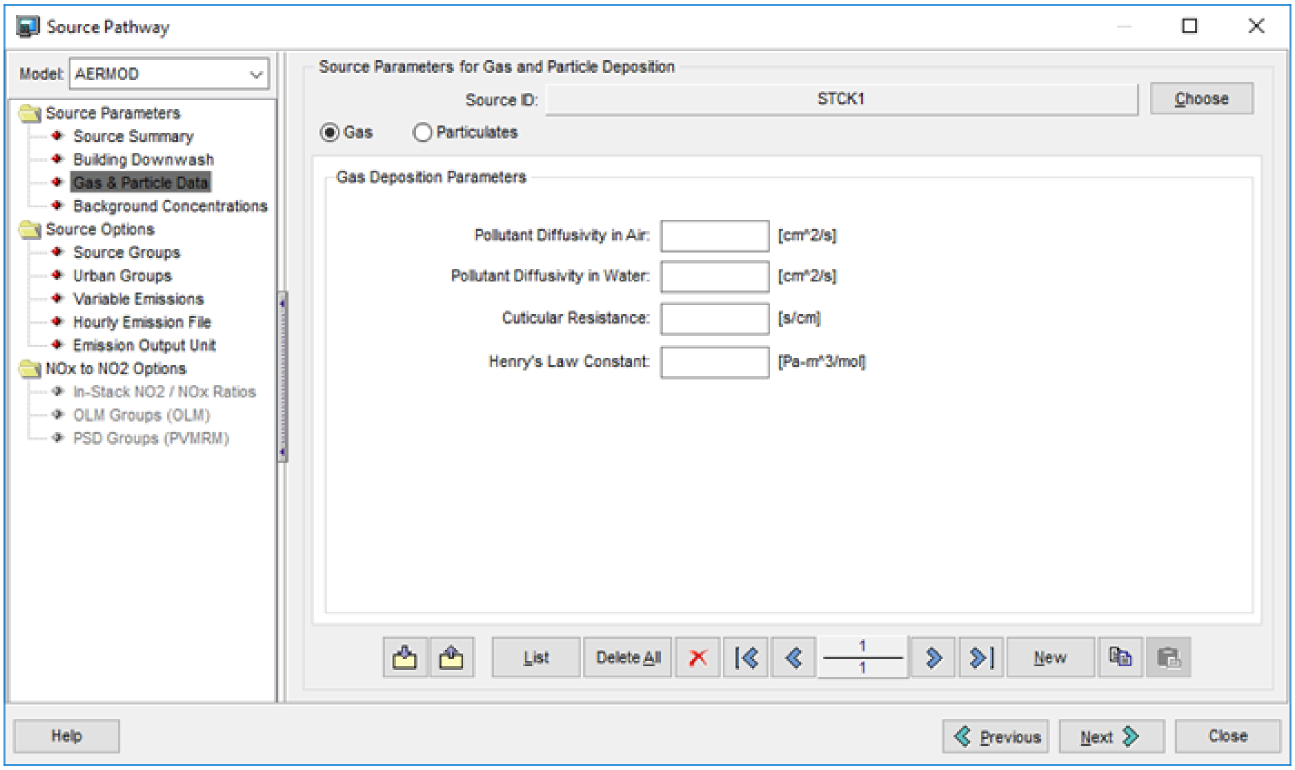
You should consider building downwash for point sources that are within the GEP 5L Area of Influence of a building. For point sources within the GEP 5L Area of Influence, building downwash information (direction-specific building heights and widths) should be included in your ISC3 modeling project. Using AERMOD View, you can easily calculate these direction-specific building heights and widths.

For regulatory applications, a building is considered sufficiently close to a stack to cause wake effects when the distance between the stack and the nearest part of the building is less than or equal to five (5) times the lesser of the building height or the projected width of the building.

Distance from stack-bldg <= 5L

Gas & Particle Data

In the **Gas & Particle Data** dialog you can specify gas phase options, particle data, and scavenging coefficients. Select **Source Pathway - Gas & Particle Data** from the tree located on the left side of the Source Pathway dialog, to display the available options.



Source Pathway dialog - Gas & Particle Data

The parameters requested in the **Gas & Particle Data** dialog depend on the type of calculations being performed and the source phase being modeled:

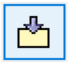
- Gas Phase Options (This option is only available if **Total Deposition** and **Gas Deposition** are selected in Dispersion Options of the Control Pathway)

- Particle Phase Options

> If tabs **Vapor Mercury**, **Vapor**, **Particle**, and **Particle Bound** are present, this means that Risk Mode is enabled. See Risk Mode - Gas & Particle Data for more details.

Gas & Particle Data can be specified for more than one source. The buttons at the bottom of the **Gas & Particle Data** dialog called the Record Navigator buttons will help you manage information for multiple sources defined.

You can also import gas & particle data for your sources, this is useful when you have several sources requiring gas and particle data.

 Click on this button to import the gas and particle data for your sources from a *.xls or *.xlsx file

 Click on this button to export the gas and particle data for your sources to a *.xls or *.xlsx file

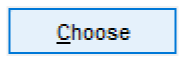
> The data will be exported automatically in the Lakes Format. Data that is imported from Excel will need to be in this format as well.

Gas Phase Options

The **Gas Phase** options are only available if you have selected either -

- Wet Deposition and/or Wet Depletion and Model Options | Gas Deposition in the Dispersion Options window (**AERMOD Only**), or

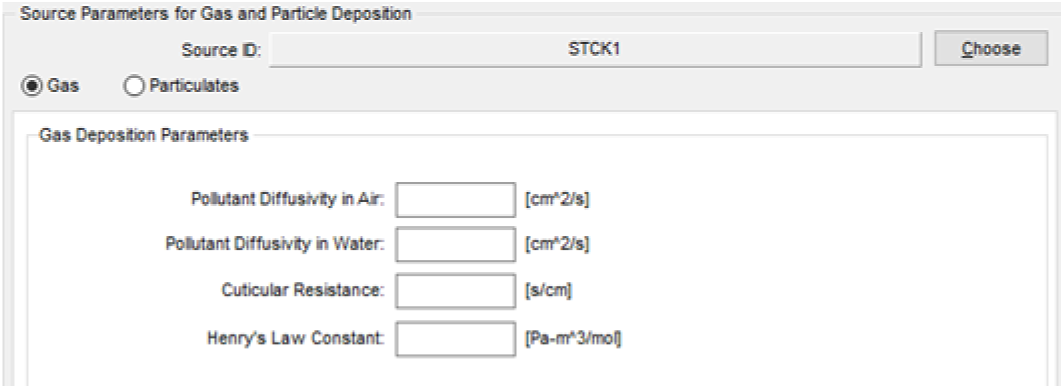
- Non-Default and TOXICS and Gas Dry Deposition option from the Dispersion Options window (**ISCST3 only**).

You must specify the source for which the gas data applies. Click the 

button to display the Source ID dialog, from where you can select the desired source. The gas phase options to be specified depend on which model is being used:

AERMOD

The AERMOD model requires the following gas phase options to be specified:

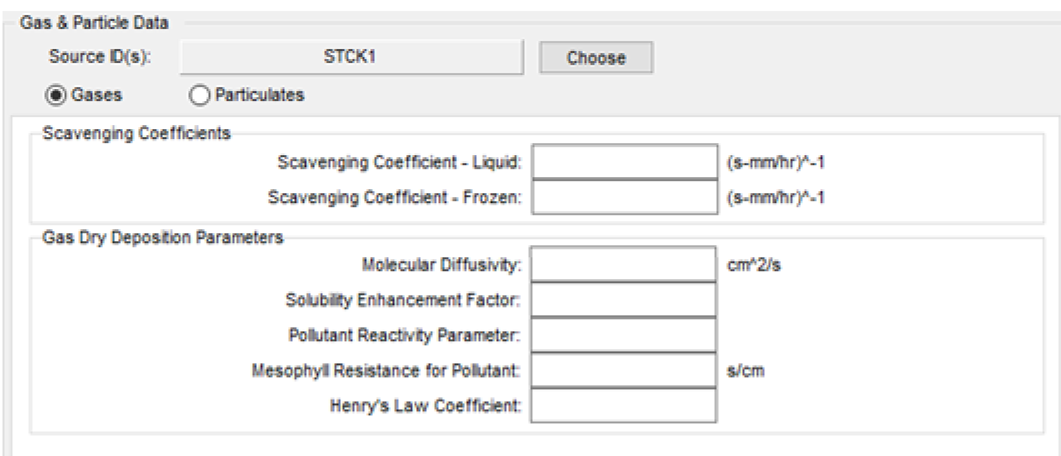


- **Pollutant Diffusivity in Air:** This is the diffusivity in air for the pollutant being modeled.
- **Pollutant Diffusivity in Water:** This is the diffusivity in water for the pollutant being modeled.
- **Cuticular Resistance:** This is the cuticular resistance to uptake by lipids for individual leaves.
- **Henry's Law Constant:** This is the Henry's Law constant.

The gas dry deposition parameters for the pollutant being modeled may be found in chemical engineering handbooks and various publications, such as the Air/Superfund National Technical Guidance Study Series (EPA, 1993).

ISCST3/ISC-PRIME

The ISCST3 and ISC-PRIME models require the following gas phase options to be specified:



Scavenging Coefficients

In this panel you must specify the liquid and frozen scavenging coefficients if you have specified Wet Deposition and/or Wet Depletion.

- **Scavenging Coefficient - Liquid:** Specify the scavenging coefficient for liquid precipitation.
- **Scavenging Coefficient - Frozen:** Specify the scavenging coefficient for frozen precipitation.

> See also Scavenging Rate Coefficients

Gas Dry Deposition Parameters

If you have selected the Gas Dry Deposition option you must specify the following parameters in this panel -

- **Molecular Diffusivity:** This is the molecular diffusivity for the pollutant being modeled.
- **Solubility Enhancement Factor:** This is the solubility enhancement factor for the pollutant. This parameter is only used when applying the algorithm over a water surface. If no water surfaces are present in a particular application, then dummy (non-zero) values may be input for this parameter.
- **Pollutant Reactivity Parameter:** This is the pollutant reactivity parameter.
- **Mesophyll Resistance for Pollutant:** This is the mesophyll resistance term for the pollutant.
- **Henry's Law Coefficient:** This is the Henry's Law coefficient for the parameter. This parameter is only used when applying the algorithm over a water surface. If no water surfaces are present in a particular application, then dummy (non-zero) values may be input for this parameter.

Scavenging Rate Coefficients

ISCST3 & ISC-PRIME Only

The ISCST3 and ISC-PRIME models include algorithms to handle the scavenging and removal by wet deposition, i.e. precipitation scavenging of gases and particulates. The scavenging coefficient depends on the characteristics of the pollutant such as solubility and reactivity for gases and size distribution for particles and the type of precipitation (liquid or frozen).

A more detailed description of scavenging rate coefficients for wet deposition can be found on Vol. II of the US EPA User's Guide for the Industrial Source Complex (ISC3) Dispersion Models, Section 1.4 - The ISC Short-Term Wet Deposition Model (US EPA 1995e). As an initial approximation, you may use scavenging coefficients from Jindal, M., Heinold, D., 1991.

The table below contains some approximated wet scavenging coefficient values for liquid precipitation that were extracted from Figure 1-11 of the U.S. EPA ISCST3 User's Guide, which presents wet scavenging rate coefficients as a function of particle size from the Jindal & Heinold, 1991 publication.

Table 1 - Wet scavenging rate coefficients as a function of particle size

Particle Diameter (microns)	Scavenging Rate Coefficient $(s\text{-}mm/hr)^{-1}$
0.2	1.25E-04
0.3	0.8E-04
0.4	0.6E-04
0.5	0.5E-04
0.6	0.4E-04
0.7	0.4E-04
0.8	0.4E-04
0.9	0.4E-04
1.0	0.4E-04
2.0	1.4E-04
3.0	2.2E-04
4.0	2.8E-04
5.0	3.6E-04
6.0	4.2E-04
7.0	4.6E-04

Particle Diameter (microns)	Scavenging Rate Coefficient $(s\text{-}mm/hr)^{-1}$
8.0	5.2E-04
9.0	6.0E-04
10.0	6.8E-04
>10	6.8E-04

The scavenging rate coefficients for frozen precipitation are expected to be reduced to about 1/3 of the values presented in the above table. The scavenging coefficient units follow from EPA User's Guide ISC Vol. I & II (US EPA 1995d,e).

$$\text{Scavenging coefficient} = \frac{\text{scavenging ratio s\^{}-1}}{\text{rate of rain fall mm h\^{}-1}} = (\text{ s-mm/h)\^{}-1}$$

Particle Phase Options

The **Particle Phase** options are available only if one or more of the following options were selected:

- Dry Deposition
- Wet Deposition
- Total Deposition
- Dry Depletion (plume depletion due to dry removal)
- Wet Depletion (plume depletion due to wet removal)
- Open Pit Source

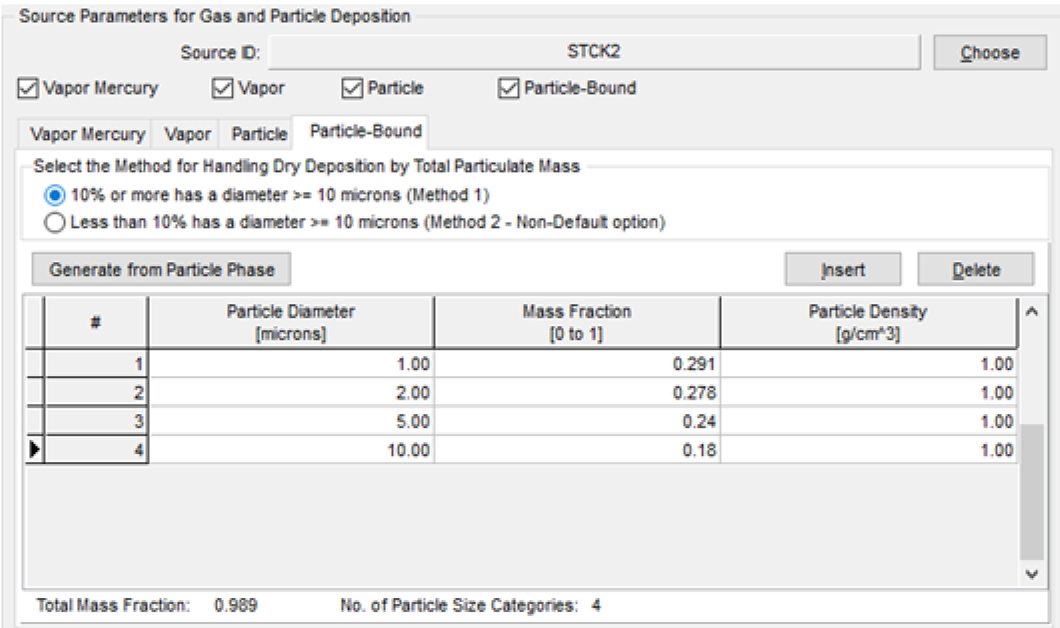
You must specify the source for which the particle data applies. Click the Choose button to display the Source ID dialog, from where you can select the desired source. The particle phase options to be specified depend on which model is being used:

AERMOD

The AERMOD model includes two methods for handling dry deposition of particulate emissions -

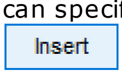
Method 1

This method is used when a significant fraction (greater than 10 percent) of the total particulate mass has a diameter of 10 microns or larger. The particle size distribution must be known reasonably well in order to use **Method 1**.



Particle Phase Options - Method 1

With **Method 1** you must specify the following parameters:

- **No.:** Identifies the order number of particle size category specified for a particular source. You can specify up to a maximum of 20 categories. You may add a category by clicking on the 

button after you have entered information for the previous category.

- **Particle Diameter:** Enter in this column the particulate diameter in microns for each particle size category, up to a maximum of 20.

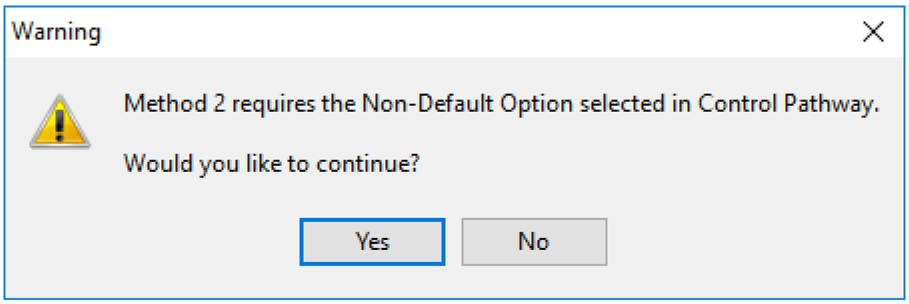
- **Mass Fraction:** In this column, you should define the mass fractions (between 0 and 1) for each of the categories you have defined. The mass fraction for each source must add up to 1.0 (within 2%). Note that the current total for the mass fraction is displayed on the bottom of the table.

- **Particle Density:** Here you can define the particle density in grams per cubic centimeter for each of the categories you have defined.

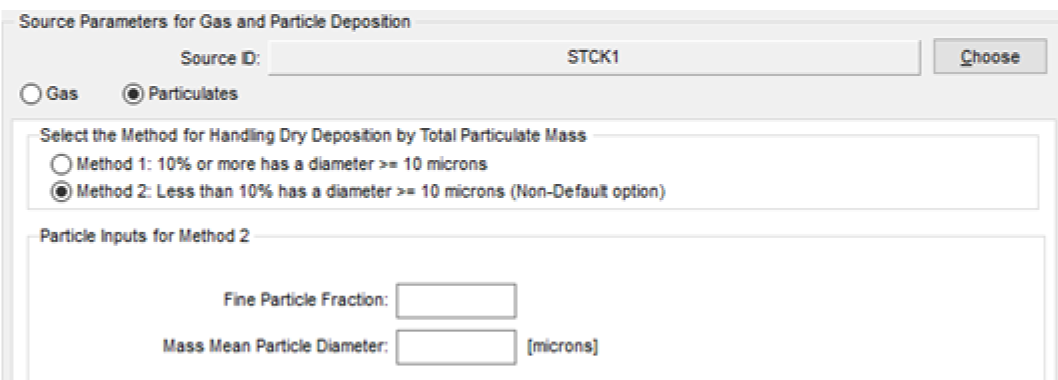
Method 2

This method is used when the particle size distribution is not well known and a small fraction (less than 10 percent of the mass) is in particles with a diameter of 10 microns or larger. The deposition velocity for Method 2 is given as the weighted average of the deposition velocity for the coarse mode, that is, greater than 2.5 microns but less than 10 microns in diameter.

Method 2 is a non-default option. If you select **Method 2** the following message will appear:



Clicking **Yes** will select **Non-Default Options** in the Dispersion Options of Control Pathway.



Particle Phase Options - Method 2

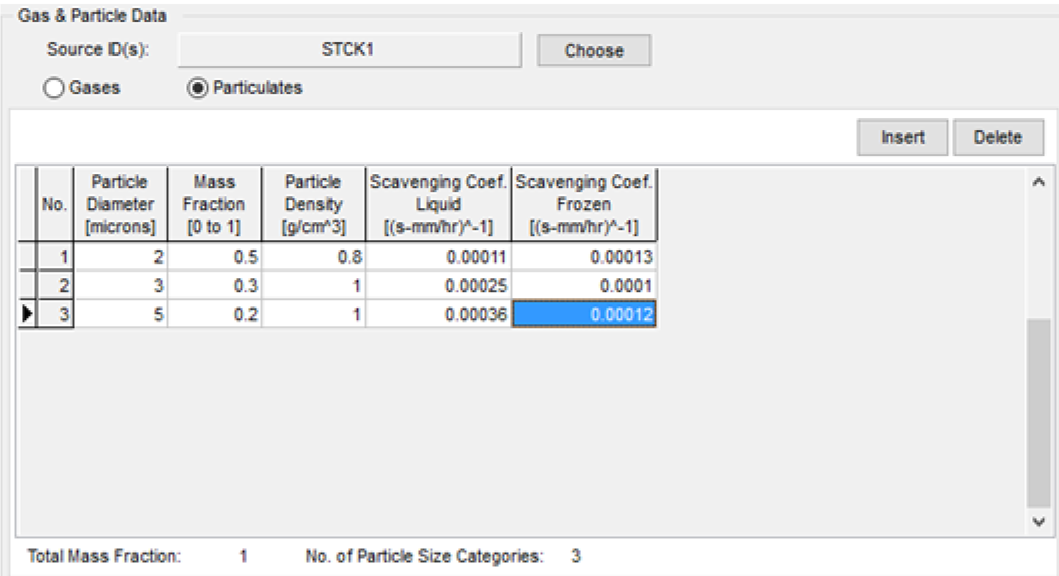
With **Method 2** you must specify the following parameters:

- **Fine Particle Fraction:** Enter in this column the fraction of particle mass emitted in the fine mode, less than 2.5 microns.

- **Mass Mean Particle Diameter:** Specify here the representative mass mean particle diameters in microns.

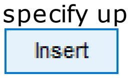
ISCST3/ISC-PRIME

The ISCST3 and ISC-PRIME models require the following particle phase options to be specified:



Particle Phase Options

You must specify the following information if you are modeling the particle-phase of a source:

- **No.:** Identifies the number of particle size categories specified for a particular source. You can specify up to a maximum of 20 categories. You may add a category by clicking on the 

 button after you have entered information for the previous category.

- **Particle Diameter:** Enter in this column the particulate diameter in microns for each particle size category, up to a maximum of 20.

- **Mass Fraction:** In this column, you should define the mass fractions (between 0 and 1) for each of the categories you have defined. The mass fraction for each source must add up to 1.0 (within 2%). Note that the current total for the mass fraction is displayed on the bottom of the table.

- **Particle Density:** Here you can define the particle density in grams per cubic centimeter for each of the categories you have defined.

- **Scavenging Coefficient - Liquid:** Enter in this column the particulate scavenging coefficient for liquid precipitation. The liquid scavenging coefficient should be entered for each particle size category.

- **Scavenging Coefficient - Frozen:** Enter in this column the particulate scavenging coefficient for frozen precipitation. The frozen scavenging coefficient should be entered for each particle size category.

> You must specify scavenging coefficients in the particle phase options for the source if one or more of the following options were selected:
> - Wet Deposition
> - Total Deposition
> - Wet Depletion

Background Concentrations

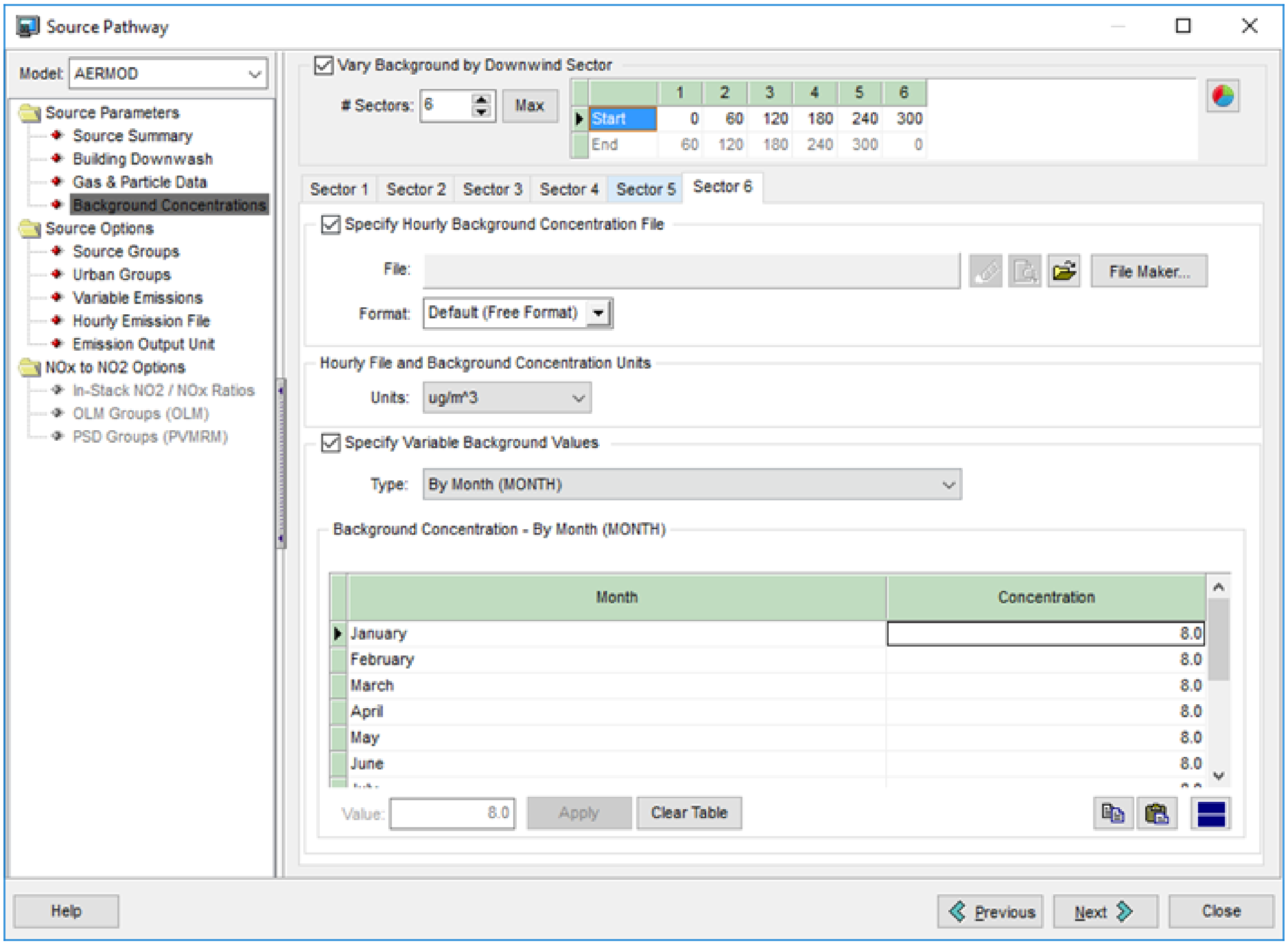
AERMOD Only

In the **Background Concentrations** screen, you can specify uniform or temporally-varying background concentration and apply it to one or more Source Groups. Select **Source Pathway - Background Concentrations** from the tree located on the left.

> You can only specify one background concentration. If you have more than one source group you cannot have different background concentrations for different source groups. You can either apply background concentration to a source group or not.

Available beginning with version 11059

This option allows you to specify uniform or temporally varying background concentrations using the **BACKGRND** keyword on the **SO Pathway**. Background concentrations can be included with any source group to estimate cumulative ambient impacts. Background concentrations can be specified using a range of options similar to those available with the Variable Emissions, and/or on an hourly basis from a separate data file.

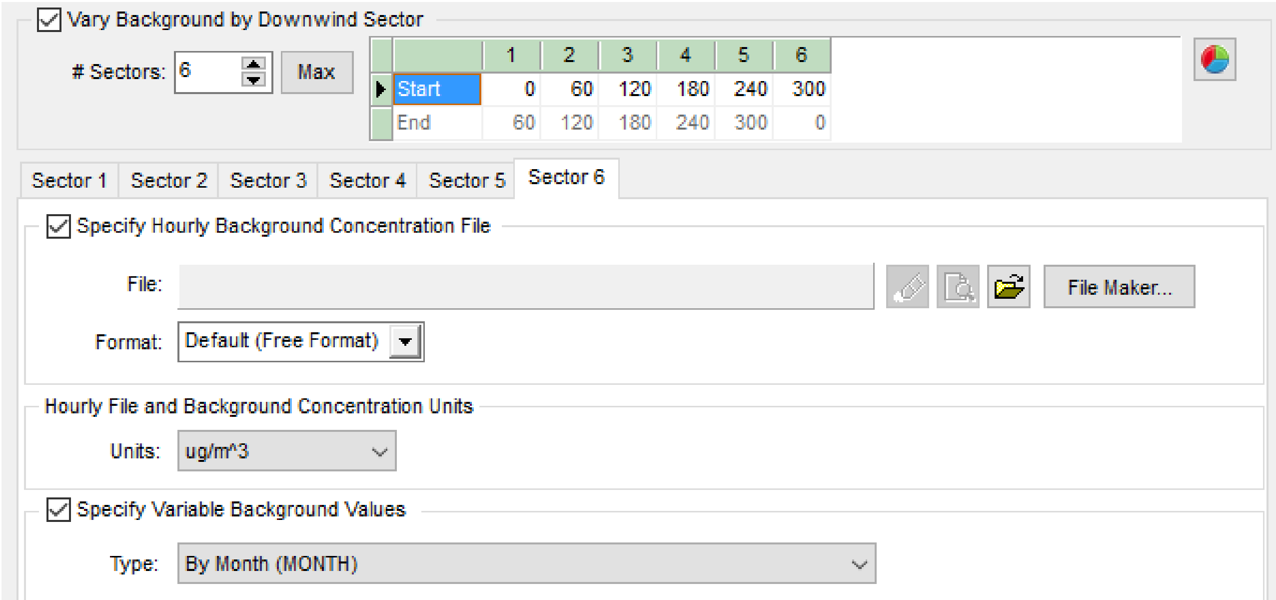


Source Pathway Dialog - Background Concentrations

Vary Background by Downwind Sector

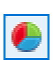
Available beginning with version 13350

This option allows you to specify the background concentrations based on the flow vector sectors.



Source Pathway Dialog - Background Concentrations - Vary by Sector enabled

Check the **Vary Background by Downwind Sector** box to enable this feature.

- **# Sectors:** Enter the number of sectors you wish to create. Click the **Max** button to create the maximum number of sectors (6). The sectors must conform to the following rules:
  - 0 degrees indicates North
  - Sectors must be defined in a clockwise direction starting with the sector that begins closest to 0 degrees.
  - Sectors must be defined using the meteorological degree notation.
  - Each sector must be at least 30 degrees.
  - **Start** coordinates MUST increase (i.e. you cannot define sectors that start with coordinates that are less than an already defined sector).
- Click the  button to view the graphical distribution of sectors once they have been defined.
- **Sectors:** Enter the start point (degrees) for each sector.

> The sectors are defined based on the **Flow Vector** not **Wind Direction**. The **Flow Vector** states where the wind is blowing TO, while the **Wind Direction** states where the wind is blowing FROM.

AERMOD View will create a tab for each sector. Use these tabs to specify the background concentrations using the sections described below.

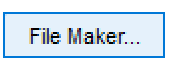
Specify Hourly Background Concentration File

Check the box to specify that an hourly background concentration file will be used.

You must the provide the following information:

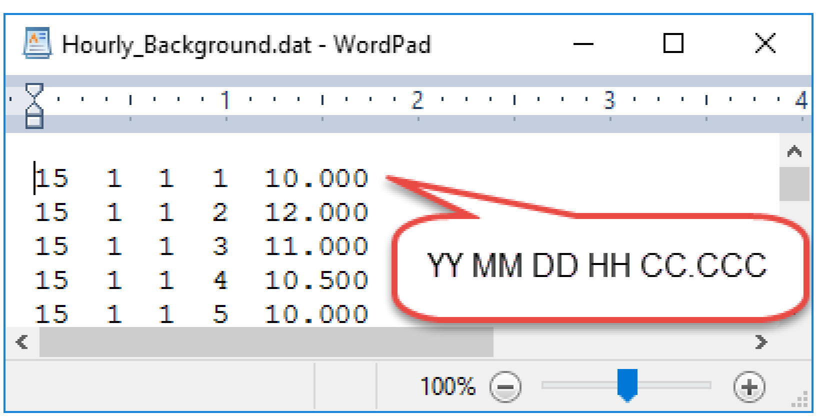
- **File:** Click on the  button to specify the name and location of the hourly background concentration file.

- **Format:** Here you can select from the drop-down list the format of the file, either **Default (Free Format)** or **Fortran**.



Click on this button to open the Concentration File Maker utility where you can create an hourly concentration data file.

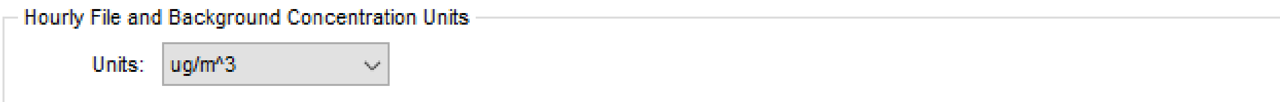
The hourly background file must include the year, month, day, and hour, followed by the background concentration, in that order. The year can be specified as either a 2-digit or 4-digit year.



If an optional Fortran format is specified , the year, month, day, and hour variables must be read as integers using the Fortran I format, and the background concentration must be read as a real variable, using the Fortran F, E, or D format, e.g., (4I2,F8.3).

> Missing values in the background concentration file should be represented by values <0 and >=900 (e.g., -99, 900).

Hourly File and Background Concentration Units

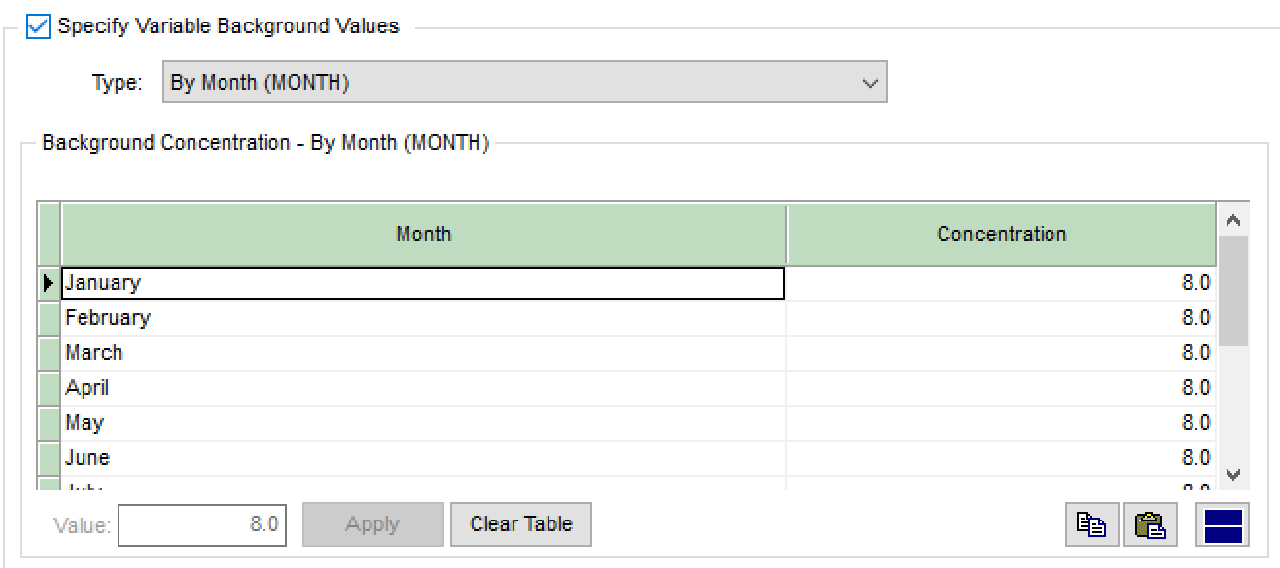


Units: Specify in which units the background concentration values are provided. Select from the drop-down list, either ug/m^3, PPB, or PPM.

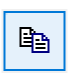
> Background concentrations specified in units of PPB or PPM are converted by the AERMOD model to ug/m3 based on reference temperature (25 °C) and pressure (1013.25 mb).

Specify Variable Background Values

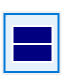
Check the box to specify that background concentrations will be specified in this section.



- **Type:** The drop-down list box offers several options for the specification of background concentration values.
    - **ANNUAL** - annual background value (n=1),
    - **SEASON** - background values vary seasonally (n=4),
    - **MONTH** - background values vary monthly (n=12),
    - **HROFDY** - background values vary by hour-of-day (n=24),
    - **WSPEED** - background values vary by wind speed (n=6),
    - **SEASHR** - background values vary by season and hour-of-day (n=96),
    - **HRDOW** - background values vary by hour-of-day, and day-of-week [M-F, Sat, Sun] (n=72),
    - **HRDOW7** - background values vary by hour-of-day, and the seven days of the week [M, Tu, W, Th, F, Sat, Sun] (n=168),
    - **SHRDOW** - background values vary by season, hour-of-day, and day-of-week [M-F, Sat, Sun] (n=288),
    - **SHRDOW7** - background values vary by season, hour-of-day, and the seven days of the week [M, Tu, W, Th, F, Sat, Sun] (n=672),
    - **MHRDOW** - background values vary by month, hour-of-day, and day-of-week [M-F, Sat, Sun] (n=864), and
    - **MHRDOW7** - background values vary by month, hour-of-day, and the seven days of the week [M, Tu, W, Th, F, Sat, Sun] (n=2,016).

 - Copy values in selected cells

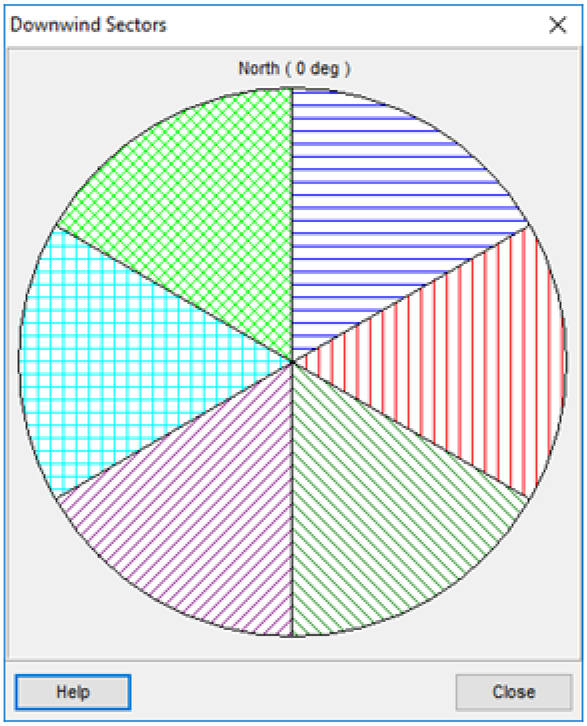
 - Paste copied values to the selected cell and ones below (or to an Excel sheet)

 - Select all rows

See Also: How to Include Background Concentrations for Source Groups

Downwind Sectors

This diagram shows the currently defined distribution of downwind sectors.

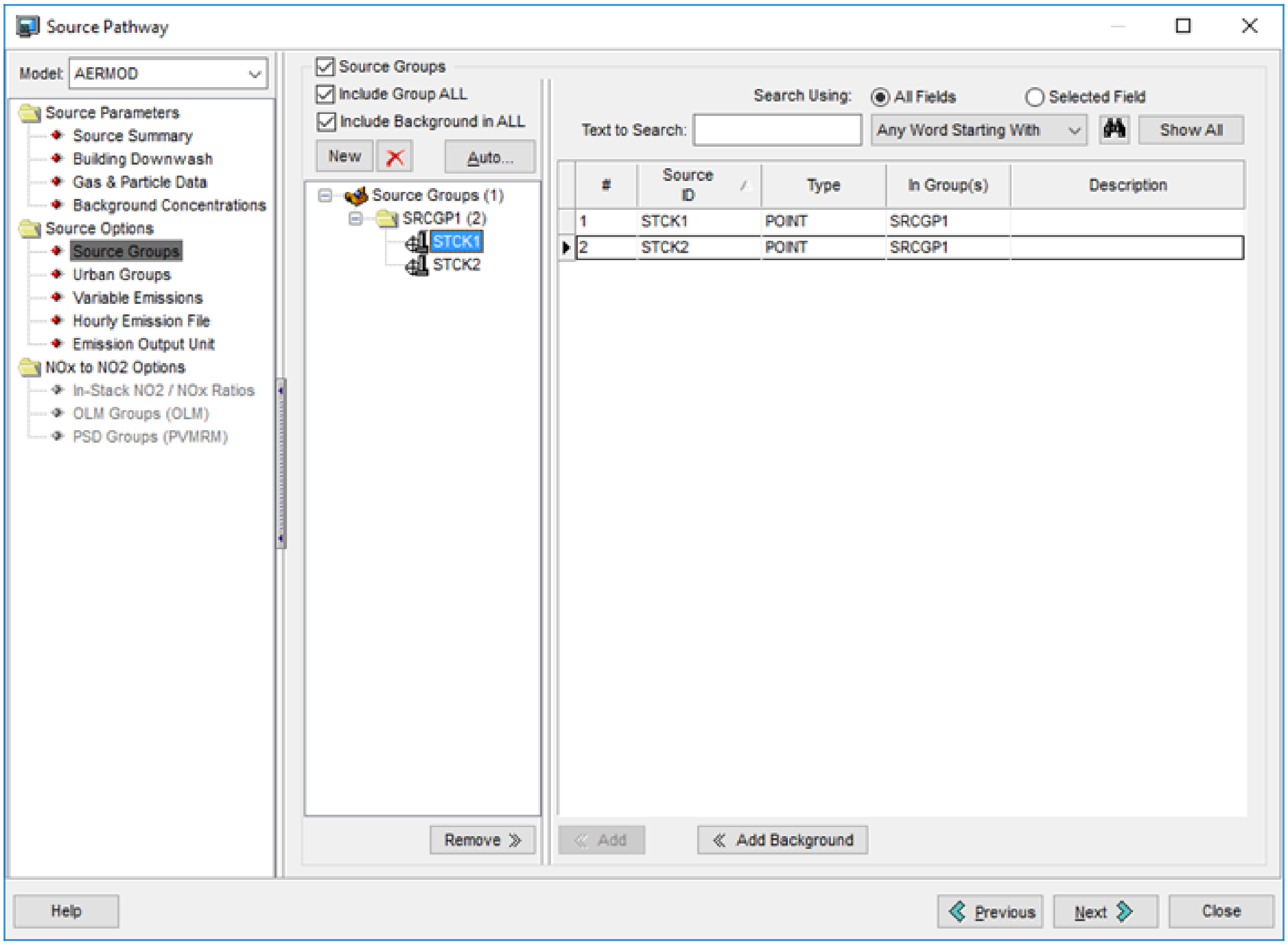


Downwind Sectors

These sectors correspond to the ones defined in the **Vary by Downwind Sector** section for either ozone (Control Pathway - Background Ozone) or background concentrations (Source Pathway - Background Concentrations).

Source Groups

In the **Source Groups** screen, you can analyze group contributions from particular sources together by creating source groups. Sources can exist in more than one group at a time. Source groups are disabled if the PSD Groups option is used. Select **Source Pathway - Source Groups** on the left side of the Source Pathway dialog to display available options.



Source Pathway - Source Groups screen

> At least one **Source Group** must be defined for each run. By default, AERMOD View automatically sets up a source group containing all sources you have defined for the current run. The **ID** for this source group is **ALL**.

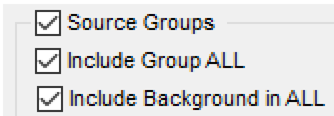
The **Source Groups** screen contains a navigation tree of existing source groups on the left, with a table containing all sources in the project on the right. In this screen, you can:

- Create a source group comprising of single sources
- Create a source group containing a source group range
- Delete a source group

In addition, the screen has the following options:

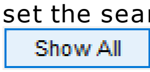
- **Include Group ALL:** Make sure this box is checked so the model can calculate the total contribution for all sources.

- **Include Background in ALL (AERMOD Only):** Check this box if you want to include background concentrations with the Source Group ALL. To use this option, you must have specified background concentrations values and/or hourly file in the Source Pathway - Background Concentration screen.

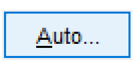


> Background concentrations can also be included with any source group to estimate cumulative ambient impacts (see How to Include Background Concentrations for Source Groups).

> You can only specify one background concentration. If you have more than one source group you cannot have different background concentrations for different source groups. You can either apply background concentration to a source group or not.

- **Search Function:** At the top of the screen is a Search bar, which allows you to search for specific strings to filter the sources. Select either the Search Using All Fields or Search Using Selected Field radio button; the selected field is the field that the sources are being sorted by and is denoted by a triangle beside the field name. Enter the text in the Text to Search box, set the search terms, and click on the  button. View the entire, unfiltered list by clicking 

.

- **Auto:** Click the 

button to access the Auto-Generated Source Groups dialog. In this dialog, you can easily create source groups containing only one source, and perform other features. See Auto-Generated Source Groups for more information.

How to Create a Source Group Containing a Single Source:

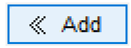
You must first define the sources for the current project before creating any source groups.

1. Check the **Source Groups** box.

2. Click on the **New** button to create a source group. A default source group ID will be displayed; you may change this if you wish. The ID identifies the source group and can be up to 8 alphanumeric characters.

3. From the list of sources on the right, select the sources to add to the source group. You can select multiple sources from the list by pressing down the Shift key while you select the items in sequence or the Ctrl key to make disjoint selections.

4. Once you have selected the sources:

   - with the source group selected, click on the 

button to add the source(s) to the source group.

   or

   - drag the source(s) into the source group

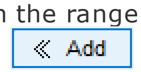
5. The selected sources should then be displayed underneath the source group you created.

You may repeat steps 1 to 4 to create more source groups.

How to Create a Source Group Containing a Source Range:

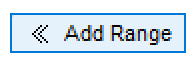
1. Check the **Source Groups** box.

2. Click on the **New** button to create a new source group. A default source group ID will be displayed; you may change this if you wish. The ID identifies the source group and can be up to 8 alphanumeric characters.

3. From the list of sources on the right, select the first source in the range to add to the source group. Drag the source into the source group, or click on the 

button.

4. From the list of sources on the right, select the last source in the range. Then:

   - in the navigation tree, select the first source in the range, and click on the 

button to add the source(s) to the source range.

   or

   - drag the source(s) on top of the first source in the range.

5. The source range will display within the source group you created.

6. Double check ranges to ensure that they are logical by double-clicking the range to open the Source ID dialog.

You may repeat steps 1 to 5 to create more source groups containing source ranges. You can also add additional single sources to the group; these sources cannot be within the source range.

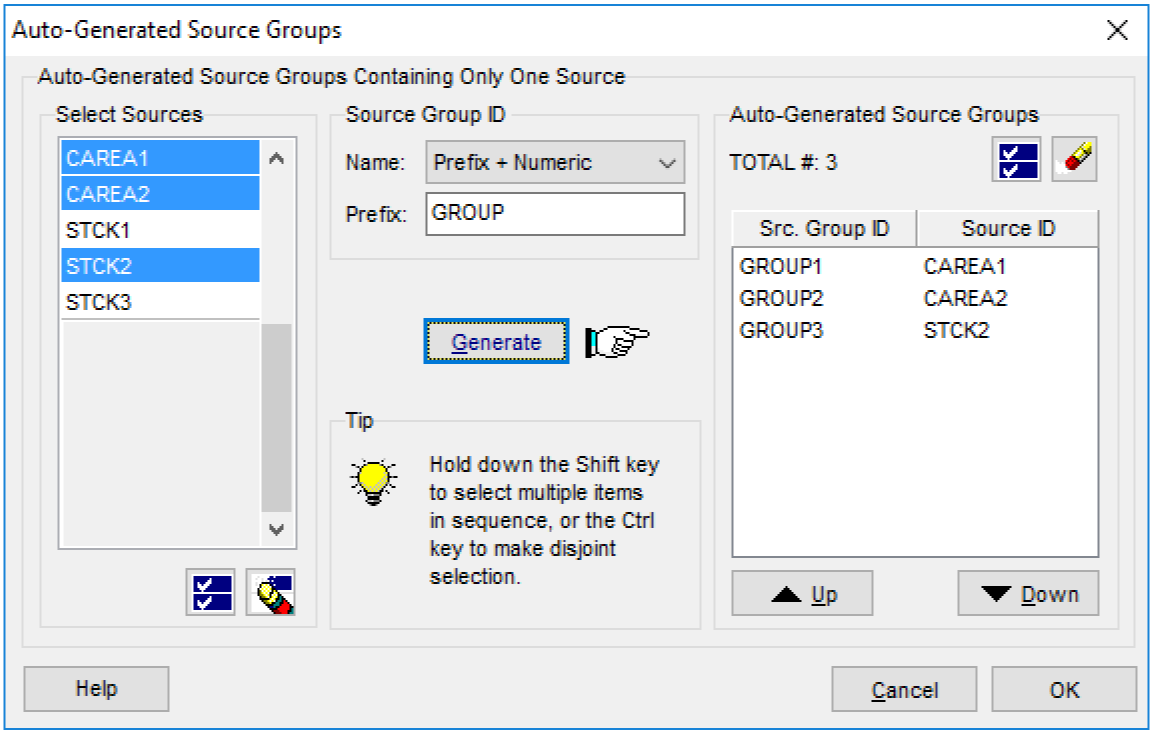
How to Delete a Source Group:

1. In the tree view, select the source group you wish to delete.

2. Click on the button to delete the selected source group or all source groups.

Auto-Generated Source Groups

The **Auto-Generated Source Groups** dialog allows you to create source groups that contain only one source. This is convenient if you have several sources in your project. For example if you have 50 sources and wish to create a source group for each source, instead of repeating the same steps 50 times you can select the sources and auto generate all the source groups in one simple step using this dialog. You have access to this dialog by selecting the button in the Source Groups window.



Auto-Generated Source Groups dialog

The following options are available in this dialog:

- **Select Sources:** This list displays all the sources defined in your current project.

- **Source Group ID:** Here you can select a name convention for the auto-generated source groups. The Name drop-down list contains the following options -

  - **Default:** This is the AERMOD View default name convention for a source group, SRCGP1, SRCGP2,...etc.

  - **Numeric (1,2,3...):** Source groups IDs created using this option will be given a numeric ID starting from 1.

  - **Prefix + Numeric:** This option allows you to specify a prefix to be used together with a numeric portion. The Prefix text box becomes available for you to specify the desired prefix. If you choose, for example, GRP as the prefix then the source group IDs will be named as GRP1, GRP2, GRP3, and so on.

  - **Numeric + Suffix:** This option allows you to specify a suffix to be used together with a numeric portion. The Suffix text box becomes available for you to specify the desired suffix. If you choose, for example _GRP as the suffix, then the source group IDs will be named as 1_GRP, 2_GRP, 3_GRP, and so on.

  - **Same as Source ID:** You can also request that the auto-generated source groups have the same ID as the source it contains. For example, the source group STCK1 contains the source STCK1.

- **Auto-Generated Source Groups:** This panel displays the generated source groups with the total number of source groups is displayed at the top of the panel. In the list the source group ID is displayed along with the source contained in that source group. The following buttons are available in this panel -

 Press this button to select all the source groups in the list.

 Press this button to delete the selected source groups from the list.

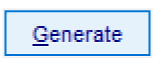
Press this button to move the selected source group one row up.

Press this button to move the selected source group one row down.

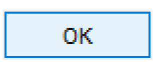
How to Auto-Generated Source Groups:

1. From the Select Sources list, select the sources for which to make source groups. You can also select multiple sources from the list by pressing down the Shift key while you select the items in sequence or the Ctrl key to make disjointed selections. You can also click on the  button to select all the sources in the list or use the  button to clear your selections.

2. From the Source Group ID panel, specify the naming convention to be used for the generated source groups.

3. Click on the 

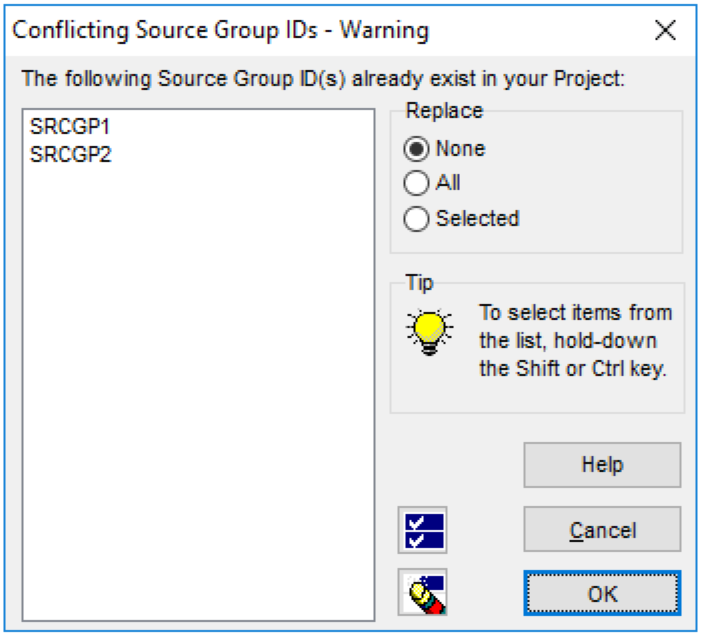
button to generate source groups for the selected sources. The generated source groups will be displayed in the Auto-Generated Source Groups window.

4. Click on the 

button to close the Auto-Generated Source Groups dialog and to transfer these source groups to the main list of source groups located in the Source Groups window. If any of the auto-generated source group IDs already exist in your project, the Conflicting Source Group IDs dialog will be displayed.

Conflicting Source Group IDs

The **Conflicting Source Group IDs** dialog is displayed when there is a conflict between existing source group IDs and source group IDs created by the Auto-Generated Source Groups dialog. A list of the source groups IDs that are conflicting will be displayed.



Conflicting Source Group IDs Warning dialog

The following options are available to resolve the conflict:

- **None:** If this option is selected, then none of the source groups listed in this dialog will be replaced.

- **All:** If this option is selected, then all the source groups listed in this dialog will replace any existing source groups with equal IDs.

- **Selected:** If this option is selected, then only the Source Groups selected from the list will replace any existing Source Groups with equal IDs. To select items from the list, press down the Shift key while you select multiple items in sequence or the Ctrl key to make disjoint selections.

How to Include Background Concentrations for Source Groups

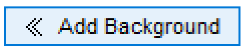
AERMOD Only

Background concentrations can be included with any source group to estimate cumulative ambient impacts. To include background concentrations with a particular source group, the reserved source ID "**BACKGROUND**" can be included for any source group, including source group ALL. The contribution of background concentrations can be tracked separately by including a source group with **BACKGROUND** as the only "source ID."

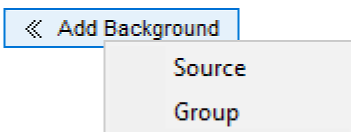
See Also: Source Groups

You can only specify one background concentration. If you have more than one source group you cannot have different background concentrations for different source groups. You can either apply background concentration to a source group or not.

How to Include Background Concentrations for Sources Groups

1. Press the 

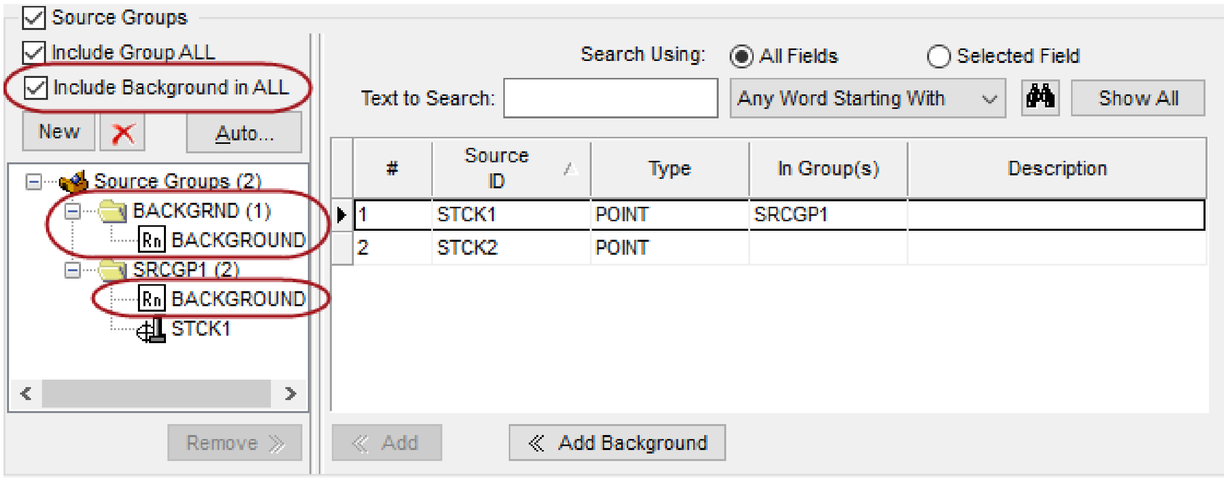
button. The following options are displayed in the pop-up menu:



- **Source:** select this option if you want to include the reserved source ID **BACKGROUND** within an already specified source group. Make sure the source group you want to add background is selected in the tree located on the left-side of the screen (e.g., STCK1 source group in image below).

- **Group:** select this option to create the source group **BACKGRND** containing only the reserved source ID "**BACKGROUND**".

2. To include background concentration in the source group **ALL**, check the box for the **Include Background in ALL.**



How AERMOD Handles Background Concentrations:

Background concentrations specified are combined with source impacts on a temporally-paired basis to estimate cumulative ambient impacts. However, since modeled concentrations are not calculated for hours with calm or missing meteorological data, background concentrations are also omitted for those hours. This may result in the background contribution being lower than expected for short-term averages of 3-hours up to 24- hours for periods when the denominator used to calculate the multi-hour average is adjusted in accordance with EPA's calms policy (see Section 8.3.4.2 of Appendix W), which is implemented within the AERMOD model.

For example, if 12 hours out of a 24-hour period are calm or missing, the calms policy dictates that the 24-hour average concentration would be based on the sum of the 12 non-calm/non-missing hours divided by 18. The contribution from background concentrations would also be based on the sum of background values for the 12 non-calm/non-missing hours, divided by 18. If background was specified as uniform during that 24-hour period, then the contribution from background would appear to be 33.3% lower than expected (i.e., 12/18).

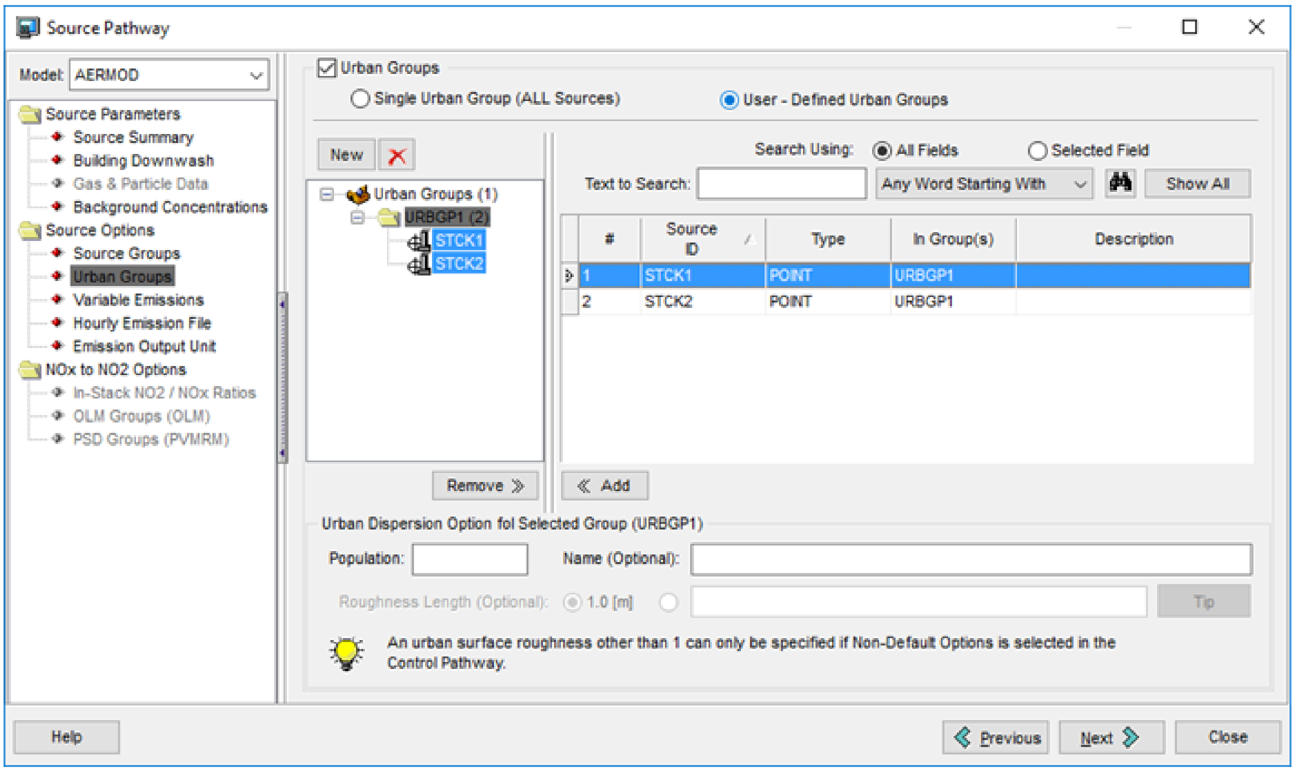
Urban Groups

AERMOD Only

In the **Urban Groups** page, you can specify multiple urban areas within a single model run by creating urban groups. Urban groups may be used, for example, to model large domains that encompass more than one identifiable urban area, where the separation is large enough to warrant separate treatment of the urban boundary layer effects. Select **Source Pathway - Urban Groups** from the tree located on the left side of the Source Pathway dialog, to display the available options.

Beginning with US EPA AERMOD version 12060, users can create more than one urban group. Sources can only exist in one urban group at a time.

The **Single Urban Group (ALL Sources)** option is **not available** in US EPA AERMOD version 09292, 11103, or 11353.



Source Pathway dialog - Urban Groups

The **Urban Groups** screen contains a navigation tree of existing urban groups on the left, with a table containing all sources in the project on the right. In this screen, you can:

- Create a Single Urban Group comprising of ALL sources by selecting the **Single Urban Group (ALL Sources)** option
- Create Multiple Urban Groups by selecting the **User-Defined Urban Groups** option

In addition, the screen also contains a search function at the top of the screen, which allows you to search for specific strings to filter the sources. Select either the **Search using All Fields** or **Search Using Selected Field** radio button; the selected field is the field that the sources are being sorted by and is denoted by a triangle beside the field name. Enter the text in the **Text to Search** box, set the search terms, and click on the button. View the entire, unfiltered list by clicking Show All.

See Also: Urban and Rural Dispersion Coefficients

How to Create an Urban Group Containing a Single Source:

You must define the sources for the current run before creating any urban groups.

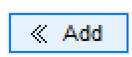
1. Click to place a checkmark in Urban Groups. Click on the 

button to create a new urban group. A default Urban Group ID will be displayed; you may change this if you wish. The Urban Group ID identifies the urban group and can be up to 8 alphanumeric characters.

When you place a checkmark in **Urban Groups**, the **Dispersion Coefficient** setting in the Pollutant/Averaging screen is set to **Urban**. If the **Urban Groups** is unchecked the **Dispersion Coefficient** setting in the Pollutant/Averaging screen is set to **Rural**. The Urban option allows you to incorporate the effects of increased surface heating from an urban area on pollutant dispersion under stable atmospheric conditions.

2. From the list of sources on the right, select the sources to add to the urban group. You can select multiple sources from the list by pressing down the Shift key while you select the items in sequence or the Ctrl key to make disjoint selections.

3. Once you have selected the sources:

   - with the urban group selected, click on the 

button to add the source(s) to the urban group.

   or

   - drag the source(s) into the urban group

4. The selected sources should then be displayed underneath the urban group you created.

5. Set the urban dispersion options for each urban group:

   - **Population:** Specify the population of the urban area.
   - **Name (Optional):** Enter the name of the urban area if you wish.
   - **Roughness Length (Optional):** A default value of 1.0m is used for the urban roughness length. If you wish to enter a user-specified value you will need to select the Non-Default

radio button option. You will then be allowed to enter a user-specified value or click on the Tip button to display the Surface Roughness Length dialog which can help you select a value to use.

6. You may repeat steps 1 to 3 to create more urban groups.

> You can only specify up to the maximum number of urban groups allowed for each model. Please see the EPA Models/Limits pages for each model in the Preferences dialog to determine the number of urban groups allowed

How to Delete a Source Group:

1. In the navigation tree, select the urban group you wish to delete.

2. Click on the button to delete the urban group.

Urban and Rural Dispersion Coefficients

The classification of a site as urban or rural, and thus the selection of either urban or rural dispersion coefficients should be based upon either the **Land Use Procedure** or **Population Density Procedure**:

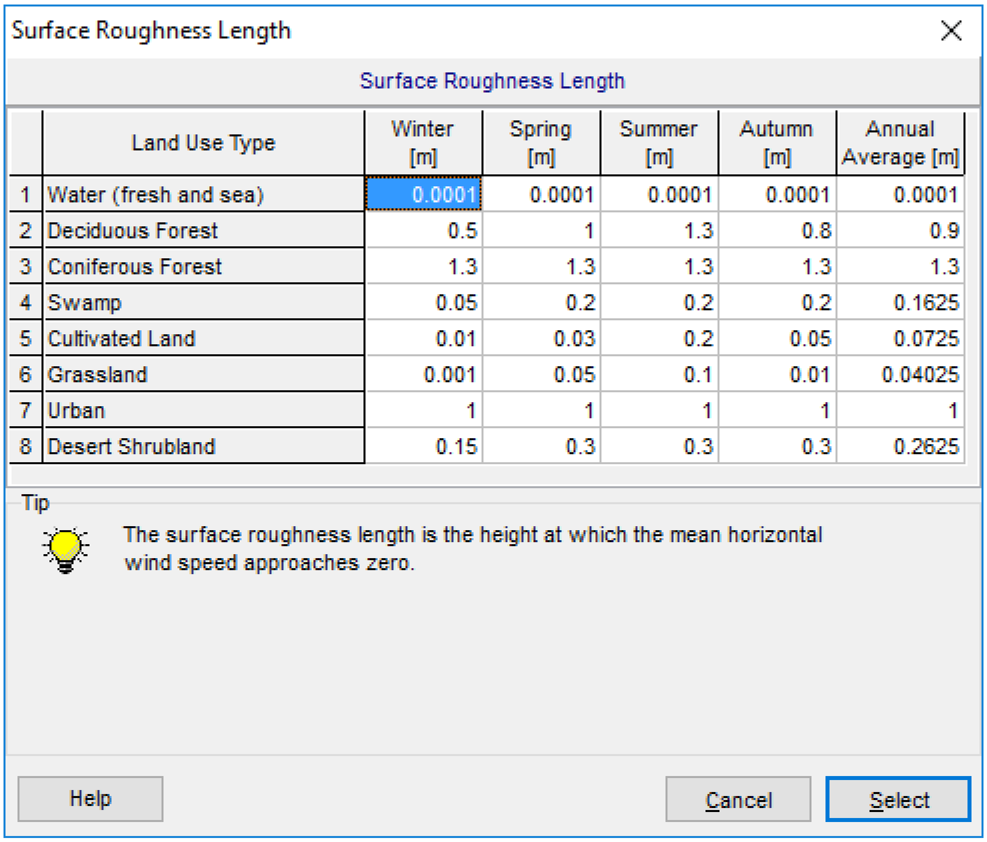
- **Land Use Procedure:** Circumscribe a 3 km radius circle, A_o, about the source using the meteorological land use typing scheme -
  - If land use types I1, I2, C1, R2, and R3 account for 50% or more of A_o, select the **Urban** option,
  - Otherwise use the **Rural** option.
- **Population Density Procedure:** Compute the average population density, ***p***, per square kilometer with A_o as defined above -
  - If ***p*** > 750 people/km2, select the **Urban** option,
  - If ***p*** <= 750 people/km2, select the **Rural** option.

Of the two methods above, the **Land Use Procedure** is considered a more definitive criterion. The **Population Density Procedure** should be used with caution and should not be applied to highly industrialized areas where the population density may be low and thus a rural classification would be indicated, but the area is sufficiently built-up so that the urban land use criteria would be satisfied. In this case, the classification should already be urban and urban dispersion parameters should be used.

Surface Roughness Length

The surface roughness length is the height at which the mean horizontal wind speed approaches zero and is related to the roughness characteristics of the terrain. It is not equal to the physical dimensions of the obstacles to the wind flow, but is generally proportional to them.

The **Surface Roughness Length** dialog provides you with empirically determined surface roughness length values (from Sheih et al., 1979) for various land use types for each season. Select an appropriate input value from this dialog.



Surface Roughness Length dialog

Variable Emissions

In the **Variable Emissions** screen, you can specify emission rate factors for individual sources or for groups of sources. Select Source **Pathway - Variable Emissions** from the tree on the left side of the **Source Pathway** dialog to display these options.

The emissions may vary by the following scenario types:

- by Season
- by Month

- by Hour-of-Day
- by Wind Speed
- by Season / Hour
- by Season / Hour / Day
- by Season / Hour / Seven Days
- by Month / Hour / Day
- by Month / Hour / Seven Days
- by Hour-of-Day / Day of Week
- by Hour-of-Day / Seven Days

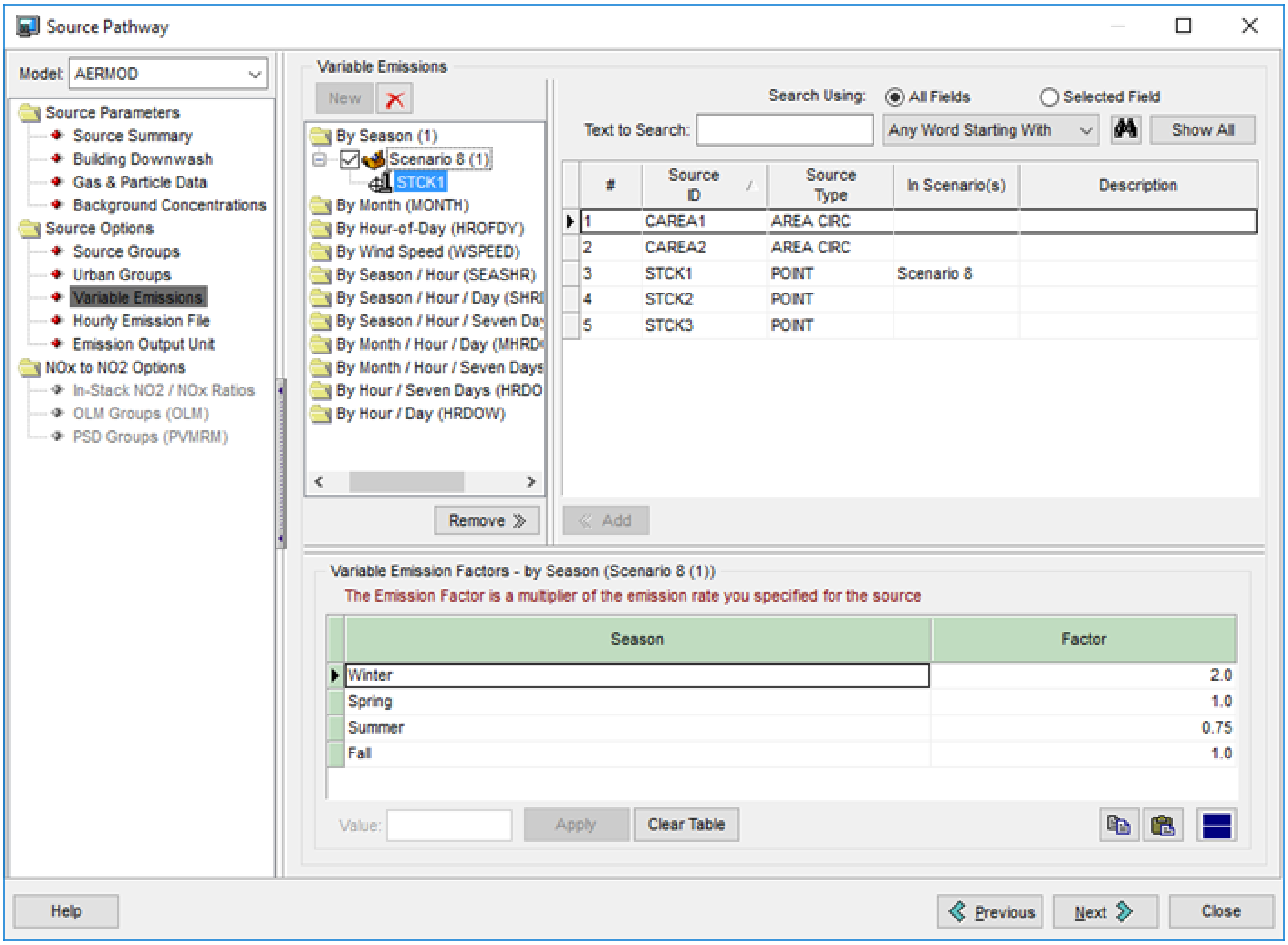
Variable emission rate factors are a multiplier of the emission rate you have specified for the source in the Source Inputs window.

For Example:

- A factor of 0 means the source is not emitting for that particular period.
- A factor of 0.5 means that the source is emitting 50% of the specified emission rate.
- A factor of 1 means that the source is emitting 100% of the specified emission rate.

After scenarios are created, single sources and source ranges can then be associated with a single scenario.

> A particular source can only be associated with one scenario.



Source Pathway dialog - Variable Emissions

The **Variable Emissions** screen contains a navigation tree with available scenario types on the upper left, with a table containing all sources in the project on the upper right. Along the bottom is a panel that displays the configuration details of a given scenario when one scenario is selected. In this screen, you can:

- create, assign emission rate factors to, and assign sources to a scenario
- delete a scenario

In addition, the screen also contains a search function at the top of the screen, which allows you to search for specific strings to filter the sources. Select either the **Search using All Fields** or **Search using Selected Field** radio button; the selected field is the field that the sources are being sorted by and is denoted by a triangle beside the field name. Enter the text in the **Text to Search** box, set the search terms, and click on the  button. View the entire, unfiltered list by clicking Show All.

How to Assign Emission Rate Factors to Sources:

You must first define the sources for the current project before creating any variable emission scenarios.

1. Select the scenario type you wish to add a scenario to. Click on the **New** button to create a new scenario. A default Scenario ID will be displayed; you may change this if you wish. The ID can be up to 8 alphanumeric characters.

2. On the table displayed on the bottom part of the screen, assign the emission rate factors for the scenario in each cell.

    - to select rows in sequence, hold down the Shift key while selecting the first and last items in the sequence

    - to select disjointed rows hold down the Ctrl key while selecting multiple items

3. After making the desired selections, enter the Emission Factor in the text box and click the Apply button.

4. The number and type of factors to be assigned will depend on the type of scenario:

Scenario Type	Keyword	# of Factors	Specifications
By Season	SEASON	4	For each season, (Winter, Fall, Summer, Spring) you must input the variable emission factor. The seasons are defined as: **WINTER**: December, January, February **SPRING**: March, April, May **SUMMER**: June, July, August **FALL**: September, October, November
By Month	MONTH	12	Specify the variable emission factor for each month of the year (January-February).
By Hour-of-Day	HROFDY	24	Specify a variable emission factor for each hour of the day.
By Wind Speed	WSPEED	6	Here you must specify a variable emission factor for each wind speed category.
By Season / Hour	SEASHR	96	Specify a variable emission factor for each hour of the day, for each season.

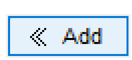
Scenario Type	Keyword	# of Factors	Specifications
By Season / Hour / Day	SHRDOW	288	Specify a variable emission factor for each hour of the day for Weekdays, Saturdays, and Sundays for each season. Move between tabs to input the data for each season.
By Season / Hour / Seven Days	SHRDOW7	672	Specify a variable emission factor for each hour of the day, for each day of the week for each season. Move between tabs to input the data for each season.
By Month / Hour / Day	MHRDOW	864	Specify a variable emission factor for each hour of the day for Weekdays, Saturdays, and Sundays for each month. Move between tabs to input the data for each month.
By Month / Hour / Seven Days	MHRDOW7	2016	Specify a variable emission factor for each hour of the day, for each day of the week for each month. Move between tabs to input the data for each month.
By Hour-of-Day / Day of Week	HRDOW	72	Specify a variable emission factor for each hour of the day for Weekdays, Saturdays, and Sundays.
By Hour-of-Day/ Seven Days	HRDOW7	168	Specify a variable emission factor for each hour of the day, for each day of the week.

For hourly emissions, the hour displayed is for the ending of the hour period. For example, the 8 am hour row will be for hour ending at 8 am (7:00:01 am to 8:00:00 am).

To Add Single Sources:

1. From the list of sources on the right, select the sources to add to the scenario. You can select multiple sources from the list by pressing down the Shift key while you select the items in sequence or the Ctrl key to make disjointed selections.

2. Once you have selected the sources

   - with the scenario selected, click on the 

 button to add the source(s) to the scenario.

   or

- drag the source(s) into the scenario

3. The selected sources should then be displayed underneath the scenario you created.

To Add Source Ranges:

1. From the list of sources on the right, select the first source in the range to add to the scenario. Drag the source into the scenario, or click on the 

button.

    - in the navigation tree, select the first source in the range, and click on the 

button to add the source(s) to the scenario.

    or

    - drag the source(s) on top of the first source in the range.

2. From the list of sources on the right, select the last source in the range. The source range will display within the scenario you created.

3. You may repeat steps 1 to 2 to create more scenarios.

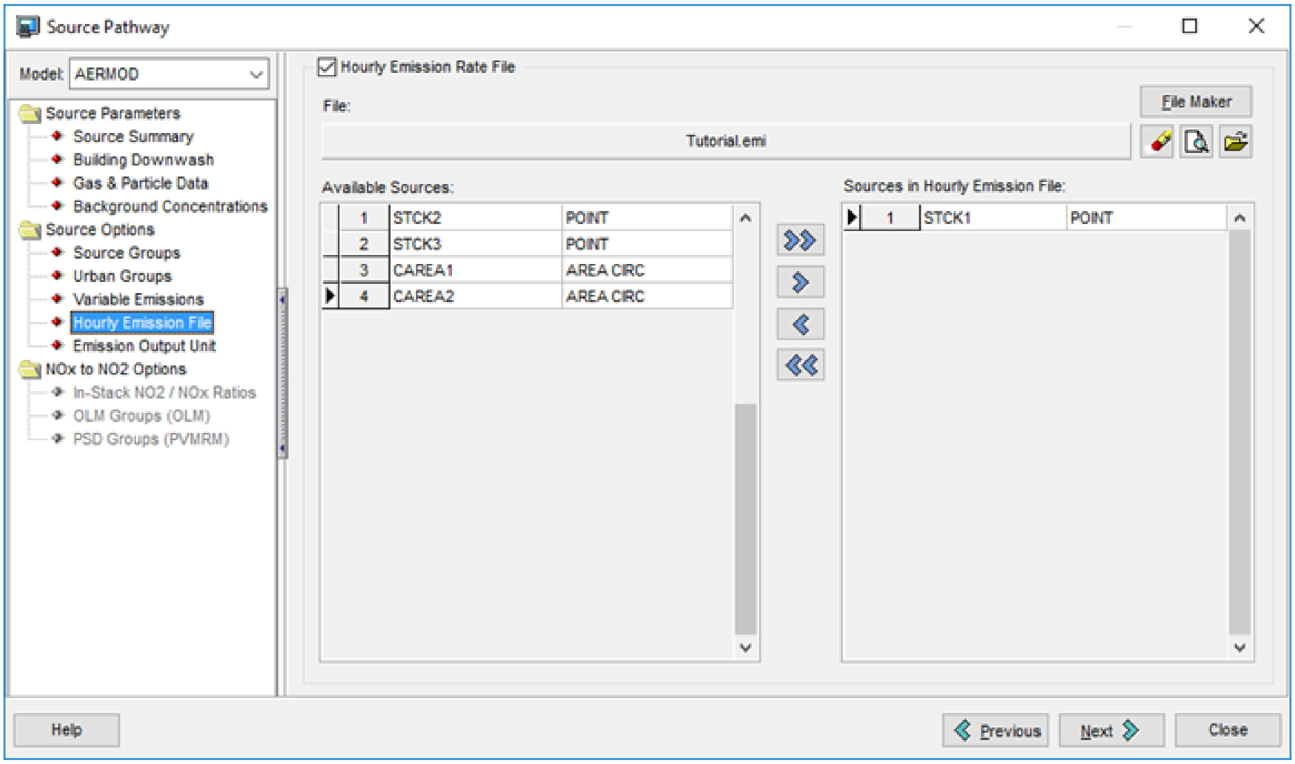
How to Delete a Scenario:

1. In the navigation tree, select the scenario you wish to delete.

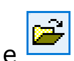
2. Click on the  button to delete the scenario.

Hourly Emission File

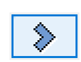
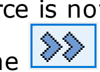
In the **Hourly Emission File** page, you can specify hourly emission rates for one or more sources. Select **Source Pathway - Hourly Emission File** from the tree located on the left side of the Source Pathway dialog, to display the available options.

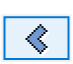


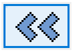
Source Pathway dialog - Hourly Emission File

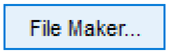
Check the **Hourly Emission Rate File** box to make this option available. Hourly emission rates must be provided in a separate file and you can provide only one hourly emission file per run. Click the  button to specify the name and location of the hourly emission rate file to be used. Click the  button to preview the selected file in a text editor.

See Also: Hourly Emission Rate File Format.

Select the sources to be modeled from the **Single Sources** dialog and click on the  button. The selected sources will be displayed in the **Sources in the Hourly File** window. If a source is not included in the list, the hourly emission rate file values will not be applied to it. Click the  button to add all available sources to the hourly emissions file.

 Click this button to remove a source from the **Sources in the Hourly File** window.

 Click this button to clear all the sources from the **Sources in the Hourly File** window.

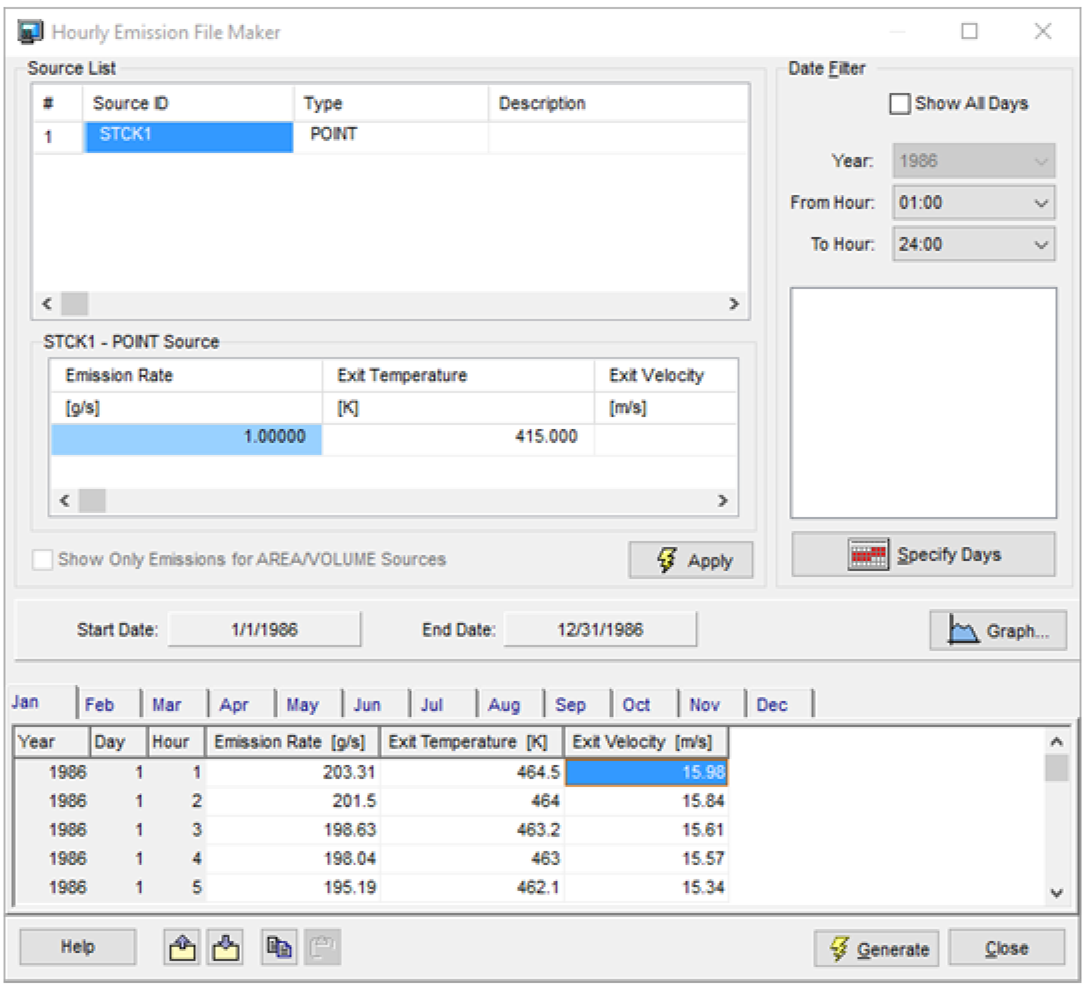


Click this button to launch the [Hourly Emissions File Maker] utility.

> Currently the **Hourly Emission File Maker** does not support **Line Area Sources** or **Line Volume Sources**.

Hourly Emission File Maker

The **Hourly Emission File Maker** allows you to create a file that contains hourly emissions data for every source you have selected. You can access this utility by clicking the **File Maker** button in the [Hourly Emission File] page of the [Source Pathway].



Hourly Emission File Maker utility

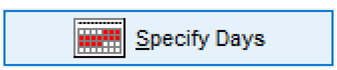
Source List

Select the source for which you wish to create hourly emissions data from the list of the available sources.

> Currently the **Hourly Emission File Maker** does not support **Line Area Sources** or **Line Volume Sources**.

Date Filter

Use the available options to narrow the range of hours for which you wish to define emissions.

- **Show All Days:** Check this box to display all available days and hours in the bottom grid.
- **Year:** If the met file contains data for more than one year, select the year you wish to work with.
- **From Hour/To Hour:** Use these settings to narrow the range of hours for each day displayed (e.g. if emissions varied for a certain time period during the day).
- Click 

button to select a specific range of dates to be displayed.

Source Emission Data

The content of this section varies depending on the source selected in the **Source List**.

Use the options in **Date Filter** to display the range of dates and hours for which you wish to define emissions different from those defined in Source Inputs.

Define the values for the parameters you wish to specify for the selected hours, then click **Apply**.

If you wish to define these values for individual hours, you can do so directly in the grid.

Show Only Emissions for AREA/VOLUME Sources

When this box is checked, only the **Emission Rate** column is visible in the table.

When this box is unchecked, **Release Height** and **Initial Vertical Dimension** parameters are available for the area sources and **Release Height**, **Initial Lateral Dimension**, and **Initial Vertical Dimension** parameters are available for the volume sources.

Met Data Range

This section displays the start and end date for the data in the met file associated with the project. The **Hourly Emission File Maker** will create a file that corresponds exactly to the data range in the provided met file.

Click the 

button to load the Graph dialog, to view the variable emission data you have specified.

Hourly Emission Table

This table allows you to specify the emission values for any hour in the data period. The table is originally populated by the data in the existing .EMI file specified in the Source Pathway | Hourly Emission File. If no file is specified, a value of "0" will be listed for all hours/parameters.

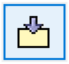
Jan | Feb | Mar | Apr | May | Jun | Jul | Aug | Sep | Oct | Nov | Dec

Year	Day	Hour	Emission Rate [g/s]	Exit Temperature [K]	Exit Velocity [m/s]
1986	1	1	203.31	464.5	15.98
1986	1	2	201.5	464	15.84
1986	1	3	198.63	463.2	15.61
1986	1	4	198.04	463	15.57
1986	1	5	195.19	462.1	15.34
1986	1	6	197.59	462.8	15.53
1986	1	7	201.77	464.1	15.86

Following options are available in the **Hourly Emission File Maker** dialog:

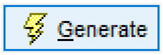
 - Click this button to export hourly emission data to MS Excel.

A separate file will be generated for each source.

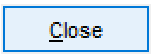
 - Click this button to import hourly emission data from MS Excel.

 - Click this button to copy data highlighted in the hourly emission table. If no data is highlighted, all data currently displayed will be copied.

 - Click this button to paste data.



- Click this button to generate the hourly emission file to be used in Source Pathway | Hourly Emission File. If a file already exists, you can select of overwrite the existing information with updated changes.



- Click this button to close the dialog. You will be have the opportunity to save the latest changes.

Hourly Emissions Rate File Format

You can create the Hourly Emission Rate File using the Hourly Emission File Maker.

Each record or line of the hourly emission rate file must include the following parameters in the order given below:

- SO HOUREMIS
- Year
- Month
- Day
- Hour
- Source ID
- Emission Rate (g/s or user units)

For **Point Sources** add the following parameters:

- Stack Gas Exit Temperature (K)
- Stack Gas Exit Velocity (m/s)

For **Area Sources** add the following parameters (optional):

- Variable Release Height (m)
- Initial Vertical Dimension (m)

For **Volume Sources** add the following parameters (optional):

- Variable Release Height (m)
- Initial Lateral Dimension (m)
- Initial Vertical Dimension (m)

For **Open Pit Source** add the following parameters:

- No additional parameters required

See below for an example of the hourly emission rate file for a single point source:

```
SO HOUREMIS 1986  1  1  1 STCK1              1.00000  415.000   11.000
SO HOUREMIS 1986  1  1  2 STCK1              0.90000  415.000   11.000
SO HOUREMIS 1986  1  1  3 STCK1              0.90000  415.000   11.000
SO HOUREMIS 1986  1  1  4 STCK1              1.00000  415.000   11.000
SO HOUREMIS 1986  1  1  5 STCK1              1.10000  415.000   11.000
SO HOUREMIS 1986  1  1  6 STCK1              1.20000  415.000   11.000
SO HOUREMIS 1986  1  1  7 STCK1              1.20000  415.000   11.000
SO HOUREMIS 1986  1  1  8 STCK1              1.00000  415.000   11.000
```

See below for an example of the hourly emission rate file for area and volume sources:

```
SO HOUREMIS 1986  1  1  1 PAREA1             2.00000    2.000   12.000
SO HOUREMIS 1986  1  1  1 VOL1             400.00000    3.000    3.000     3.000
SO HOUREMIS 1986  1  1  2 PAREA1             2.00000    2.000   12.000
SO HOUREMIS 1986  1  1  2 VOL1             400.00000    3.000    3.000     3.000
SO HOUREMIS 1986  1  1  3 PAREA1             2.00000    2.000   12.000
SO HOUREMIS 1986  1  1  3 VOL1             400.00000    3.000    3.000     3.000
SO HOUREMIS 1986  1  1  4 PAREA1             2.00000    2.000   12.000
SO HOUREMIS 1986  1  1  4 VOL1             400.00000    3.000    3.000     3.000
SO HOUREMIS 1986  1  1  5 PAREA1             2.00000    2.000   12.000
```

> The above example also illustrates how multiple sources are organized in the hourly emission rate file.

See below for an example of the hourly emission rate file for the open pit source:

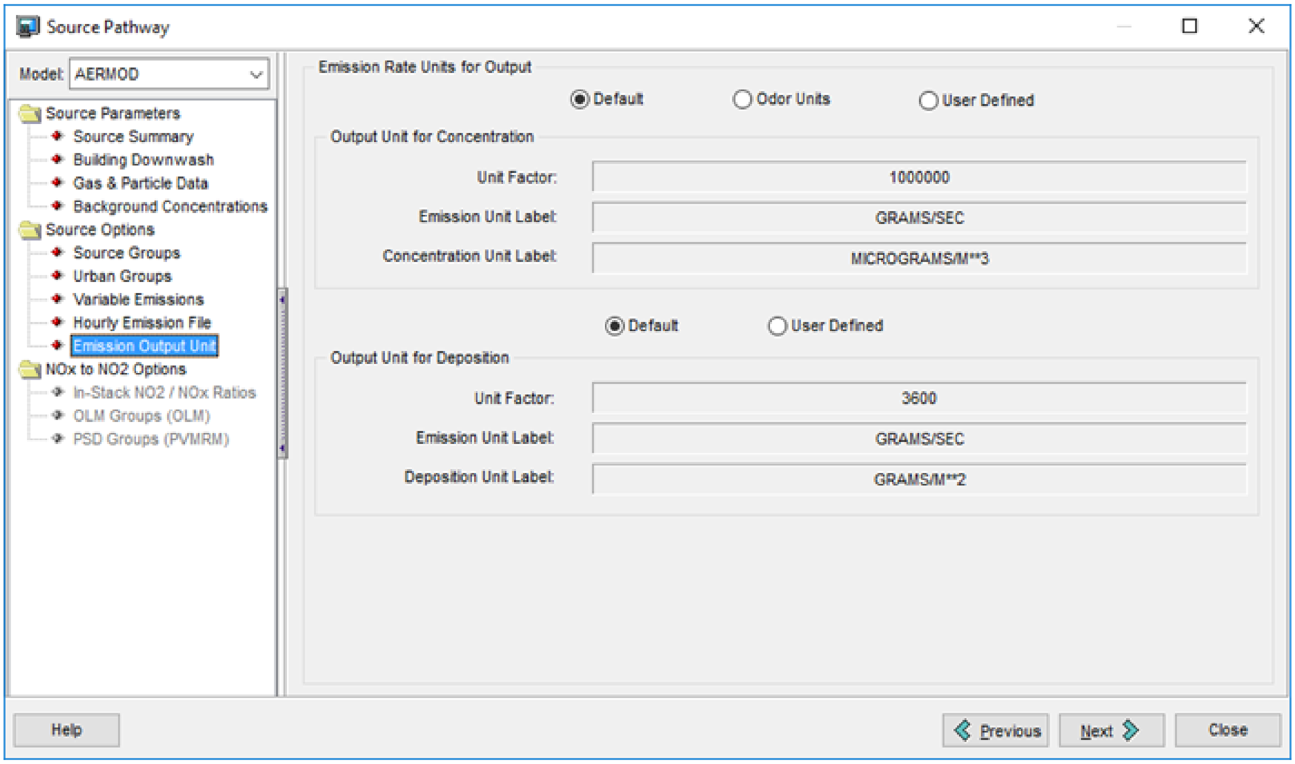
```
SO HOUREMIS 1986  1  1  1 OPIT1              1.90000
SO HOUREMIS 1986  1  1  2 OPIT1              1.90000
SO HOUREMIS 1986  1  1  3 OPIT1              2.00000
SO HOUREMIS 1986  1  1  4 OPIT1              1.80000
SO HOUREMIS 1986  1  1  5 OPIT1              2.00000
SO HOUREMIS 1986  1  1  6 OPIT1              2.10000
SO HOUREMIS 1986  1  1  7 OPIT1              2.30000
SO HOUREMIS 1986  1  1  8 OPIT1              2.00000
```

It is not necessary to process the entire hourly emissions file on each model run. The correct emissions data will be read if the **Data Period** or the **Days Range** option in the **Data Period** display are used, as long as all the dates (including those that are processed and those that are skipped) match the meteorological data files.

The model will use the **Stack Release Height** and the **Stack Inside Diameter** defined on the Source Inputs window, but will use the **Emission Rate**, **Stack Gas Exit Temperature** and **Stack Gas Exit Velocity** from the hourly emission file. If the **Emission Rate**, the **Stack Gas Exit Temperature**, and the **Stack Gas Exit Velocity** are not included for a particular hour (any or all of these fields are blank), the model will interpret emissions data for that hour as missing and will set the parameters to zero. Since the emission rate will be zero, there will be no calculations made for that hour and that source.

Emission Output Unit

The **Emission Output Unit** dialog allows you to specify different output units for concentration, deposition and odor calculations. The U.S. EPA models ISCST3, AERMOD, and ISC-PRIME use default output units of micrograms per cubic meter for concentration calculations and grams per square meter for deposition calculations. Select **Source Pathway - Emission Output Unit** from the tree located on the left side of the Source Pathway dialog, to display the available options.



Source Pathway dialog - Emission Output Unit

Default

If you do not specify user defined output units, AERMOD View will use the following default output units for concentration and deposition calculations -

Source Type	Input Unit	Output Unit Concentration	Output Unit Deposition
POINT and VOLUME	Grams/sec	Micrograms/m^3	Grams/m^2
AREA and OPEN PIT	Grams/sec/m^2	Micrograms/m^3	Grams/m^2

- **Concentration (Micrograms/m³):** The unit factor of 1000000 converts grams to micrograms.
- **Deposition (Grams/m²):** The unit factor of 3600 essentially converts grams/sec to grams/hour.

Odor Units

Select the **Odor Units** option if you wish to specify any odor units. The following must be specified for the **Odor Units** option:

- **Unit Factor:** This is the unit factor used to convert the emission rate input units to the odor output units. The emission rate unit factor is set to be 1 and it applies to all sources for a given run.
- **Emission Unit Label:** The emission rate unit is set to be OU/s.
- **Concentration Unit Label:** The concentration unit is set to be OU/m^3.

User-Defined

Select the **User Defined** option if you wish to specify output units other than the default for **Concentration** and/or **Deposition** calculations. The following must be specified for the user defined output unit option:

- **Unit Factor:** This is the unit factor used to convert the emission rate input units to the output units. The emission rate unit factor applies to all sources for a given run.
- **Emission Unit Label:** Specify here the emission rate unit label, up to 40 characters in length with no blank spaces.
- **Concentration/Deposition Unit Label:** This is the output unit label for concentration or deposition calculations. No blank spaces are allowed in this field.

In-Stack NO2/NOx Ratios

AERMOD Only

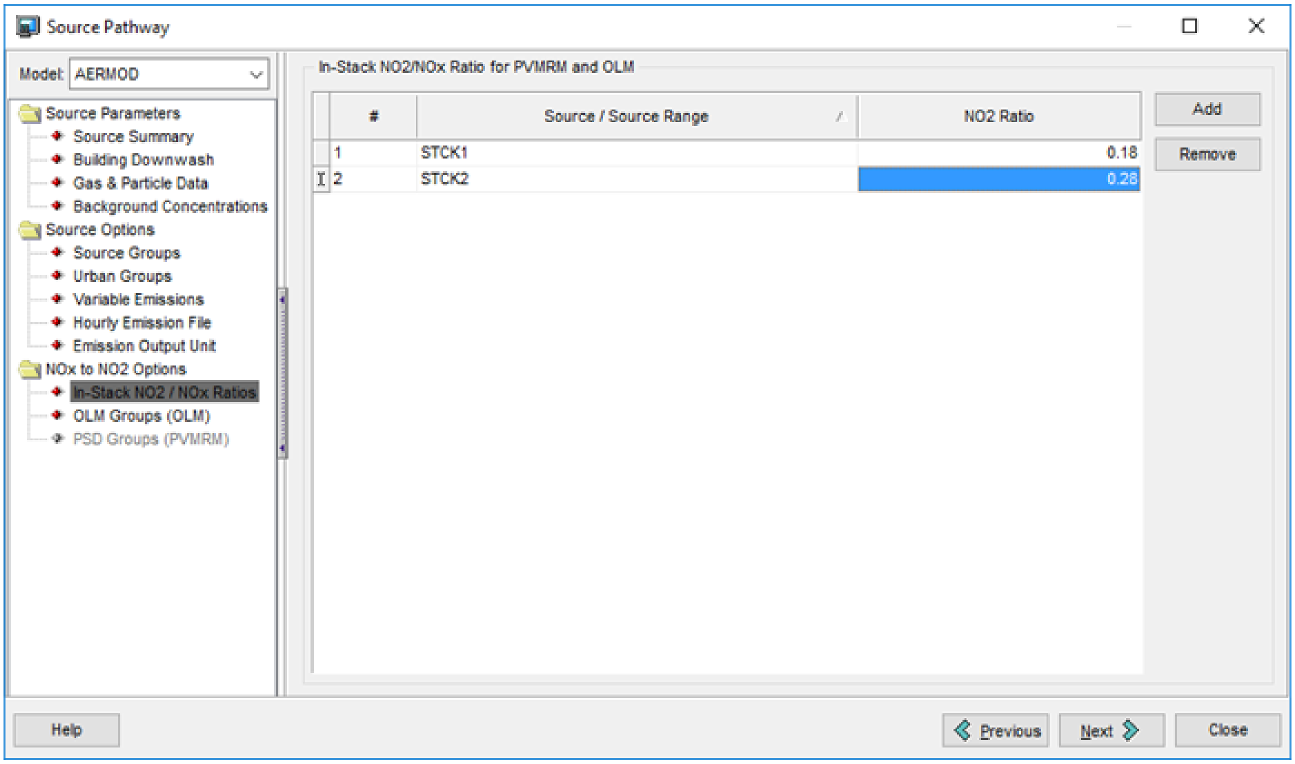
The **NO2 Ratio for PVMRM and OLM** screen allows you to specify the NO2 Ratio for single sources or source ranges; this ratio is used when modeling NO2 conversions. Select **Source Pathway - In-Stack NO2/NOx Ratios** from the tree located on the left side of the Source Pathway dialog, to display the available options.

The **NO2 Ratio for PVMRM and OLM** screen is unavailable until you take the following steps:

1. Enable **Modeling conversion of NOx to NO2** in the Dispersion Options screen of the Control Pathway.
2. Select any of the **Tier 3** options (**OLM**, **PVMRM**, **PVMRM2**) in the NOx to NO2 Options screen of the Control Pathway.

The PVMRM and OLM options for modeling NO2 conversion assume a default in-stack NO2/NOx ratio of 0.10 (10%).

> If the NO2 Ratio is omitted for any source, the value specified in the Control Pathway – NOx to NO2 Options for the option NO2/NOx Ratio will be used by the model.



Source Pathway dialog - N02 Ratios

The **NO2 Ratio for PVMRM and OLM** screen contains a table in which you can define the following:

- the **NO2 Ratio** for any particular source
- the **NO2 Ratio** for a source range

> You can only specify one ratio per source. If you duplicate a source, a message will display in the Details dialog.

How to Define the NO2 Ratio for a Particular Source or Source Range:

You must define the sources for the current run before using this option.

1. Click on the Add button to create a new entry in the table.

2. Click in the **Source |Source Range** cell for the new entry. The field becomes editable; click the ... button to open the Source ID (Range) dialog. From the From/To drop-down lists, select a source. Close the Source ID (Range) dialog.

3. In the **NO2 Ratio** cell, enter the NO2 ratio for the source or source range you just specified.

4. You may repeat steps 1 to 3 to define additional entries in the table.

How to Delete a Source Range:

1. Select the entry/row you wish to delete.

2. Click on the Remove button to delete the entry. Confirm the deletion.

OLM Groups

In the **OLM Source Groups** screen, you can analyze N02 conversion using the option to model sources as combined plumes by creating OLM source groups. For a technical explanation of the OLM Source Group option, refer to Specifying Combined Plumes for OLM in the Addendum to the User's Guide for the AERMOD Model (EPA, 2017) available in the US EPA SCRAM web site: http://www.epa.gov/scram001/dispersion_prefrec.htm#aermod.

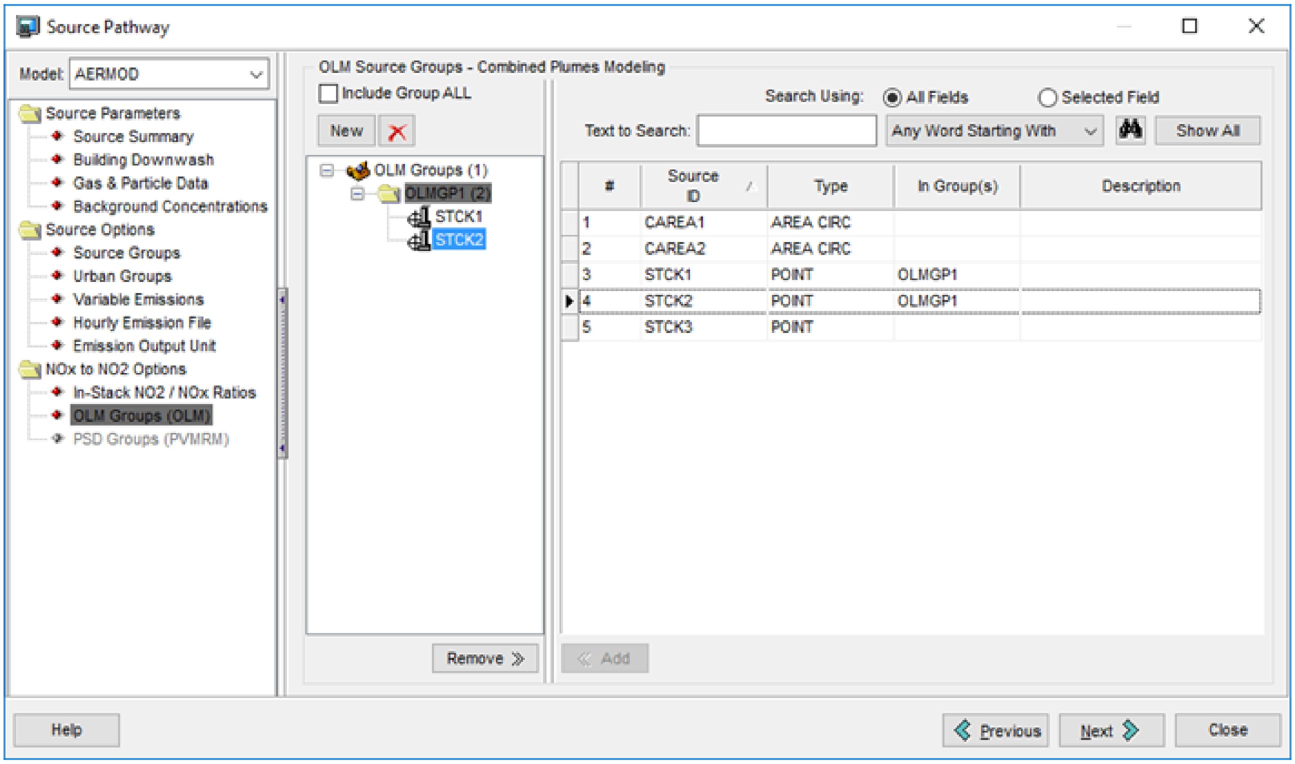
OLM Source Groups are unavailable until you take the following steps:

1. Enable **Modeling conversion of NOx to NO2** in the Dispersion Options screen of the Control Pathway.

2. Select the **OLM (Ozone Limiting Method)** option in the NOx to NO2 screen of the Control Pathway.

3. Select **Source Pathway - OLM Groups (OLM)** from the tree located on the left side of the Source Pathway dialog, to display the available options.

> Sources can only exist in one OLM source group at a time. If you place a source in more than one group, the data in its **In Group(s)** column of the table will display in red.



Source Pathway dialog - OLM Groups

As with source groups, you can specify the source group ALL, which applies OLM on a combined plume basis to all sources. The results will only be output for an OLM source group if the same group of sources is also identified in Source Groups.

The OLM Source Groups screen contains a navigation tree of existing OLM source groups on the left, with a table containing all sources in the project on the right. In this screen, you can:

- create an OLM source group comprising single sources
- create an OLM source group containing source ranges
- delete an OLM source group

You can use the **Include Group ALL** features to include the OLM Group ALL in you project, however this will disable the ability to create new OLM source groups.

In addition, the screen also contains a search function at the top of the screen, which allows you to search for specific strings to filter the sources. Select either the **Search using All Fields** or **Search using Selected Field** radio button; the selected field is the field that the sources are being sorted by and is denoted by a triangle beside the field name. Enter the text in the **Text to Search** box, set the search terms, and click on the button. View the entire, unfiltered list by clicking Show All.

How to Create an OLM Source Group Containing Single Sources:

> You must define the sources for the current run before creating any OLM source group.

1. Click on the New button to create a new OLM source group. A default OLM Source Group ID will be displayed; you may change this if you wish. The OLM Source Group ID identifies the OLM source group and can be up to 8 alphanumeric characters.

2. From the list of sources on the right, select the sources to add to the OLM source group. You can select multiple sources from the list by pressing down the Shift key while you select the items in sequence or the Ctrl key to make disjoint selections.

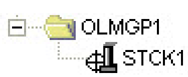
3. Once you have selected the sources:

   - with the OLM source group selected, click on the « Add button to add the source(s) to the OLM source group.

   or

   - drag the source(s) into the OLM source group

4. The selected sources should then be displayed underneath the OLM source group you created.



5. You may repeat steps 1 to 3 to create more source groups.

How to Create an OLM Source Group Containing a Source Range:

1. Click on the New button to create a new OLM source group. A default Source Group ID will be displayed; you may change this if you wish. The Source Group ID identifies the OLM source group and can be up to 8 alphanumeric characters.

2. From the list of sources on the right, select the first source in the range to add to the OLM source group. Drag the source into the OLM source group, or click on the « Add button.

3. From the list of sources on the right, select the last source in the range. Then:

- in the navigation tree, select the first source in the range, and click on the  button to add the source(s) to the OLM source range.

or

- drag the source(s) on top of the first source in the range.

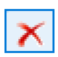
4. The source range will display within the OLM source group you created.

5. Double check ranges to ensure that they are logical by double-clicking the range to open the Source ID dialog.

6. You may repeat steps 1 to 4 to create more OLM source groups containing source ranges. You can also add additional single sources to the group; these sources cannot be within the source range.

How to Delete an OLM Source Group:

1. In the navigation tree, select the OLM source group you wish to delete.

2. Click on the  button to delete the OLM source group.

PSD Groups

> The functionality of PSD source groups is a US EPA BETA-test option in the AERMOD model. Consult with your reviewing authority for further guidance when modeling increment credits for NO2.

In the **PSD Source Groups** screen, you can specify the sources to include when calculating increment consumption when modeling PSD (Prevention of Significant Deterioration) increment credits for NO2. This allows you to properly account for NOx conversion chemistry under the PVMRM option. For a technical explanation of the PSD Source Group option, refer to Modeling N02 Increment Credits with PVMRM in the Addendum to the User's Guide for the AERMOD Model (EPA, 2017) available in the US EPA SCRAM web site: http://www.epa.gov/scram001/dispersion_prefrec.htm#aermod.

PSD source groups are fixed; you cannot add or delete PSD source groups, only add sources to the groups. The three PSD source groups are:

- INCRCONS - increment-consuming sources; these can be new sources or modifications to existing sources

- NONRBASE - existing, non-retired baseline sources
- RETRBASE - retired (increment-expanding or PSD credit) baseline sources

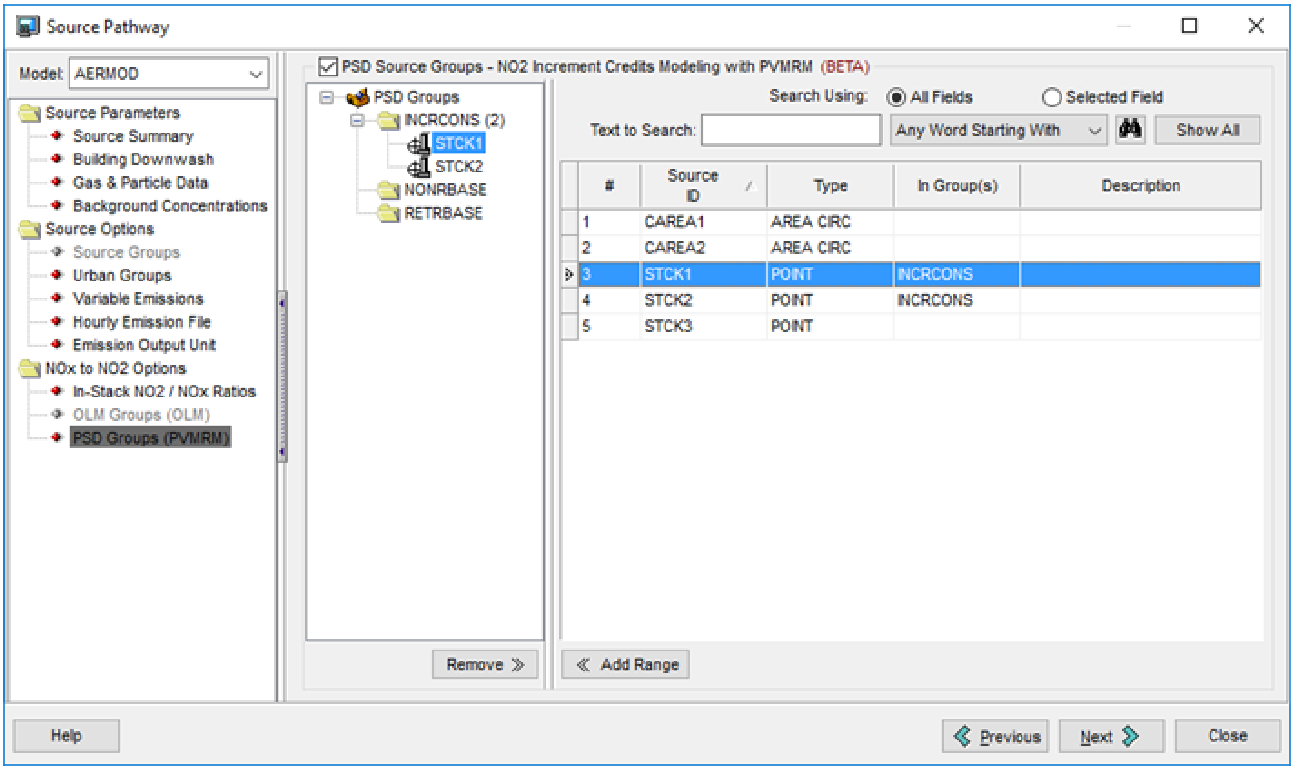
PSD Source Groups are unavailable until you take the following steps:

1. Enable **Modeling conversion of NOx to NO2** in the Dispersion Options screen of the Control Pathway.
2. Select the **PVMRM (Plume Volume Molar Ratio Method)** modeling option in the NOx to NO2 screen of the Control Pathway.
3. Enable **PSD Source Groups - NO2 Increment Credits Modeling with PVMRM (BETA)** in the PSD Groups screen.

> The Source Groups option is incompatible with PSD Source Groups. Enabling PSD source groups will disable Source Groups.

Select **Source Pathway - PSD Groups (PVMRM)** from the tree located on the left side of the Source Pathway dialog to display the PSD Group options.

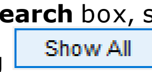
> Sources can only exist in one PSD source group at a time. If you place a source in more than one group, the data in its In Groups column of the table will display in red.



Source Pathway dialog - PSD Groups

The **PSD Source Groups** screen contains a navigation tree of the PSD source group on the left, with a table containing all sources in the project on the right. In this screen, you can:

- add a single source to any one of the three PSD source groups
- add a source range to any one of the three PSD source groups.

In addition, the screen also contains a search function at the top of the screen, which allows you to search for specific strings to filter the sources. Select either the **Search using All Fields** or **Search using Selected Field** radio button; the selected field is the field that the sources are being sorted by and is denoted by a triangle beside the field name. Enter the text in the **Text to Search** box, set the search terms, and click on the  button. View the entire, unfiltered list by clicking 
.

How to Add a Single Source to a PSD Source Group:

> You must define the sources for the current run before adding sources to any PSD source group.

1. From the list of sources on the right, select the sources to add to the PSD source group. You can select multiple sources from the list by pressing down the Shift key while you select the items in sequence or the Ctrl key to make disjoint selections.

2. Once you have selected the sources:

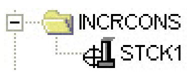
   - with the PSD source group selected, click on the 

 button to add the source(s) to the PSD source group.

   or

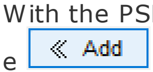
   - drag the source(s) into the PSD source group

3. The selected sources should then be displayed underneath the PSD source group you created.



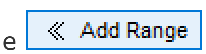
4. You may repeat steps 1 to 2 to add sources to the other PSD source groups.

How to Add a Source Range to a PSD Source Group:

1. From the list of sources on the right, select the first source in the range. With the PSD source group selected, drag the source into the PSD source group, or click on the 

 button.

2. From the list of sources on the right, select the last source in the range. Then:

   - in the navigation tree, select the first source in the range, and click on the 

 button to add the source(s) to the PSD source range.

   or

   - drag the source(s) on top of the first source in the range.

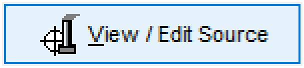
3. The source range will display within the PSD source group you created.

4. Double check ranges to ensure that they are logical by double-clicking the range to open the Source ID dialog.

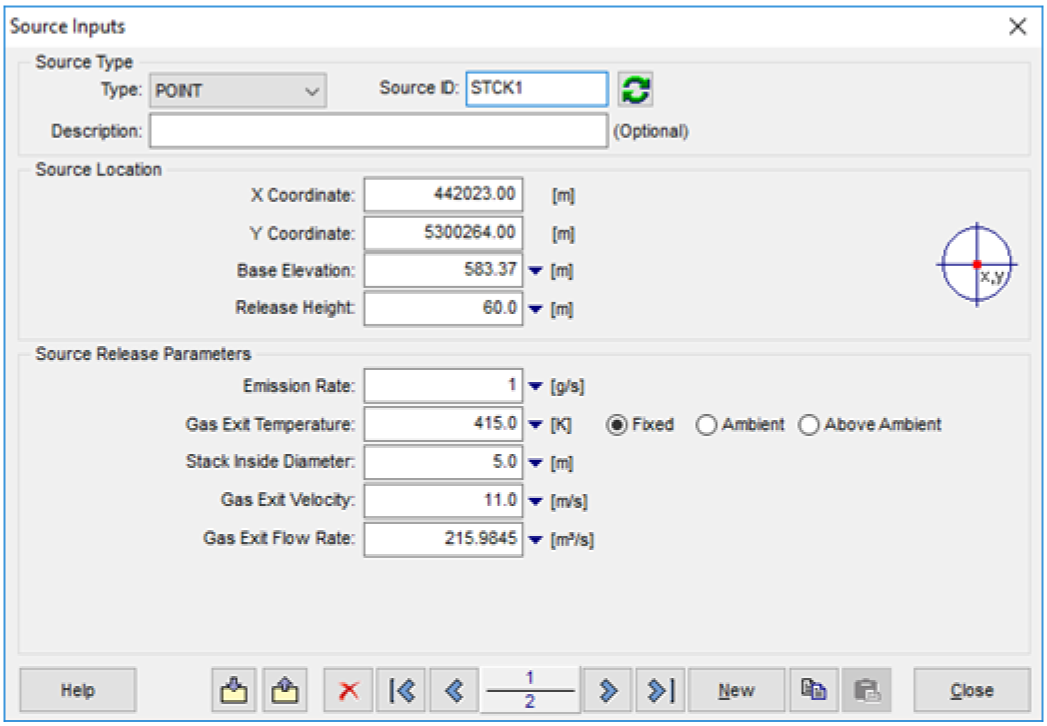
5. You may repeat steps 1 to 3 to add source ranges to the other PSD source groups. You can also add additional single sources to the group; these sources cannot be within the source range.

Source Inputs

You can specify all your sources in the Source Inputs dialog. You have access to this dialog by:

- Selecting **Data | Sources | Edit Sources...** from the menu.
- Clicking on the 

Source button located on the bottom of the Source Summary window.
- Double-click on a source in the Source Summary table.
- Right-click on a source in the Source Summary table to display the pop-up menu. Select the **View/Edit Source** option.



Source Inputs dialog

You can either define your sources in text mode or in graphical mode. To define sources in the text mode, simply enter all the necessary information directly into the **Source Inputs** dialog. To define sources in the graphical mode, you can use any of the source tools located on the Application Toolbar. The **Source Inputs** dialog opens automatically with some parameters already defined. Specify the additional parameters for your source. Here you can specify your source type, assign a source ID, source description and other parameters. Certain parameters requested in the **Source Inputs** dialog depend on the type of source being defined. The following source types are supported:

- Point
- Flare

- Area
- Open Pit
- Volume
- Circular Area
- Polygon Area
- Line Volume
- Line Area
- Buoyant Line
- Line

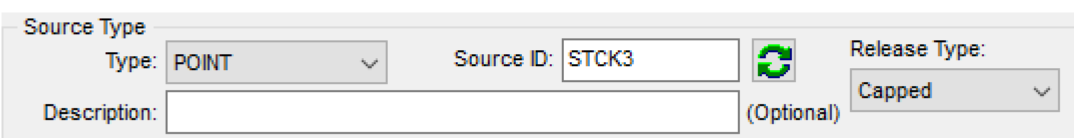
At the bottom of the Source Inputs dialog are the Record Navigator buttons, which will help you manage information for multiple sources defined.

> You can click on the or button to import or export source data to/from MS Excel. The data will be exported automatically in the Lakes Format. Data that is imported from Excel will need to be in this format as well.

Point Sources

Point sources are used if you want to model releases from sources like stacks and isolated vents. For a point source, you must provide the following information in the Source Inputs dialog:

Source Type



- **Type:** Select POINT from the drop-down list.

- **Source ID:** Enter here an identification name for the source being defined. The ID can be up to 8 characters long (12-characters for AERMOD) and are always in upper case. Press the button to get default Source IDs automatically set up for you.

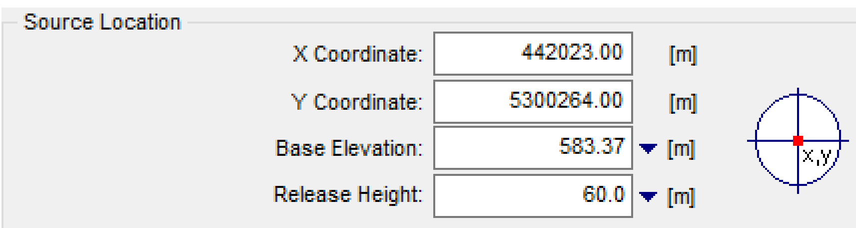
- **Release Type:** **AERMOD Only** Here you can specify whether your point source is a vertical, capped or horizontal point source.

This is a **BETA Non-Default AERMOD** option for version 15181 and earlier, available only if it has been selected in the Control Pathway. It will be overridden if the Regulatory Default Option is selected. For version 16216r and later, this is a default option.

See Also: Capped and Horizontal Stack Releases.

- **Description:** Enter here any description for the source up to 68 characters long, if you wish.

Source Location



All values entered in English units will be automatically written to the input file in metric units as the U.S. EPA models require that inputs be in metric units (with the exception of base elevation when ELEVUNIT equals FEET).

- **X Coordinate [m]:** Enter here the X coordinate for the source location in meters. The X coordinate may be input as UTM (Universal Transverse Mercator) coordinates or may be referenced to a user-defined origin.

- **Y Coordinate [m]:** Enter here the Y coordinate for the source location in meters. The Y coordinate may be input as UTM (Universal Transverse Mercator) coordinates or may be referenced to a user-defined origin.

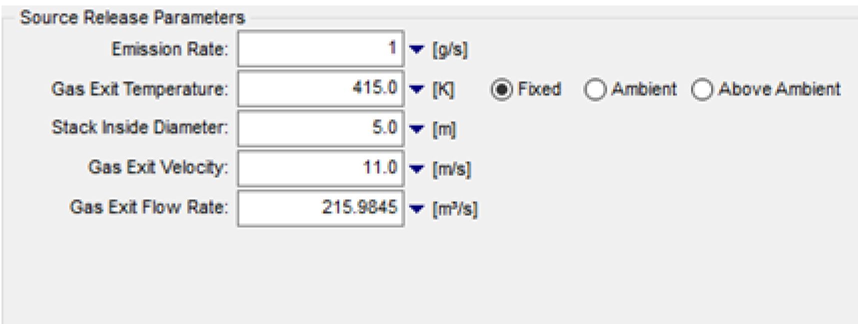
- **Base Elevation [m]:** Specify the source base elevation above mean sea level. The model only uses the source base elevation if Elevated Terrain is being used. You can specify the source base elevation in either meters [m] or in feet [ft].

AERMOD Only - If the non-default option Flat & Elevated has been selected in the Control Pathway then you will be able to disable the Base Elevation by unchecking the check box, indicating that the source is flat.

- **Release Height [m]:** Specify the release height above the ground in meters [m] or in feet [ft].

Release Height value must be less than 3000 m (9842 ft).

Source Release Parameters



- **Emission Rate:** Enter the emission rate of the pollutant in grams per second [g/s] or in pounds per hour [lb/hr]. The same emission rate is used for both concentration and deposition calculations.

- **Gas Exit Temperature:** Enter the exit temperature of the stack gas in degrees Kelvin [K] or in degrees Fahrenheit [F] or in degrees Celcius [C]. You can specify whether the plume exit temperature is released at ambient temperature, above ambient temperature, or exceeds the ambient temperature by a fixed amount.

  - **Fixed:** The temperature specified in this field is always used for this source.

  - **Ambient:** The ambient air temperature from the meteorological data file is used for this source for each hour of modeling. In the AERMOD input file, a value of 0 Kelvin indicates ambient temperature.

  - **Above Ambient:** A constant temperature above the ambient air temperature from the meteorological data file is used for this source. For example, if the user enters 50 degrees K, the model will use an exit temperature of the ambient air temperature plus 50 degrees for each hour of modeling. In hour 1, if the ambient temperature is 290 K, the model would run with an exit temperature of 340 K. In the AERMOD input file, a negative temperature value in Kelvin indicates this condition.

Gas Exit Temperature must be above 200 degrees K.

- **Stack Inside Diameter:** Enter the stack inside diameter in meters [m] or in feet [ft].

- **Gas Exit Velocity:** Enter the exit velocity of the stack gas in meters per second [m/s] or in feet per second [ft/s] or in kilometers per hour [km/hr]. The exit velocity can be determined from the following formula:

  $$V_s = \frac{4V}{\pi d^2_s}$$

  where:

  V_s = Exit Velocity

  V = Flow Rate

  d_s = Stack Inside Diameter

- **Gas Exit Flow Rate:** Enter the exit flow rate of the stack gas in cubic meters per second [m^3/s] or in cubic feet per second [ft^3/s] or in cubic feet per minute [ft^3/min]. The Gas Exit Flow Rate is calculated based on the Stack Inside Diameter and the Gas Exit Velocity (see the formula for Gas Exit Velocity).

> If the NO2/NOx ratio is omitted for a specific source, the value specified in the Control Pathway – NOx to NO2 for the option NO2/NOx Ratio will be used by the model.

Capped and Horizontal Stack Releases

> The capped and horizontal stack options are **non-deafult BETA** in AERMOD version 15181 and earlier. Staring with version 16216 this is a default option.

The AERMOD model contains options for modeling releases from capped and horizontal stacks. For sources that are not subject to building downwash, the plume rise for capped and horizontal stack releases is simulated based on an EPA Model Clearinghouse Memorandum. The procedure for these sources entails setting the exit velocity very low (0.001m/s) to account for suppression of vertical momentum for the plume, and using an effective stack diameter that maintains the actual flow rate of the plume. This procedure also addresses the issue of stack-tip downwash for these source release types.

The Model Clearinghouse procedure, however, is not considered appropriate for sources subject to downwash influences with the PRIME downwash algorithm. This is because the PRIME algorithm uses the stack diameter to define the initial radius of the plume for the numerical plume rise calculation; use of an effective diameter adjusted to maintain flow rate is not appropriate and could produce unrealistic results.

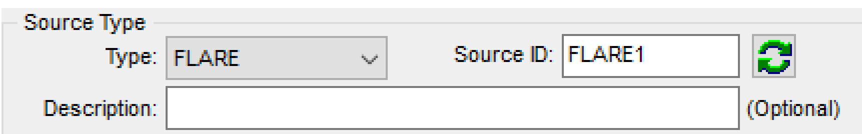
Therefore, the procedure for modeling capped and horizontal sources subject to downwash influences using this option has been adapted for the PRIME numerical plume rise formulation. If you have selected these options to model capped and horizontal sources, specify the actual stack parameters. The AERMOD model performs the necessary adjustments internally to account for plume rise and stack-tip downwash. For horizontal releases, the model currently assumes that the release is oriented with the wind direction.

Flare Sources

Flare sources are used as control devices for a variety of sources. Flares must comply with requirements specified in 40 CFR 60.18 and a minimum 98% reduction of all combustible components of the original emission must be achieved (U.S. EPA 1992b). The U.S. EPA models ISCST3, AERMOD, and ISC-PRIME do not have a specific source type option for flare sources but AERMOD View implemented a flare source type option. Flare sources can be treated in a similar way as point sources, except that there are buoyancy flux reductions associated with radiative heat losses and a need to account for flame length in estimating plume height (U.S. EPA 1992b).

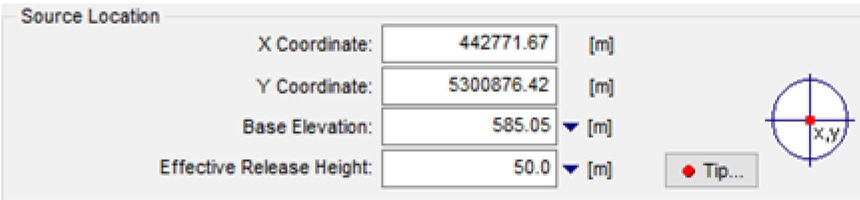
For a flare source, you must provide the following information in the Source Inputs dialog:

Source Type



- **Type:** Select FLARE from the drop-down list.
- **Source ID:** Enter here an identification name for the source being defined. The ID can be up to 8 characters long (12-characters for AERMOD) and are always in upper case. Press the button to get default Source IDs automatically set up for you.
- **Description:** Enter here any description for the source up to 68 characters long, if you wish.

Source Location



> All values entered in English units will be automatically written to the input file in metric units as the U.S. EPA models require that inputs be in metric units (with the exception of base elevation when ELEVUNIT equals FEET).

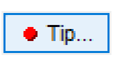
- **X Coordinate [m]:** Enter here the X coordinate for the source location in meters. The X coordinate may be input as UTM (Universal Transverse Mercator) coordinates or may be referenced to a user-defined origin.

- **Y Coordinate [m]:** Enter here the Y coordinate for the source location in meters. The Y coordinate may be input as UTM (Universal Transverse Mercator) coordinates or may be referenced to a user-defined origin.

- **Base Elevation [m]:** Specify the source base elevation above mean sea level. The model only uses the source base elevation if Elevated Terrain is being used. You can specify the source base elevation in either meters [m] or feet [ft].

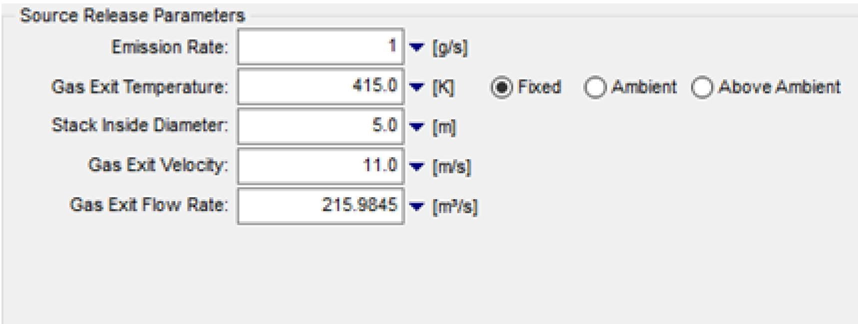
- **Effective Release Height [m]:** Specify the effective release height above the ground in meters [m] or feet [ft]. The effective release height should be given as the stack height plus the flare height.

> **Release Height** value must be less than 3000 m (9842 ft).



Click on this button to open the Effective Release Height dialog which allows you to calculate the effective release height above ground for flare sources.

Source Release Parameters



- **Emission Rate [g/s]:** Enter the emission rate of the pollutant in grams per second [g/s] or in pounds per hour [lb/hr]. The same emission rate is used for both concentration and deposition calculations.

- **Gas Exit Temperature [K]:** Enter the temperature of the released gas in degrees Kelvin [K] or in degrees Fahrenheit [F].

    - **Fixed:** The temperature specified in this field is always used for this source.

    - **Ambient:** The ambient air temperature from the meteorological data file is used for this source for each hour of modeling. In the AERMOD input file, a value of 0 Kelvin indicates ambient temperature.

    - **Above Ambient:** A constant temperature above the ambient air temperature from the meteorological data file is used for this source. For example, if the user enters 50 degrees K, the model will use an exit temperature of the ambient air temperature plus 50 degrees for each hour of modeling. In hour 1, if the ambient temperature is 290 K, the model would run with an exit temperature of 340 K. In the AERMOD input file, a negative temperature value in Kelvin indicates this condition.

> **Gas Exit Temperature** must be above 200 degrees K.

- **Stack Inside Diameter [m]:** Enter the stack inside diameter in meters [m] or in feet [ft].

- **Gas Exit Velocity [m/s]:** Enter the exit velocity of the gas in meters per second or feet per second [ft/sec] or kilometers per hour [km/hr]. The exit velocity can be determined from the following formula:

$$V_s = \frac{4V}{\pi d_s^2}$$

where:

V_s = Exit Velocity

V = Flow Rate

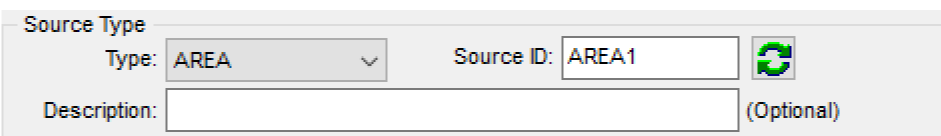
d_s = Stack Inside Diameter

- **Gas Exit Flow Rate [m^3/s]:** Enter the gas exit flow rate in cubic meters per second [m^3/s] or in cubic feet per second [ft^3/s] or in cubic feet per minute [ft^3/min].

Area Sources

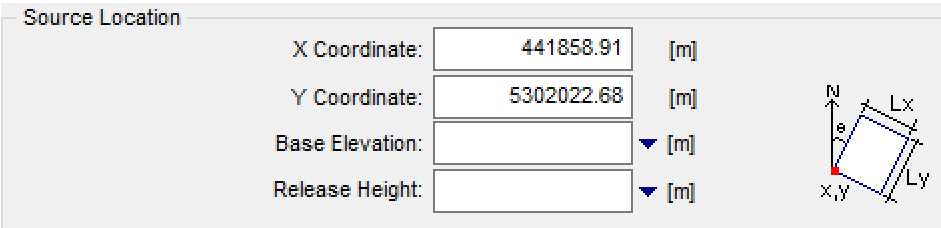
Area sources are used to model low-level or ground-level releases with no plume rise such as storage piles, slag dumps, and lagoons. The U.S. EPA models ISCST3, ISC-PRIME, and AERMOD accept rectangular areas that may also have a rotation angle specified relative to a north-south orientation. For an area source, you must provide the following information in the Source Inputs dialog:

Source Type



- **Type:** Select AREA from the drop-down list.
- **Source ID:** Enter here an identification name for the source being defined. The ID can be up to 8 characters long (12-characters for AERMOD) and are always in upper case. Press the button to get default Source IDs automatically set up for you.
- **Description:** Enter here any description for the source up to 68 characters long, if you wish.

Source Location

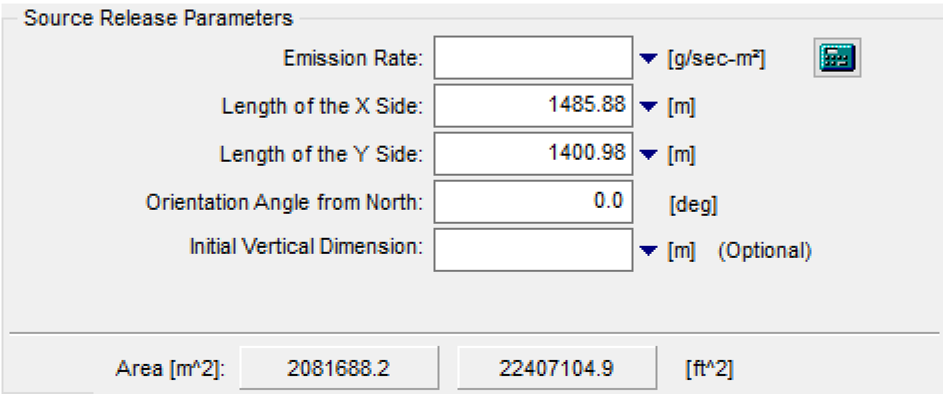


All values entered in English units will be automatically written to the input file in metric units as the U.S. EPA models require that inputs be in metric units (with the exception of base elevation when ELEVUNIT equals FEET).

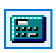
- **X Coordinate [m]:** Enter here the X coordinate for the vertex of the area source that occurs in the southwest quadrant of the source.
- **Y Coordinate [m]:** Enter here the Y coordinate for the vertex of the area source that occurs in the southwest quadrant of the source.
- **Base Elevation [m]:** Specify the source base elevation above mean sea level. The model only uses the source base elevation if Elevated Terrain is being used. You can specify the source base elevation in either meters [m] or feet [m].
- **Release Height [m]:** Enter the release height above ground in meters [m] or in feet [ft].

Release Height value must be less than 3000 m (9842 ft).

Source Release Parameters



- **Emission Rate [g/(s-m2)]:** Specify here the emission rate of the pollutant. The emission rate for area sources is input as an emission rate per unit area. The same emission rate is used for both concentration and deposition calculations. The emission rate can be entered in grams per second per square meter [g/(s-m2)] or in pounds per hour per square foot [lb/(hr-ft2)].

   Click on this button to display the Auto-Calculate Area Emissions Tool dialog to calculate the emission rate from grams per second [g/s] or from pounds per hour [lb/hr].

- **Length of the X Side [m]:** If the angle is 0 degrees, then this is the length, in meters [m] or in feet [ft], of the side of the area source that is in the east-west direction. If the angle is not equal to zero, then the X side dimension is measured from the side of the area source that is counterclockwise along the perimeter from the origin/vertex (x, y) defined by the X coordinate and Y coordinate values. See Figure 1 below for an illustration of this parameter.

- **Length of the Y Side [m]:** If the angle is 0 degrees, then this is the length, in meters [m] or in feet [ft], of the side of the area source that is in the north-south direction. If the angle is not equal to zero, then the Y side dimension is measured from the side of the area source that is clockwise along the perimeter from the origin/vertex (x, y) defined by the X coordinate and Y coordinate values. See Figure 1 below for an illustration of this parameter. If this parameter is omitted, then the model assumes that the area is a square and will use the same value entered for the length of the X side of the area.

- **Orientation Angle from North [deg]:** This the orientation angle for the rectangular area in degrees from North. The angle parameter is measured as the orientation relative to North of the side that is clockwise from the vertex (X and Y coordinate location), i.e. the side with Y side length. If this parameter is omitted, then the model assumes that the area is oriented in the north-south and east-west directions, i.e., angle = 0.0 degrees. If the angle parameter is input and the value is different from 0.0 degrees, then the model will rotate the area clockwise around the vertex defined in the X coordinate and Y coordinate input fields. The angle parameter must be positive for clockwise rotation and negative for counterclockwise rotation.

- **Initial Vertical Dim. of the Plume (Opt.) [m]:** This optional parameter may be used to specify an initial vertical dimension of the area source plume in meters [m] or in feet [ft]. This parameter is similar to the Initial Vertical Dimension parameter for Volume Sources. This parameter may be important when the area source algorithm is used to model mechanically-generated emission sources, such as mobile sources. In these cases, the emissions may be turbulently mixed near the source by the process that is generating the emissions, and therefore occupy some initial depth. For more passive area source emissions, such as evaporation or wind erosion, the Initial Vertical Dimension parameter may be omitted, which is equivalent to using an initial sigma-z of zero.

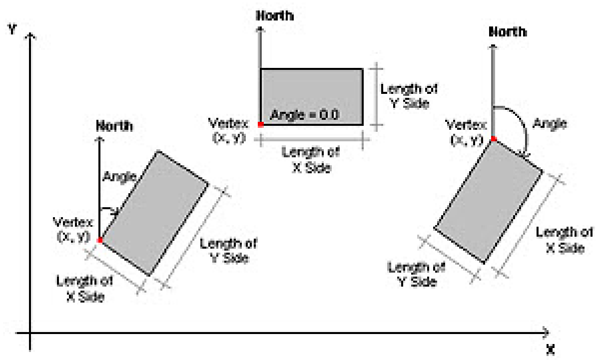


Figure 1. Relationship of area source parameters. Based on Figure 3-1 of the U.S. Environmental Protection Agency, 1995 User's Guide for the Industrial Source Complex (ISC3) Dispersion Models - Volume I, EPA-454/B-95-003a. U.S. Environmental Protection Agency. Research Triangle Park, NC 27711.

The aspect ratio, Length/Width, for area sources should be less than 10 to 1. If this is exceeded, the area should be divided to achieve a 10 to 1 aspect ratio, or less, for all sub-areas.

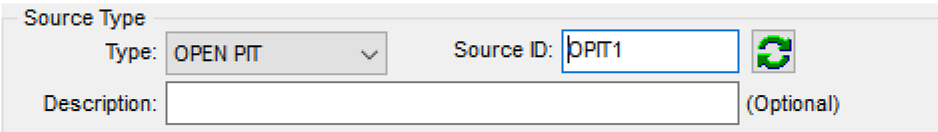
There are no restrictions on the location of receptors relative to area sources. Receptors may be placed within the area and at the edge of an area. The U.S. EPA models ISCST3, AERMOD, and ISC-PRIME will integrate over the portion of the area that is upwind of the receptor. The numerical integration is not performed for portions of the area that are closer than 1.0 meter upwind of the receptor. Therefore, caution should be used when placing receptors within or adjacent to areas that are less than a few meters wide.

Open Pit Sources

Open Pit sources are used to simulate fugitive emissions from below-grade open pits, such as surface coal mines and stone quarries. The Open Pit algorithm uses an effective area for modeling pit emissions, based on meteorological conditions, and then utilizes the numerical integration area source algorithm to model the impact of emissions from the effective area sources. The models accept rectangular pits with an optional rotation angle specified relative to a north-south orientation.

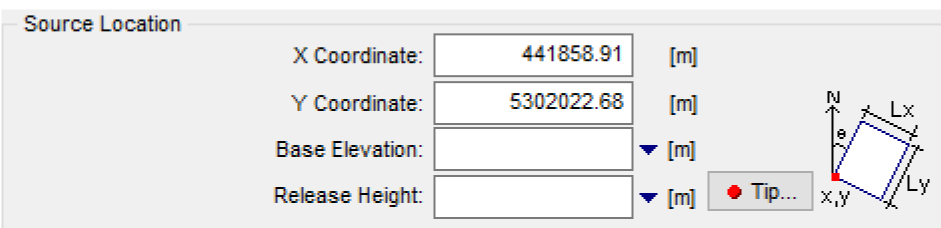
For an Open Pit source, you must provide the following information in the Source Inputs dialog:

Source Type



- **Type:** Select OPEN PIT from the drop-down list.
- **Source ID:** Enter here an identification name for the source being defined. The ID can be up to 8 characters long (12-characters for AERMOD) and are always in upper case. Press the button to get default Source IDs automatically set up for you.
- **Description:** Enter here any description for the source up to 68 characters long, if you wish.

Source Location



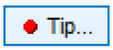
> All values entered in English units will be automatically written to the input file in metric units as the U.S. EPA models require that inputs be in metric units (with the exception of base elevation when ELEVUNIT equals FEET).

- **X Coordinate [m]:** Enter here the X coordinate for the vertex of the open pit source that occurs in the southwest quadrant of the source.
- **Y Coordinate [m]:** Enter here the Y coordinate for the vertex of the open pit source that occurs in the southwest quadrant of the source.
- **Base Elevation [m]:** Specify the source base elevation above mean sea level. The model only uses the source base elevation if Elevated Terrain is being used. You can specify the source base elevation in either meters [m] or feet [ft].
- **Release Height [m]:** Enter the average release height above the base of the pit in meters [m] or in feet [ft]. This parameter cannot exceed the effective depth of the pit, which is

calculated by the model based on the formula below. An average release height of 0.0 indicates emissions that are released from the base of the pit.

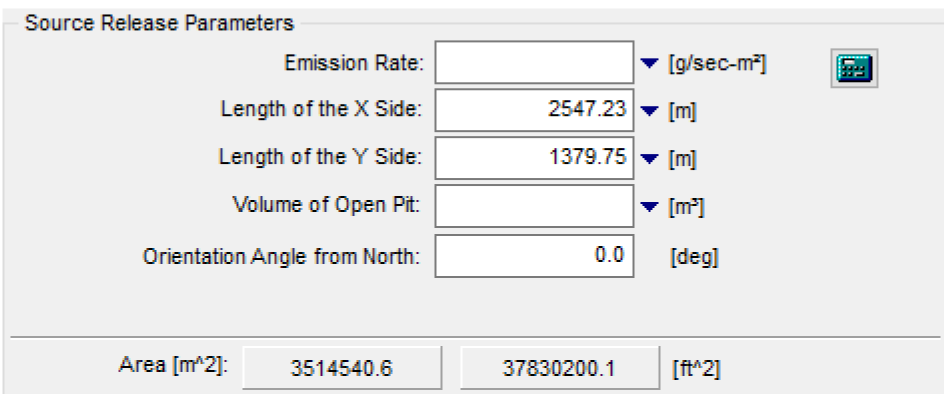
Release Height value must be less than 3000 m (9842 ft).

Effective Pit Depth = Pit Volume / (Pit Width x Pit Length)



Click on this button to open the Open Pit Parameters dialog which allows you to calculate the Average Release Height Above Pit Base.

Source Release Parameters



- **Open Pit Emission Rate [g/(s-m2)]:** Specify here the emission rate of the pollutant. The emission rate for open pit sources is input as an emission rate per unit area. The same emission rate is used for both concentration and deposition calculations. The emission rate can be entered in grams per second per square meter [g/(s-m2)] or in pounds per hour per square foot [lb/(hr-ft2)].

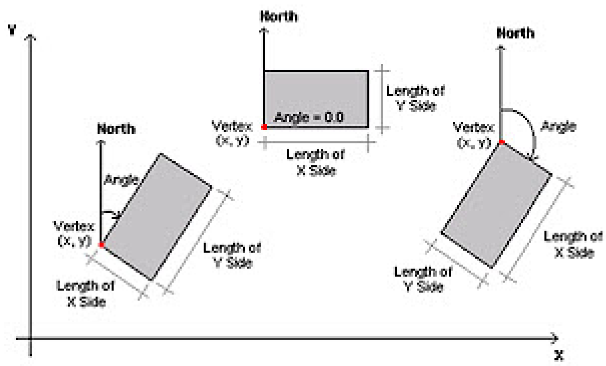
- Click on the button to display the Auto-Calculate Area Emissions Tool dialog to calculate the emission rate from grams per second [g/s] or from pounds per hour [lb/hr].

- **Length of the X Side [m]:** If the angle is 0.0 degrees, then this is the length, in meters [m] or in feet [ft], of the side of the open pit that is in the east-west direction. If the angle is not equal to zero, then the X side dimension is measured from the side of the open pit that is counterclockwise along the perimeter from the origin/vertex (x, y) defined by the X coordinate and Y coordinate values. See Figure 1 below for illustration of this parameter.

- **Length of Y the Side [m]:** If the angle is 0.0 degrees, then this is the length, in meters [m] or in feet [ft], of the side of the open pit that is in the north-south direction. If the angle is not equal to zero, then the Y side dimension is measured from the side of the open pit that is clockwise along the perimeter from the origin/vertex (x, y) defined by the X coordinate and Y coordinate values. See Figure 1 below for illustration of this parameter.

- **Volume of Open Pit [m3]:** This is the volume of the open pit in cubic meters [m^3] or in cubic feet [ft^3].

- **Orientation Angle from North [deg]:** This the orientation angle for the rectangular open pit in degrees from North. The angle parameter is measured as the orientation relative to North of the side that is clockwise from the vertex (X coordinate and Y coordinate location), i.e. the side with Y side length. If this parameter is omitted, then the model assumes that the open pit is oriented in the north-south and east-west directions, i.e., angle = 0.0 degrees. If the angle parameter is input and the value is different from 0.0 degrees, then the model will rotate the open pit clockwise around the vertex defined on the X coordinate and Y coordinate input fields. The angle parameter may be positive for clockwise rotation and negative for counterclockwise rotation. The value range accepted for the orientation is between -180 degrees and 180 degrees.



Relationship of open pit source parameters. Based on Figure 3-1 of the U.S. Environmental Protection Agency, 1995 User's Guide for the Industrial Source Complex (ISC3) Dispersion Models - Volume I, EPA-454/B-95-003a. U.S. Environmental Protection Agency. Research Triangle Park, NC 27711.

The aspect ratio, Length/Width, of open pit sources should be less than 10 to 1. However, since the pit algorithm generates an effective area for modeling emissions from the pit, and the size, shape and location of the effective area is a function of wind direction, an open pit cannot be subdivided into a series of smaller sources. If the aspect ration is greater than 10 to 1 the model will generate an warning message but it is allowed.

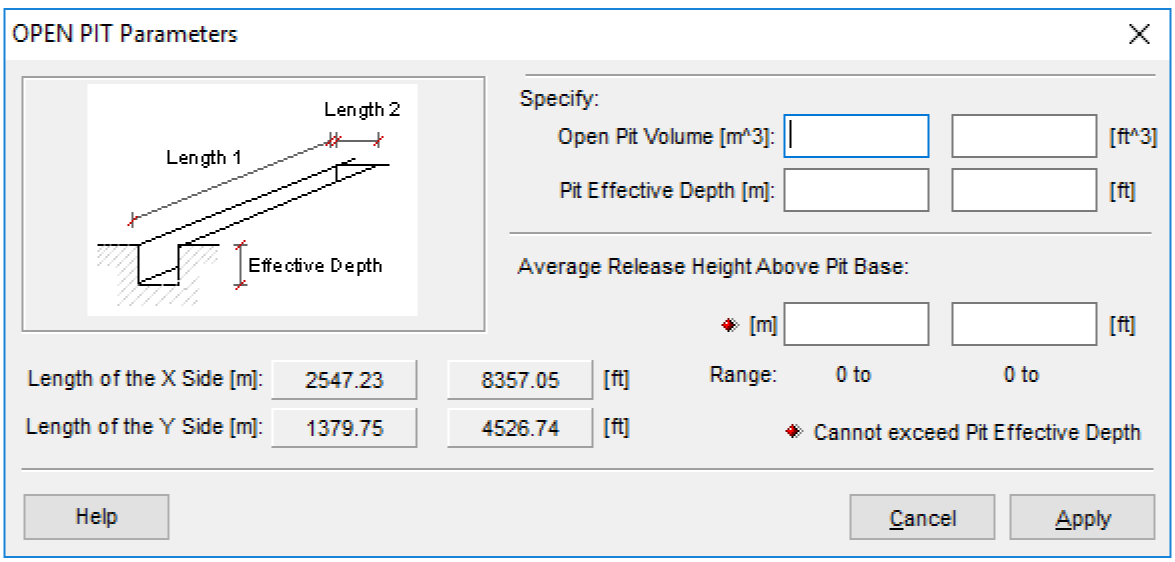
Since open pit sources cannot be subdivided, you should characterize irregularly-shaped pit areas by a rectangular shape of equal area.

Receptors should not be located within the boundaries of the pit. The models will set the concentration and/or deposition values at such receptors to zero. Such receptors will be identified during model setup and will be flagged in the summary of inputs.

Since the open pit algorithm is applicable for particulate emissions, the particle categories for an open pit source must be defined in the SO-Gas & Particle Data window.

Open Pit Parameters

The **Open Pit Parameters** dialog allows you to calculate the average release height above pit base for open pit sources. You have access to this dialog by pressing the Tip... button located in the Source Inputs dialog, only for Open Pit Sources.



Open Pit Parameters dialog

To calculate the Average Release Height Above Pit Base, the following parameters are requested:

- **Length of the X Side:** The length of the X side, in meters [m] or in feet [ft], that was entered in the Source Inputs - Open Pit dialog will be automatically displayed and used here.
- **Length of the Y Side:** The length of the Y side, in meters [m] or in feet [ft], that was entered in the Source Inputs - Open Pit dialog will be automatically displayed and used here.
- **Open Pit Volume:** Specify the volume of the open pit in cubic meters [m^3] or in cubic feet [ft^3].

- **Pit Effective Depth:** Enter a value for the effective depth of the open pit in meters [m] or in feet [ft].

> You do not have to enter both the **Open Pit Volume** and **Pit Effective Depth**. Define either one and press the Enter or Tab key to calculate and display the undefined parameter.

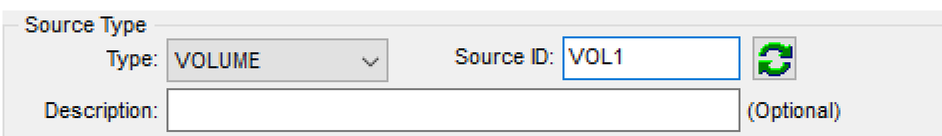
A range for the **Average Release Height Above Pit Base** will be calculated and displayed based on the **Pit Effective Depth**. Choose a value within that range, enter it in the text box and click the 

button.

Volume Sources

Volume sources are used to model releases from a variety of industrial sources, such as building roof monitors, multiple vents, and conveyor belts. For a volume source, you must provide the following information in the Source Inputs dialog:

Source Type

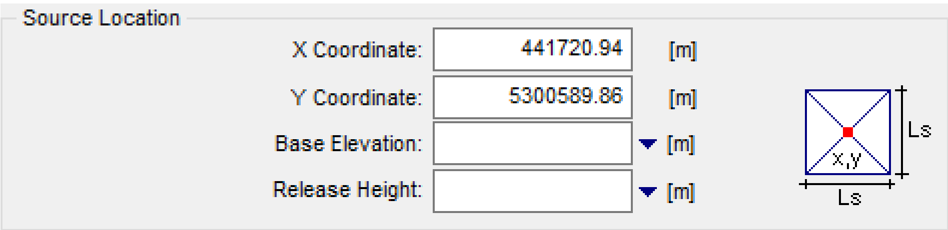


- **Type:** Select VOLUME from the drop-down list.

- **Source ID:** Enter here an identification name for the source being defined. The ID can be up to 8 characters long (12-characters for AERMOD) and are always in upper case. Press the  button to get default Source IDs automatically set up for you.

- **Description:** Enter here any description for the source up to 68 characters long, if you wish.

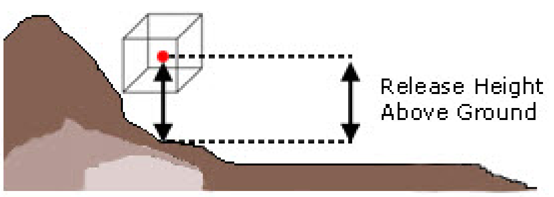
Source Location



> All values entered in English units will be automatically written to the input file in metric units as the U.S. EPA models require that inputs be in metric units (with the exception of base elevation when ELEVUNIT equals FEET).

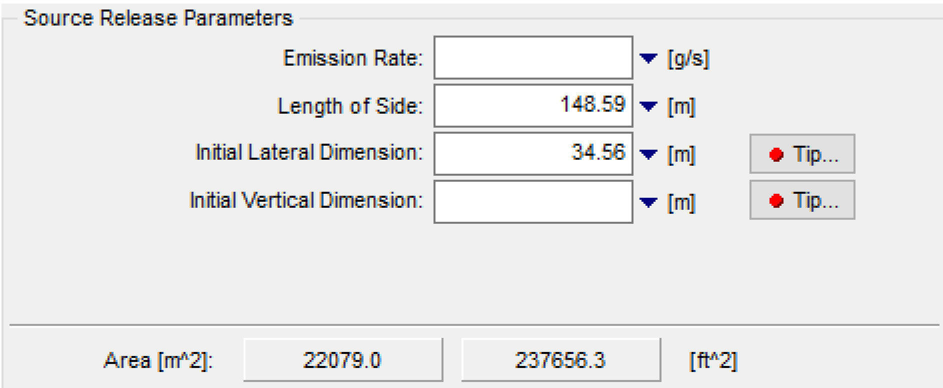
- **X Coordinate [m]:** Enter here the X coordinate for the source location in meters. This location is the center of the volume source.
- **Y Coordinate [m]:** Enter here the Y coordinate for the source location in meters. This location is the center of the volume source.
- **Base Elevation:** Specify the source base elevation above mean sea level. The model only uses the source base elevation if Elevated Terrain is being used. You can specify the source base elevation in either meters [m] or feet [ft].
- **Release Height:** Enter the release height above ground for the center of the volume source in meters [m] or in feet [ft]. See the diagram below for an example of where the release height above ground is measured.

> **Release Height** value must be less than 3000 m (9842 ft).



Release Height Above Ground for Volume Sources

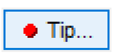
Source Release Parameters



- **Emission Rate:** Enter the emission rate of the pollutant in grams per second [g/s] or in pounds per hour [lb/hr]. The same emission rate is used for both concentration and deposition calculations.

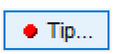
- **Length of Side:** Enter the length of the side of the volume source in meters [m] or in feet [ft]. The volume source cannot be rotated and has the X side equal to the Y side.

- **Initial Lateral Dimension:** Specify here the initial lateral dimension of the volume source.

  Click on this button to display the Initial Lateral Dimension dialog to automatically calculate the initial lateral dimension of the volume source.

- **Initial Vertical Dimension:** Specify here the initial vertical dimension of the volume source.

  Click on this button to display the Initial Vertical Dimension dialog to automatically calculate the initial vertical dimension of the volume source.

See Also: Procedures for Obtaining Initial Dimensions for US EPA suggested procedures and equations.

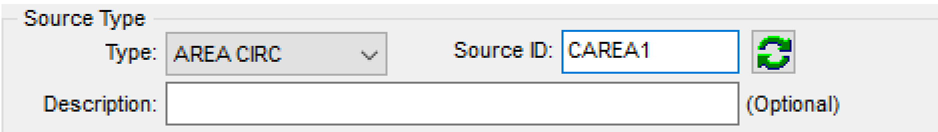
Circular Area Sources

ISCST3 & AERMOD Only

The Circular Area source may be used to specify a circular-shaped area source which is treated as an equal-area polygon. The circular area source type uses the same numerical integration algorithm for estimating impacts from area sources, and is merely a different option for specifying the shape of the area source.

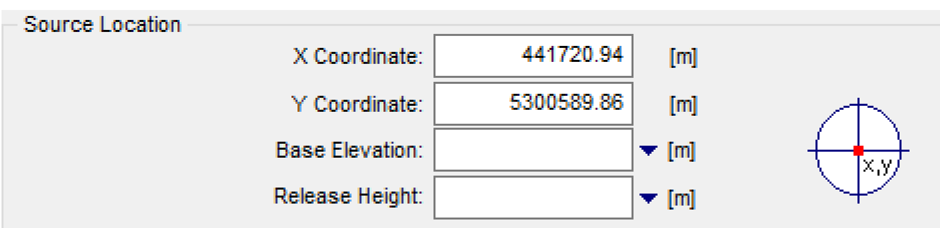
For a circular area source, you must provide the following information in the Source Inputs dialog:

Source Type



- **Type:** Select AREA CIRC from the drop-down list.
- **Source ID:** Enter here an identification name for the source being defined. The ID can be up to 8 characters long (12-characters for AERMOD) and are always in upper case. Press the button to get default Source IDs automatically set up for you.
- **Description:** Enter here any description for the source up to 68 characters long, if you wish.

Source Location



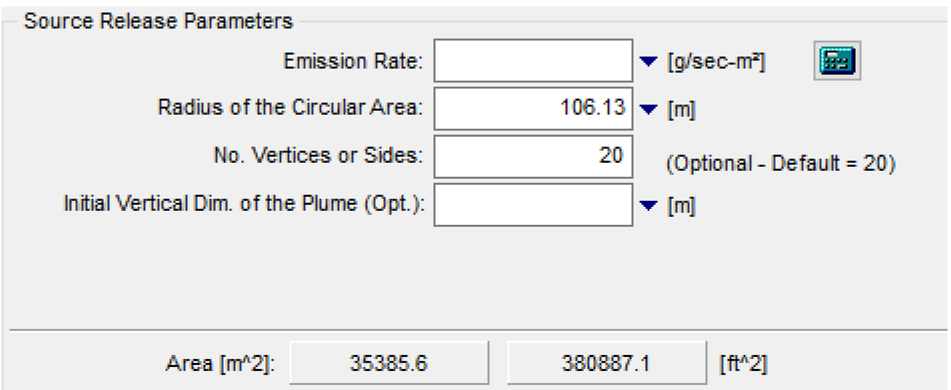
> All values entered in English units will be automatically written to the input file in metric units as the U.S. EPA models require that inputs be in metric units (with the exception of base elevation when ELEVUNIT equals FEET).

- **X Coordinate [m]:** Enter here the X coordinate for the center of the circular area source.
- **Y Coordinate [m]:** Enter here the Y coordinate for the center of the circular area source.
- **Base Elevation [m]:** Specify the source base elevation above mean sea level. The model only uses the source base elevation if Elevated Terrain is being used. You can specify the source base elevation in either meters [m] or feet [ft].

- **Release Height [m]:** Specify the release height above the ground in meters [m] or in feet [ft].

> **Release Height** value must be less than 3000 m (9842 ft).

Source Release Parameters



- **Emission Rate [g/(s-m2)]:** Enter the emission rate of the pollutant. The emission rate for circular area sources is input as an emission rate per unit area. The same emission rate is used for both concentration and deposition calculations. The emission rate can be entered in grams per second per square meter [g/(s-m2)] or in pounds per hour per square foot [lb/(hr-ft2)]. Click on the button to display the Auto-Calculate Area Emissions Tool dialog to calculate the emission rate from grams per second [g/s] or from pounds per hour [lb/hr].

- **Radius of the Circular Area [m]:** Enter here the radius of the circular area source in meters [m] or in feet [ft].

- **No. of Vertices or Sides:** This is the number of vertices or sides of the circular area source polygon. The polygon will have the same area as that specified for the circle.

> The U.S. EPA model ISCST3 can generate a regular polygon of up to 20 sides to approximate the circular area source. If a polygon with over 20 vertices is created, the ISCST3 run will be aborted (unsuccessful run) and no error message will be displayed in the Output File.

- **Initial Vertical Dim. of the Plume (Opt.) [m]:** This optional parameter may be used to specify an initial vertical dimension of the circular area source plume in meters [m] or in feet [ft]. This parameter is similar to the Initial Vertical Dimension parameter for Volume Sources.

The Initial Vertical Dimension parameter may be omitted, which is equivalent to using an initial sigma-z of zero.

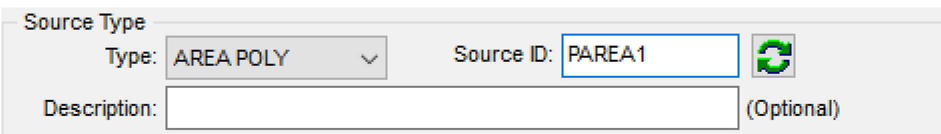
Polygon Area Sources

ISCST3 & AERMOD Only

The Polygon Area source may be used to specify an area source as an arbitrarily shaped polygon. This source type option gives considerable flexibility for specifying the shape of an area source. The polygon area source type uses the same numerical integration algorithm for estimating impacts from area sources and is merely a different option for specifying the shape of the area source.

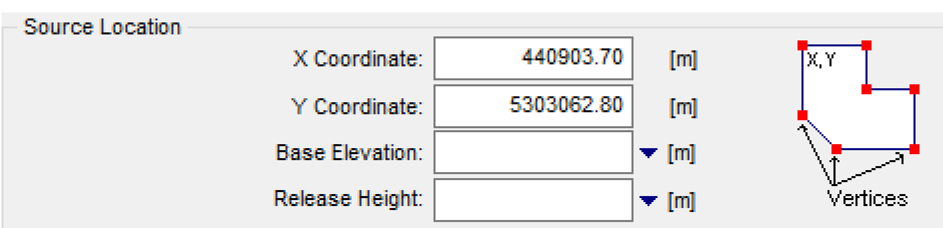
For a polygon area source, you must provide the following information in the Source Inputs dialog:

Source Type



- **Type:** Select AREA POLY from the drop-down list.
- **Source ID:** Enter here an identification name for the source being defined. The ID can be up to 8 characters long (12-characters for AERMOD) and are always in upper case. Press the button to get default Source IDs automatically set up for you.
- **Description:** Enter here any description for the source up to 68 characters long, if you wish.

Source Location

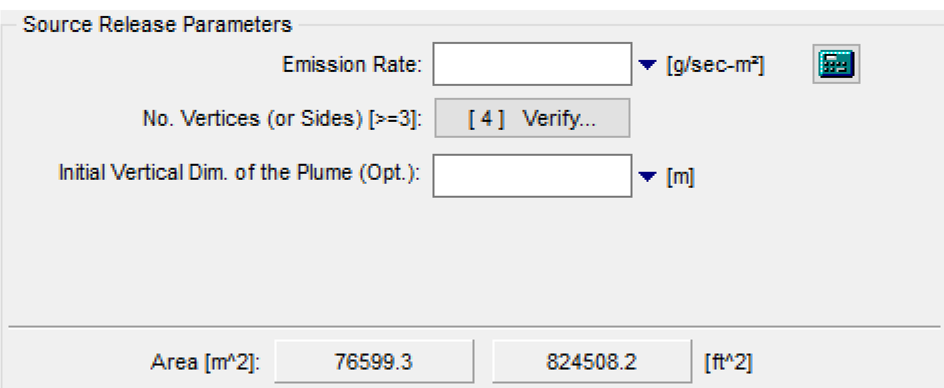


> All values entered in English units will be automatically written to the input file in metric units as the U.S. EPA models require that inputs be in metric units (with the exception of base elevation when ELEVUNIT equals FEET).

- **X Coordinate [m]:** Enter here the X coordinate for one of the vertices of the polygon area source.
- **Y Coordinate [m]:** Enter here the Y coordinate for one of the vertices of the polygon area source.
- **Base Elevation [m]:** Specify the source base elevation above mean sea level. The model only uses the source base elevation if Elevated Terrain is being used. You can specify the source base elevation in either meters [m] or feet [m].
- **Release Height [m]:** Enter the release height above ground in meters [m] or in feet [ft].

> **Release Height** value must be less than 3000 m (9842 ft).

Source Release Parameters



- **Emission Rate [g/(s-m2)]:** Enter the emission rate of the pollutant. The emission rate for the polygon area source is input as an emission rate per unit area. The same emission rate is used for both concentration and deposition calculations. The emission rate can be entered in grams per second per square meter [g/(s-m2)] or in pounds per hour per square foot [lb/(hr-ft2)]. Click on the button to display the Auto-Calculate Area Emissions Tool dialog to calculate the emission rate from grams per second [g/s] or from pounds per hour [lb/hr].
- **No. of Vertices (or Sides):** Click on the [?] Specify... button to display the Area Poly Vertex Coordinates dialog were you can specify or verify the coordinates for the vertices of

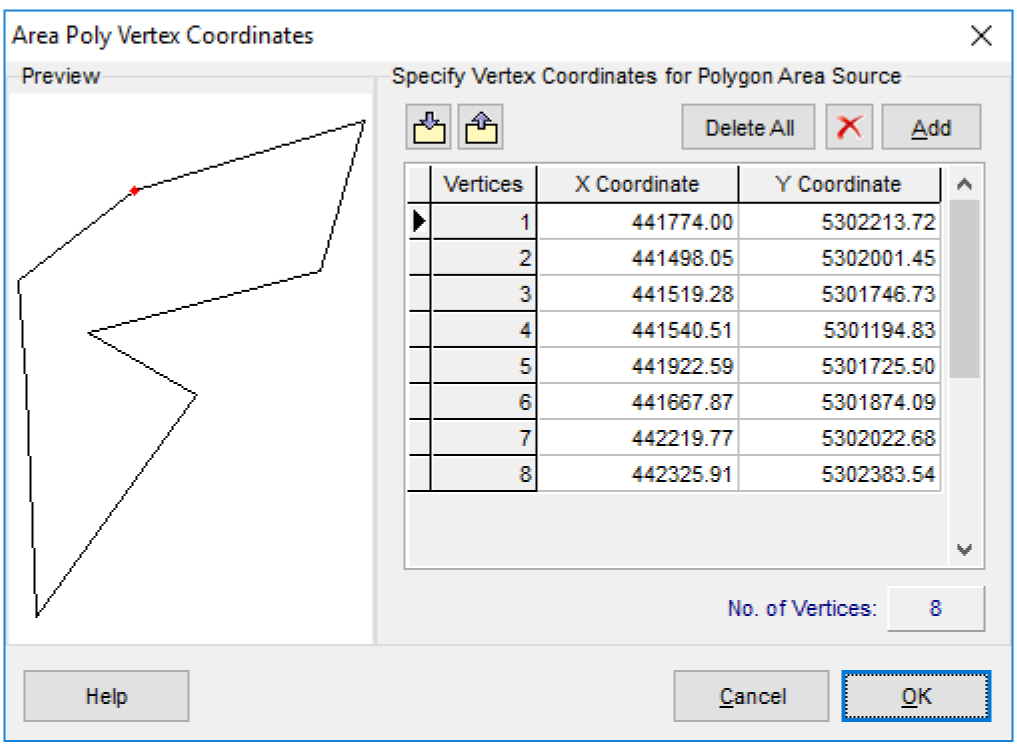
your polygon area. Once the vertices are defined, the [4] Verify... button will display the number of vertices for you polygon area.

> The U.S. EPA model ISCST3 can generate a regular polygon of between 3 and 20 sides. If a polygon with over 20 vertices is created, the ISCST3 run will be aborted (unsuccessful run) and no error message will be displayed in the Output File. For the AERMOD model, the maximum number of vertices is dynamically allocated and is therefore limited only by the amount of memory available for model execution.

- **Initial Vertical Dim. of the Plume (Opt.) [m]:** This optional parameter may be used to specify an initial vertical dimension of the polygon area source plume in meters [m] or in feet [ft]. This parameter is similar to the Initial Vertical Dimension parameter for Volume Sources. The Initial Vertical Dimension parameter may be omitted, which is equivalent to using an initial sigma-z of zero.

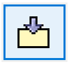
Area Poly Vertex Coordinates

You have access to the **Area Poly Vertex Coordinates** dialog by pressing the [6] Verify... or [?] Specify... button in the **Source Inputs** dialog for Polygon Area Sources only. Here you can specify or verify the coordinates for the vertices of your polygon area source.



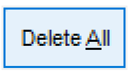
Area Poly Vertex Coordinates dialog

If you are simply verifying the vertex coordinates, they will already be displayed in the table where you can modify them if you wish. The following buttons are available:

 Click on this button to import the vertices for a polygon area source in *.csv format

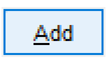
 Click on this button to export the vertices for a polygon area source in *.csv format

> Please note that the current version of AERMOD View is limited to importing 250 vertices for an area polygon source.



Press this button to delete all the vertices

 Press this button to delete the selected vertex from the polygon area source.



Press this button to add a vertex to the polygon area source.

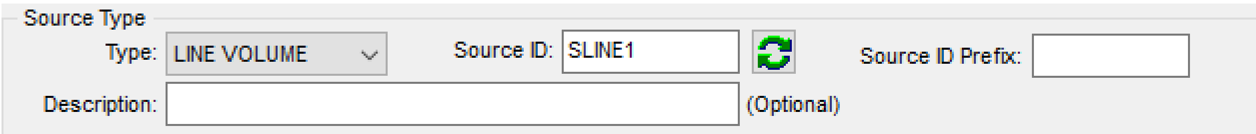
Line Volume Sources

The U.S. EPA models ISCST3, ISC-PRIME, and AERMOD can handle line sources as a series of volume sources. AERMOD View can automatically generate these volume sources to represent the line source that you specify. Examples of line sources include haul roads, conveyor belts, rail lines, etc.

> AERMOD View supports two methods - The "Line Source Represented by Separated Volume Sources" method and the "Line Source Represented by Adjacent Volume Sources" as described in Volume II of the U.S. EPA User's Guide for the Industrial Source Complex (ISC3) Dispersion Models (US EPA 1995e).

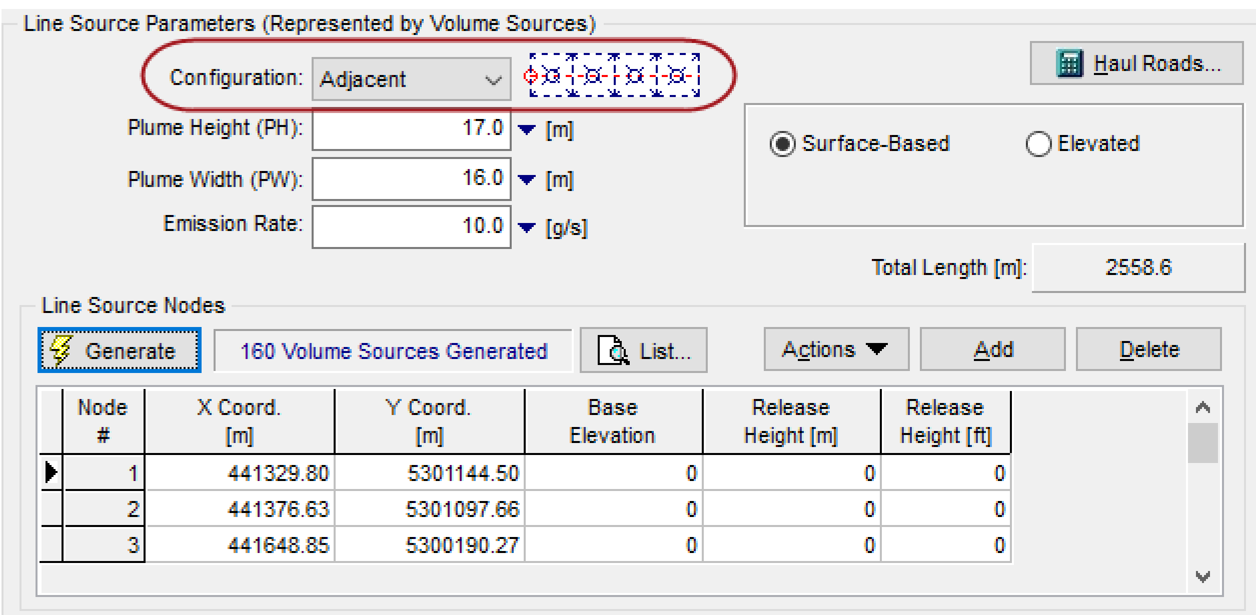
For a LINE VOLUME source, you must provide the following information in the Source Inputs dialog:

Source Type

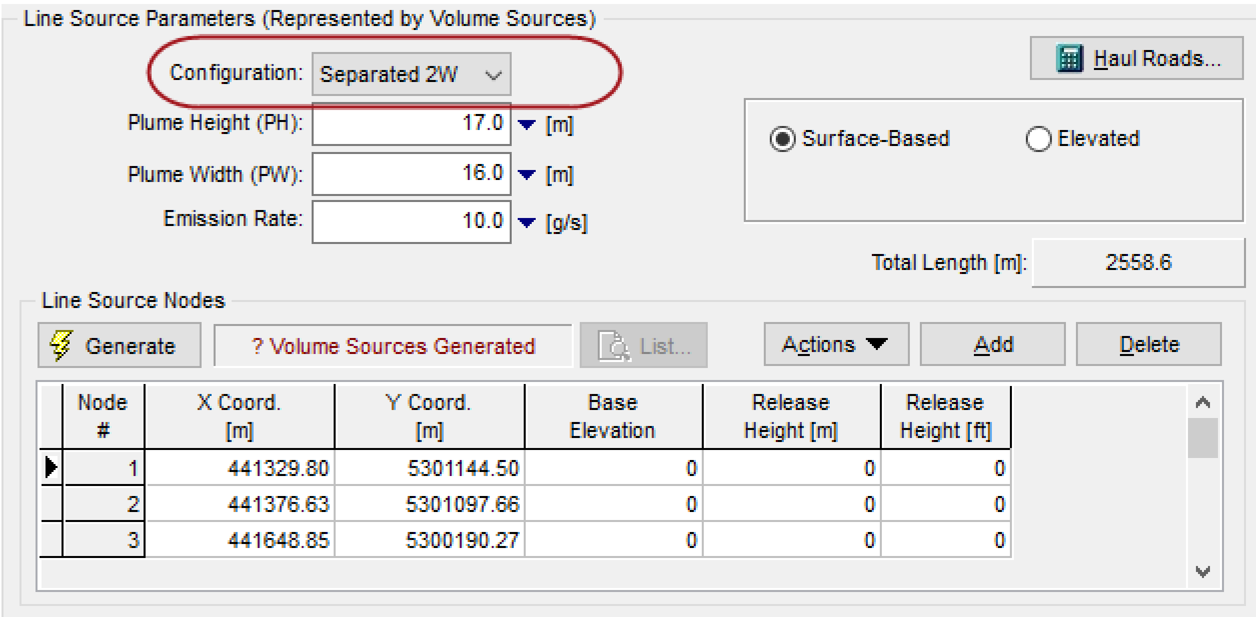


- **Type:** Select LINE VOLUME from the drop-down list.
- **Source ID:** Enter here an identification name for the source being defined. The ID can be up to 8 characters long (12-characters for AERMOD) and are always in upper case. Press the button to get default Source IDs automatically set up for you.
- **Source ID Prefix:** Enter here a prefix up to 4 characters long for the identification name for the volume sources to be generated for the LINE VOLUME source.
- **Description (Optional):** you can specify here any description for the source up to 68 characters long.

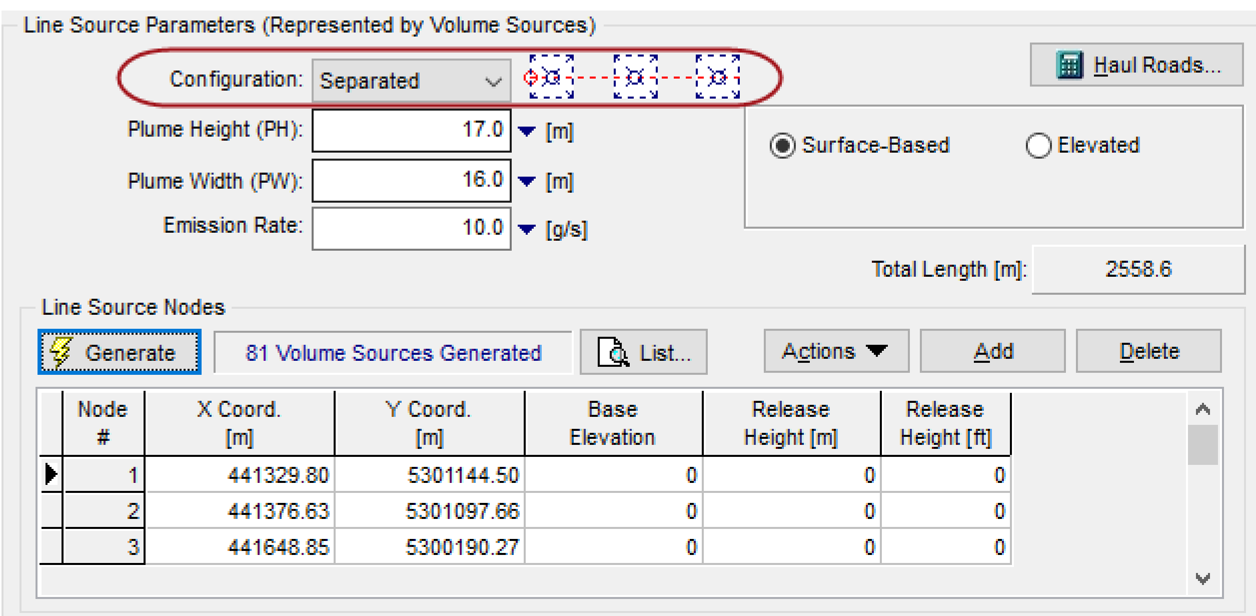
Line Source Parameters - Represented by Adjacent Volume Sources



Line Source Parameters - Represented by Separated 2W Volume Sources (regulatory standard)



Line Source Parameters - Represented by Separated Volume Sources



- **Configuration:** Three volume source configurations are available:
  - **Adjacent:** In this configuration, volume sources are placed adjacent to each other.
  - **Separated 2W:** In this configuration, volume source centers are separated by 2 X volume source width. If there is not sufficient space to place another volume source, the line is truncated. This configuration is based on the source type description in the ISCST3 model user's guide and is preferred to the **Separated** configuration by the US EPA.
  - **Separated:** In this configuration, volume sources are separated from each other with a space. The line starts and ends with a volume source and the volume sources are distributed along the line so that the distance between the volume source centers is <= volume source width.
- **Plume Height (PH) [m]:** If your line is surface-based or elevated and not on or adjacent to a building, then you must specify the vertical dimension/plume height of the line source here in meters [m] or in feet [ft]. Note that if the source is on or adjacent to a building this option is not available.
- **Plume Width (PW) [m]:** Enter the length of the side of the line source (plume width) in meters [m] or in feet [ft].
- **Emission Rate [g/s]:** Enter the emission rate of the pollutant in grams per second [g/s] or in pounds per hour [lb/hr]. The same emission rate is used for both concentration and deposition calculations.
- **Building Height [m]:** If your line source is elevatedand is on or adjacent to a building, then you need to specify the building height. The building height will be used to calculate the Initial Vertical Dimension of the source. Note that if the source is surface-based, then this option is not available.
- **Source Elevation:** In this panel, you should specify whether the source is surface-based or elevated. The selection of one option or the other will determine how the Initial Vertical Dimension (Initial Sigma Z) of the generated volume sources will be calculated.
- **Surface-Based:** An example of a surface-based volume source is a surface rail line or haul road.
- **Elevated:** An example of an elevated line/volume source is an elevated rail line.
- **Total Length [m]:** This is the total length for all line segments.
- **# Volume Sources Generated:** This field shows the number of volume sources generated to represent the line segments.

> Do not locate receptors within the Volume Source Exclusion Zone. The exclusion zone is calculated as [(2.15 x Sigma Y) + 1 m].

> All values entered in English units will be automatically written to the input file in metric units as the U.S. EPA models require that inputs be in metric units (with the exception of base elevation when ELEVUNIT equals FEET).

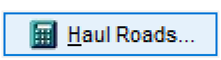
Line Source Nodes

For each node of the line source, you must specify the following information:

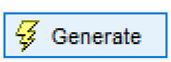
- **X Coordinate [m]:** Enter here the X coordinate for the node in meters.
- **Y Coordinate [m]:** Enter here the Y coordinate for the node in meters.
- **Base Elevation [m]:** Enter here base elevation at the node location. The model only uses the source base elevation if Elevated Terrain is being used. You can specify the source base elevation in either meters [m] or feet [m].
- **Release Height [m]:** Enter the release height above ground in meters [m] at the node location.
- **Release Height [ft]:** Enter the release height above ground in feet [ft] at the node location.

> **Release Height** value must be less than 3000 m (9842 ft).

Available Buttons



Press this button to have access to the Haul Road Calculator.

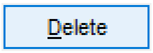


Press the button to re-generate the list of volume sources.

Press this button to display the Generated Volume Sources dialog where you can view the source parameters for each generated area source.

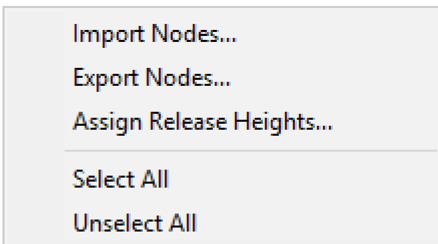
Press this button to have access to several options which are described below.

Press this button to add a new row where you can enter the parameters for a node.



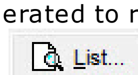
Press this button to delete the selected row/node.

Actions button Menu Options

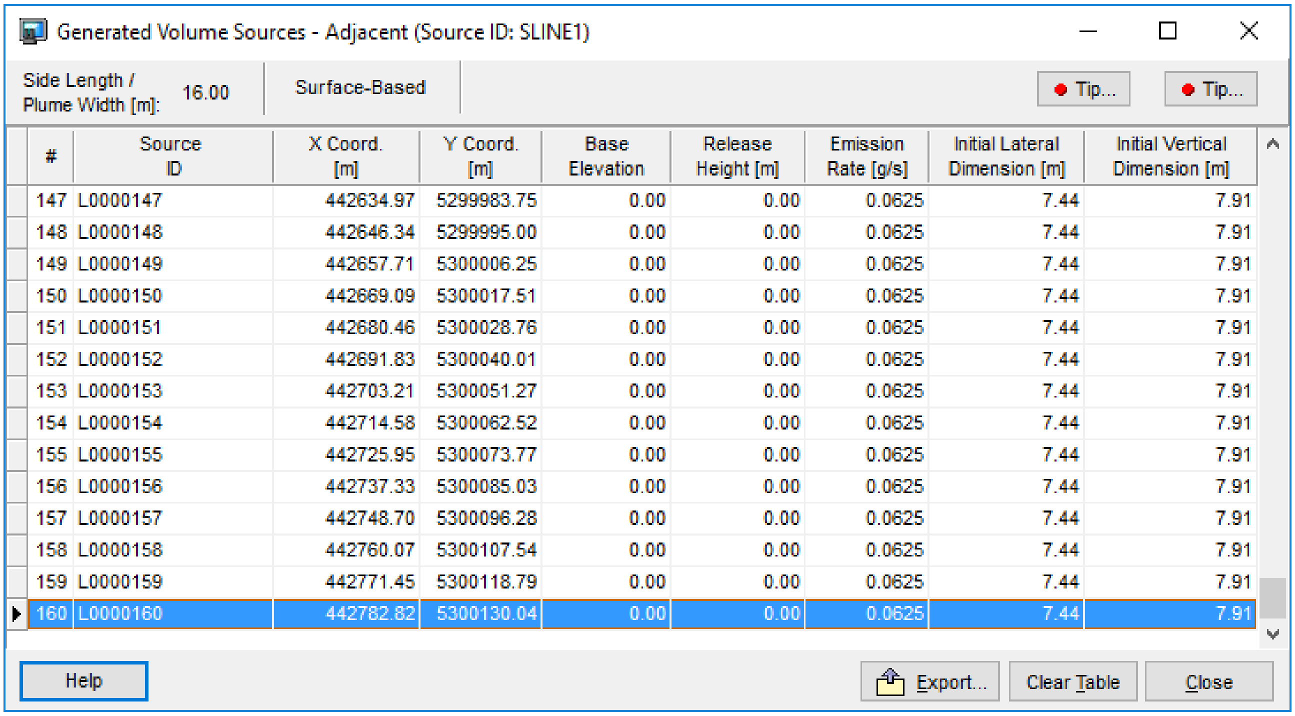


- **Import Nodes:** Allows you to import line source nodes from a CSV file (X,Y,Base_Elevation,Release_Height)
- **Export Notes:** Allows you to export line source nodes to a CSV file (X,Y,Base_Elevation,Release_Height)
- **Assign Release Heights:** Displays the Assign Release Heights dialog from where you can specify the Release Height value to be assigned to all selected nodes.
- **Select All:** Automatically selects all nodes/rows from the table.
- **Unselect All:** Automatically unselects all nodes/rows from the table.

Generated Volume Sources

The **Generated Volume Sources** dialog allows you to view the parameters that are automatically calculated by AERMOD View for the volume sources generated to represent the line source you have defined. You have access to this dialog by pressing the 

button located in the Source Inputs dialog for Line Volume Sources and Line Area Sources.



#	Source ID	X Coord. [m]	Y Coord. [m]	Base Elevation	Release Height [m]	Emission Rate [g/s]	Initial Lateral Dimension [m]	Initial Vertical Dimension [m]
147	L0000147	442634.97	5299983.75	0.00	0.00	0.0625	7.44	7.91
148	L0000148	442646.34	5299995.00	0.00	0.00	0.0625	7.44	7.91
149	L0000149	442657.71	5300006.25	0.00	0.00	0.0625	7.44	7.91
150	L0000150	442669.09	5300017.51	0.00	0.00	0.0625	7.44	7.91
151	L0000151	442680.46	5300028.76	0.00	0.00	0.0625	7.44	7.91
152	L0000152	442691.83	5300040.01	0.00	0.00	0.0625	7.44	7.91
153	L0000153	442703.21	5300051.27	0.00	0.00	0.0625	7.44	7.91
154	L0000154	442714.58	5300062.52	0.00	0.00	0.0625	7.44	7.91
155	L0000155	442725.95	5300073.77	0.00	0.00	0.0625	7.44	7.91
156	L0000156	442737.33	5300085.03	0.00	0.00	0.0625	7.44	7.91
157	L0000157	442748.70	5300096.28	0.00	0.00	0.0625	7.44	7.91
158	L0000158	442760.07	5300107.54	0.00	0.00	0.0625	7.44	7.91
159	L0000159	442771.45	5300118.79	0.00	0.00	0.0625	7.44	7.91
160	L0000160	442782.82	5300130.04	0.00	0.00	0.0625	7.44	7.91

Generated Volume Sources dialog

> At the present time AERMOD View uses the "Line Source Represented by Separated Volume Sources" method as described in Volume II of the U.S. EPA User's Guide for the Industrial Source Complex (ISC3) Dispersion Models (US EPA 1992).

The following parameters are displayed for each volume source:

- **Source ID:** Identification name for the generated volume source.
- **X Coord:** X coordinate for the source location in meters. This location is for the center of the volume source.
- **Y Coord:** Y coordinate for the source location in meters. This location is for the center of the volume source.
- **Base Elevation:** The source base elevation is displayed in either feet or meters depending on what has been selected in the Source Inputs window. The base elevation is interpolated from the base elevation specified for the two nodes of the same line segment.
- **Release Height:** Release height above ground in meters (center of volume source). The release height is interpolated from the release height specified for the two nodes of the same line segment.

- **Emission Rate:** Emission rate of the pollutant in grams per second. The same emission rate is used for both concentration and deposition calculations. The emission rate is calculated by dividing the specified emission rate for the line source equally among the individual volume sources.

- **Initial Lateral Dimension:** This parameter is calculated by AERMOD View.

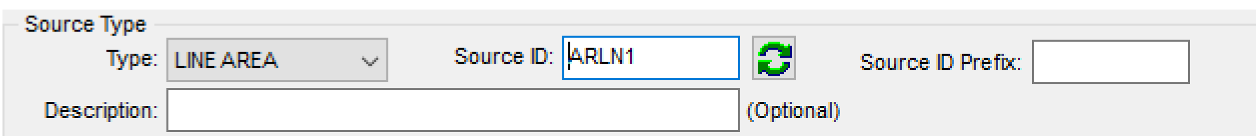
- **Initial Vertical Dimension:** This parameter is calculated by AERMOD View.

Line Area Sources

The **Line Area Source** tool is an option specific to AERMOD View which allows you to define a series of area sources to represent a line.

For the line area source, you must provide the following information in the Source Inputs dialog:

Source Type



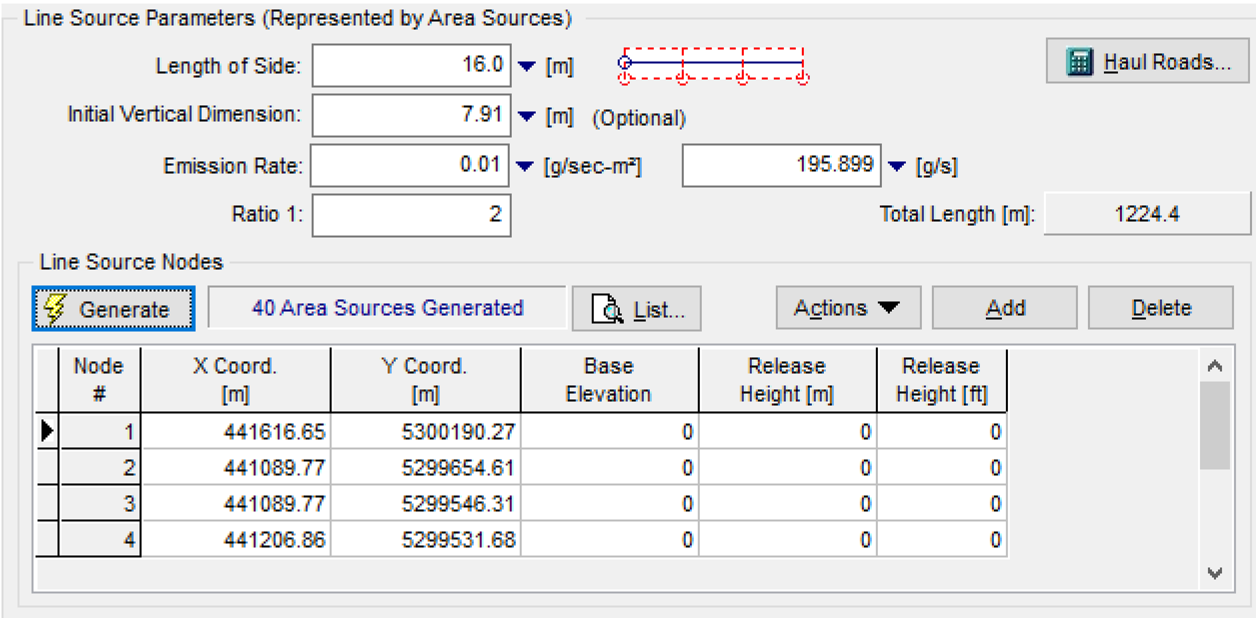
- **Type:** Select LINE AREA from the drop-down list.

- **Source ID:** Enter here an identification name for the source being defined. The ID can be up to 8 characters long (12-characters for AERMOD) and are always in upper case. Press the  button to get default Source IDs automatically set up for you.

- **Source ID Prefix:** Enter here a prefix up to 4 characters long for the identification name for the area sources to be generated for the line area source.

- **Description (Optional):** you can specify here any description for the source up to 68 characters long.

Line Source Parameters (Represented by Area Sources)



- **Length of Side [m]:** Enter the length of the side of the line source in meters [m] or in feet [ft].

- **Initial Vertical Dimension (Optional) [m]:** Also known as Sigma Z, this optional parameter may be used to specify an initial vertical dimension of the area source plume in meters [m] or in feet [ft]. This parameter may be important when the area source algorithm is used to model mechanically-generated emission sources, such as mobile sources. In these cases, the emissions may be turbulently mixed near the source by the process that is generating the emissions, and therefore occupy some initial depth. For more passive area source emissions, such as evaporation or wind erosion, the Initial Vertical Dimension parameter may be omitted, which is equivalent to using an initial Sigma Z of zero.

- **Emission Rate [g/s-m^2]:** Enter the emission rate of the pollutant in grams per second per square meter. The value is also automatically converted and displayed as grams per second for the entire area. You may also choose to enter the Emission Rate as grams per second, in which case it will also be converted into grams per second per meter squared.

- **Ratio 1:** This is the aspect ratio, Length/Width, for area sources which is usually recommended to be less than 100 to 1. If you want the area sources created to be square in shape, than specify a Ratio of 1 (1:1). If an aspect ration of 1:10 is required, specify a value Ratio value of 10.

- **Total Length [m]:** This is the total length for all line segments.

- **# Area Sources Generated:** This field shows the number of area sources generated to represent the line segments.

All values entered in English units will be automatically written to the input file in metric units as the U.S. EPA models require that inputs be in metric units (with the exception of base elevation when ELEVUNIT equals FEET).

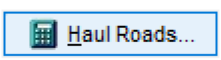
Line Source Nodes

For each node of the line source, you must specify the following information:

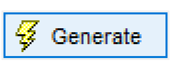
- **X Coordinate [m]:** Enter here the X coordinate for the node in meters.
- **Y Coordinate [m]:** Enter here the Y coordinate for the node in meters.
- **Base Elevation [m]:** Enter here base elevation at the node location. The model only uses the source base elevation if Elevated Terrain is being used. You can specify the source base elevation in either meters [m] or feet [m].
- **Release Height [m]:** Enter the release height above ground in meters [m] at the node location.
- **Release Height [ft]**: Enter the release height above ground in feet [ft] at the node location.

Release Height value must be less than 3000 m (9842 ft).

Available Buttons

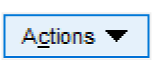


Press this button to have access to the Haul Road Calculator.



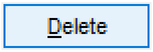
Press the button to re-generate the list of area sources.

Press this button to display the Generated Area Sources dialog where you can view the source parameters for each generated area source.



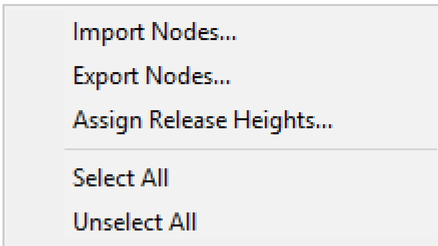
Press this button to have access to several options which are described below.

Press this button to add a new row where you can enter the parameters for a node.



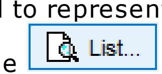
Press this button to delete the selected row/node.

Actions button Menu Options

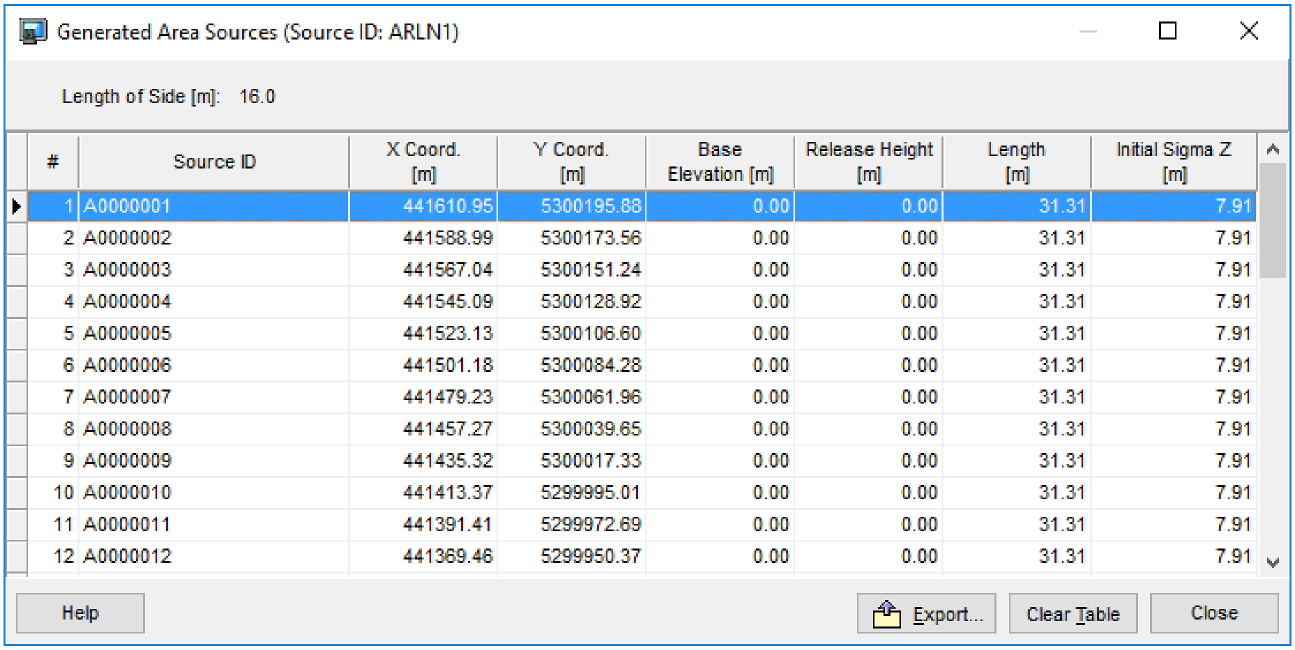


- **Import Nodes:** Allows you to import line source nodes from a CSV file (X,Y,Base_Elevation,Release_Height)
- **Export Notes:** Allows you to export line source nodes to a CSV file (X,Y,Base_Elevation,Release_Height)
- **Assign Release Heights:** Displays the **Assign Release Heights** dialog from where you can specify the Release Height value to be assigned to all selected nodes.
- **Select All:** Automatically selects all nodes/rows from the table.
- **Unselect All:** Automatically unselects all nodes/rows from the table.

Generated Area Sources

The **Generated Area Sources** dialog allows you to view the parameters that are automatically calculated by AERMOD View for the area sources generated to represent the line area source you have defined. You have access to this dialog by pressing the 

button located in the Source Inputs dialog for Line Area Sources.



#	Source ID	X Coord. [m]	Y Coord. [m]	Base Elevation [m]	Release Height [m]	Length [m]	Initial Sigma Z [m]
1	A0000001	441610.95	5300195.88	0.00	0.00	31.31	7.91
2	A0000002	441588.99	5300173.56	0.00	0.00	31.31	7.91
3	A0000003	441567.04	5300151.24	0.00	0.00	31.31	7.91
4	A0000004	441545.09	5300128.92	0.00	0.00	31.31	7.91
5	A0000005	441523.13	5300106.60	0.00	0.00	31.31	7.91
6	A0000006	441501.18	5300084.28	0.00	0.00	31.31	7.91
7	A0000007	441479.23	5300061.96	0.00	0.00	31.31	7.91
8	A0000008	441457.27	5300039.65	0.00	0.00	31.31	7.91
9	A0000009	441435.32	5300017.33	0.00	0.00	31.31	7.91
10	A0000010	441413.37	5299995.01	0.00	0.00	31.31	7.91
11	A0000011	441391.41	5299972.69	0.00	0.00	31.31	7.91
12	A0000012	441369.46	5299950.37	0.00	0.00	31.31	7.91

Generated Area Sources dialog

The following parameters are displayed for each generated area source:

- **Source ID:** Identification name for the generated area source.
- **X Coord [m]:** X coordinate for the source location in meters. This location is for the southwest corner of the area source.
- **Y Coord [m]:** Y coordinate for the source location in meters. This location is for the southwest corner of the area source.
- **Base Elevation:** The source base elevation is displayed in either feet or meters depending on what has been selected in the Source Inputs window. The base elevation is interpolated from the base elevation specified for the two nodes of the same line segment.
- **Release Height [m]:** This is the release height above ground in meters. The release height is interpolated from the release height specified for the two nodes of the same line segment.
- **Initial Sigma Z [m]:** This value corresponds to the **Initial Vertical Dimension** values specified by the user for the **Line Area Source**. This parameter is an optional.

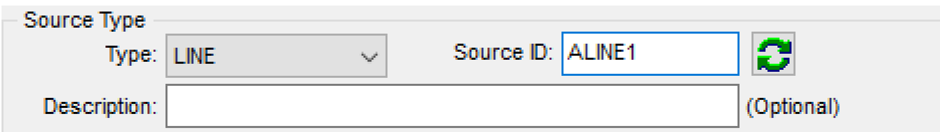
Line Sources

Warning: The Line Source can only be used with US EPA AERMOD model version

12346 or higher. It cannot be used with ISCST3 or ISC-PRIME models.

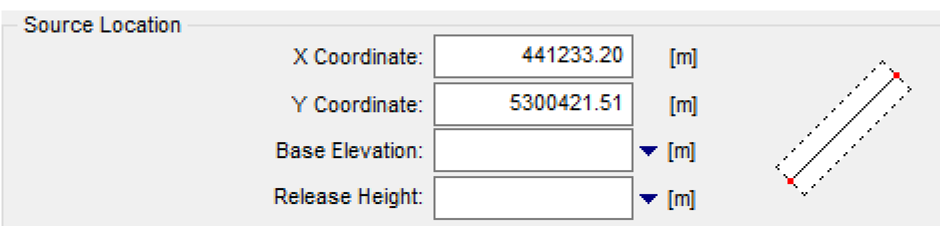
Introduced with US EPA AERMOD version 16216r, AERMOD line sources are very similar to area sources, which are used to model low-level or ground-level releases with no plume rise such as storage piles, slag dumps, and lagoons. Using **AERMOD Line Source** tool you may define a single-segment line-type source based on start-point, end-point, and width. If you wish to define a multi-segment line source, please use [Area Line Source] tool. For the line source, you must provide the following information in the [Source Inputs] dialog:

Source Type



- **Type:** Select LINE from the drop-down list.
- **Source ID:** Enter here an identification name for the source being defined. The ID can be up to 8 characters long (12-characters for AERMOD) and are always in upper case. Press the button to get default Source IDs automatically set up for you.
- **Description:** Enter here any description for the source up to 68 characters long, if you wish.

Source Location



All values entered in English units will be automatically written to the input file in metric units as the U.S. EPA models require that inputs be in metric units (with the exception of base elevation when ELEVUNIT equals FEET).

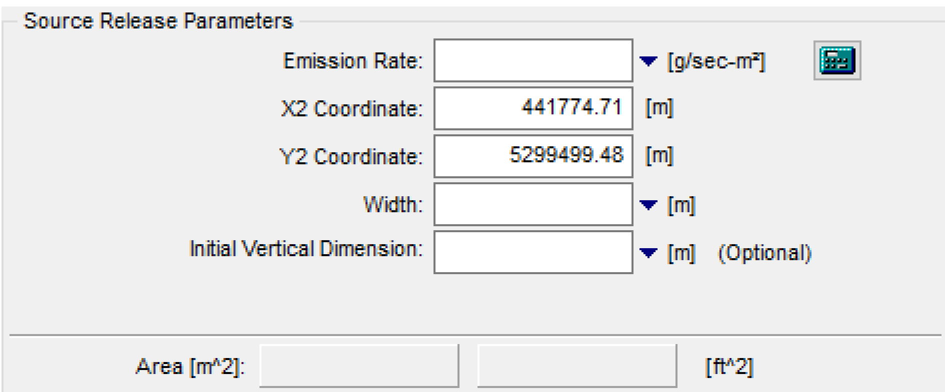
- **X Coordinate [m]:** Enter here the X coordinate for the start point of the line source.
- **Y Coordinate [m]:** Enter here the Y coordinate for the start point of the line source.

- **Base Elevation [m]:** Specify the source base elevation above mean sea level. The model only uses the source base elevation if Elevated Terrain is being used. You can specify the source base elevation in either meters [m] or feet [m].

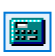
- **Release Height [m]:** Enter the release height above ground in meters [m] or in feet [ft].

> **Release Height** value must be less than 3000 m (9842 ft).

Source Release Parameters



- **Emission Rate [g/(s-m2)]:** Specify here the emission rate of the pollutant. The emission rate for this line source is input as an emission rate per unit area and is assumed to be evenly distributed across the dimensions of the line source.

 Click on this button to display the Auto-Calculate Emissions Tool dialog to calculate the emission rate from grams per second [g/s] to [g/sm^2] or the imperial equivalent.

- **X2 Coordinate [m]:** The X coordinate of the end point of the line source.

- **Y2 Coordinate [m]:** The Y coordinate of the end point of the line source.

- **Width [m]:** Width of the line source. Width must be a minimum of 1 m.

- **Initial Vertical Dimension [m] (Optional):** This optional parameter may be used to specify an initial vertical dimension of the line source plume in meters [m] or in feet [ft]. This parameter is similar to the Initial Vertical Dimension parameter for Volume Sources. This parameter may be important when the line source algorithm is used to model mechanically-generated emission sources, such as mobile sources. In these cases, the emissions may be turbulently mixed near the source by the process that is generating the emissions, and therefore occupy some initial depth. For more passive area source emissions, such as evaporation or wind erosion, the Initial Vertical Dimension parameter may be omitted, which is equivalent to using an initial sigma-z of zero.

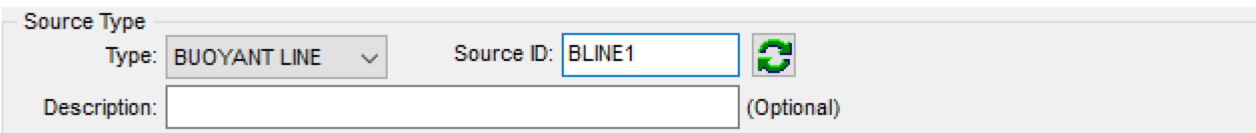
Buoyant Line Sources

A Buoyant Line Source is designed to treat plume rise and dispersion for sources such as roof top vents of a smelting facility. The emissions are calculated using a specialized algorithm based on the Buoyant Line and Point (BLP) Source Dispersion Model (Schulman and Scire, 1980).

> **Warning: Buoyant Line Source** is NOT appropriate for road modeling. To model emissions from roads, use Volume Line Source or Area Line Source tool.

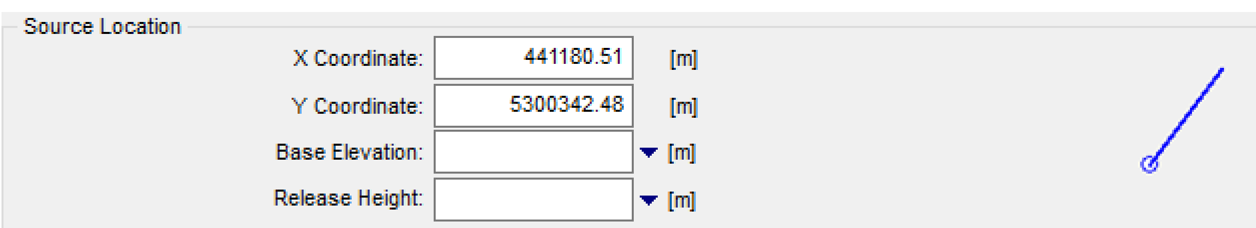
For a **BUOYANT LINE** source, you must provide the following information in the **Source Inputs** dialog:

Source Type



- **Type:** Select **BUOYANT LINE** from the drop-down list.
- **Source ID:** Enter here an identification name for the source being defined. The ID can be up to 8 characters long (12-characters for AERMOD) and are always in upper case. Press the button to get default Source IDs automatically set up for you.
- **Description:** Enter here any description for the source up to 68 characters long, if you wish.

Source Location



> All values entered in English units will be automatically written to the input file in metric units as the U.S. EPA models require that inputs be in metric units (with the exception of base elevation when ELEVUNIT equals FEET).

- **X Coordinate [m]:** Enter here the X coordinate for the start point of the line source.

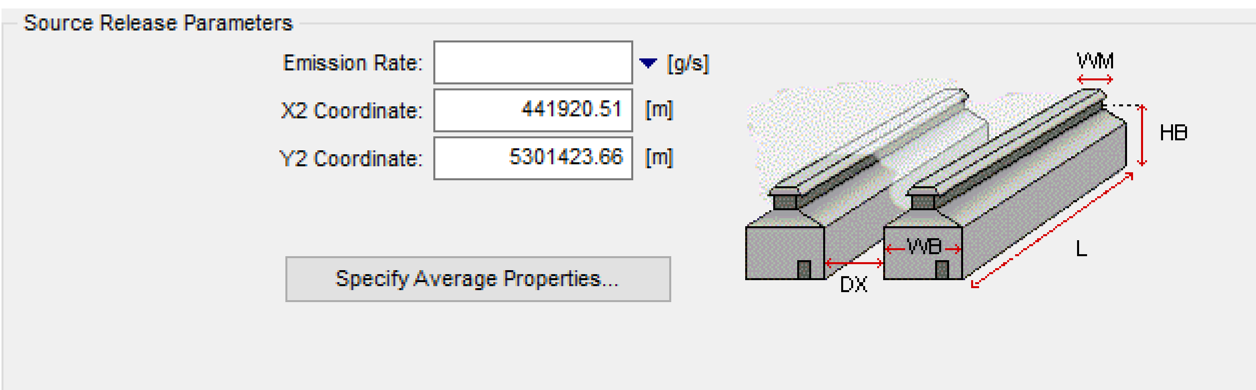
- **Y Coordinate [m]:** Enter here the Y coordinate for the start point of the line source.

- **Base Elevation [m]:** Specify the source base elevation above mean sea level. The model only uses the source base elevation if Elevated Terrain is being used. You can specify the source base elevation in either meters [m] or feet [m].

- **Release Height [m]:** Enter the release height above ground in meters [m] or in feet [ft].

Release Height value must be less than 3000 m (9842 ft).

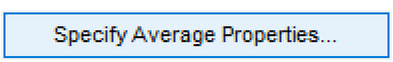
Source Release Parameters



- **Emission Rate [g/s]:** Specify here the emission rate of the pollutant. The emission rate for this line source is assumed to be evenly distributed across the dimensions of the line source.

- **X2 Coordinate [m]:** The X coordinate of the end point of the line source.

- **Y2 Coordinate [m]:** The Y coordinate of the end point of the line source.

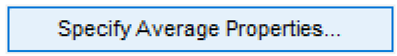
- Click on the 

button to open the Average Properties for Buoyant Line Sources dialog.

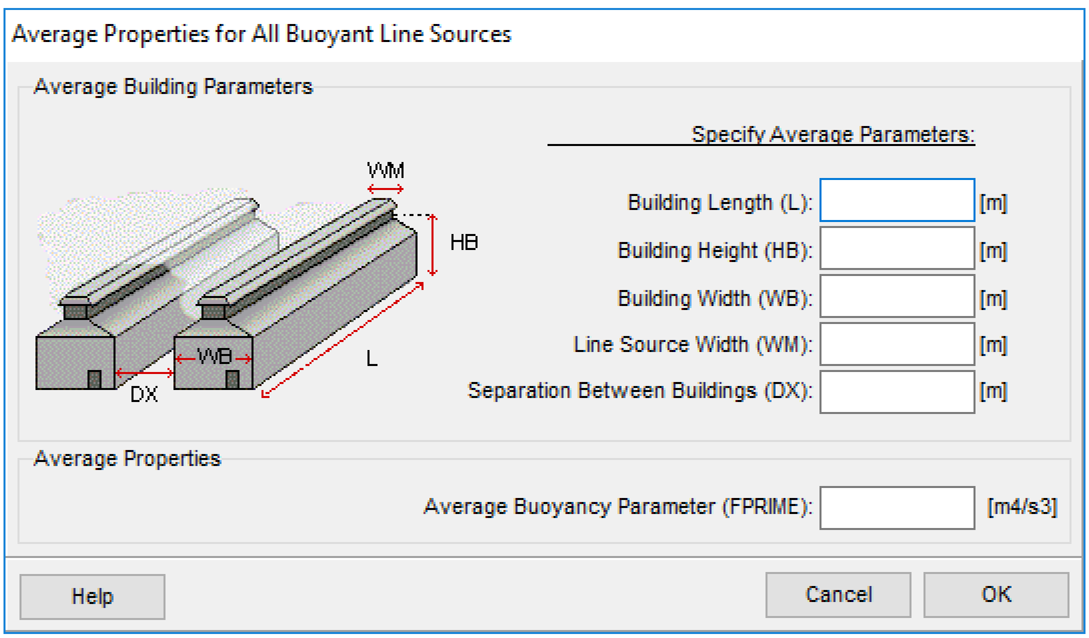
Average Properties for Buoyant Line Sources

For AERMOD model version 15181 or above, specialized algorithms are implemented in order to simulate concentrations from buoyant line sources using techniques from the Buoyant Line and Point (BLP) Source Dispersion Model (Schulman and Scire, 1980). Line source attributes used in BLP define the geometry of one or more long buildings associated with the emissions. The coordinates of the beginning and ending locations of each line are used to determine the points of release, and the orientation of the lines. In addition, the average source attributes are needed for a group of such buildings. These parameters can be defined in the **Average Properties for Buoyant Line Sources** dialog.

> The values entered in this dialog apply to ALL buoyant line sources in the project.

You have access to this dialog by clicking on the 

button in the Buoyant Line Sources dialog.



Average Properties for Buoyant Line Sources dialog

Average Building Parameters

- **Building Length [m]:** Enter the average building length.
- **Building Height [m]:** Specify the average height of the building.
- **Building Width [m]:** Specify the average building width.

- **Line Source Width [m]:** Specify here the line source width in meters.

- **Separation Between Buildings [m]:** Enter the average separation between buildings in meters.

Average Properties

- **Average Buoyancy Parameter (FPRIME) [m**4/s**3]:** Enter a value for the average line source buoyancy parameter used in the plume rise calculations.

According to the Buoyant Line and Point Source (BLP) Dispersion Model User's Guide, the **Average Buoyancy Parameter** can be calculated as follows (page 2-37, equation 2-47):

F' = [g L Wm w (Ts - Ta)]/Ts

where:

F' = average line source buoyancy parameter (m**4/s**3)

g = acceleration of gravity (9.81 m/s**2)

L = average line source length (m)

Wm = average line source width (m)

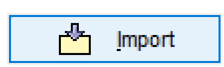
w = exit velocity (m/s)

Ts = exit temperature (K)

Ta = ambient air temperature (K)

> For multiple line sources of comparable buoyancy flux, the buoyancy parameter is calculated for each line source and then averaged.

Import Sources

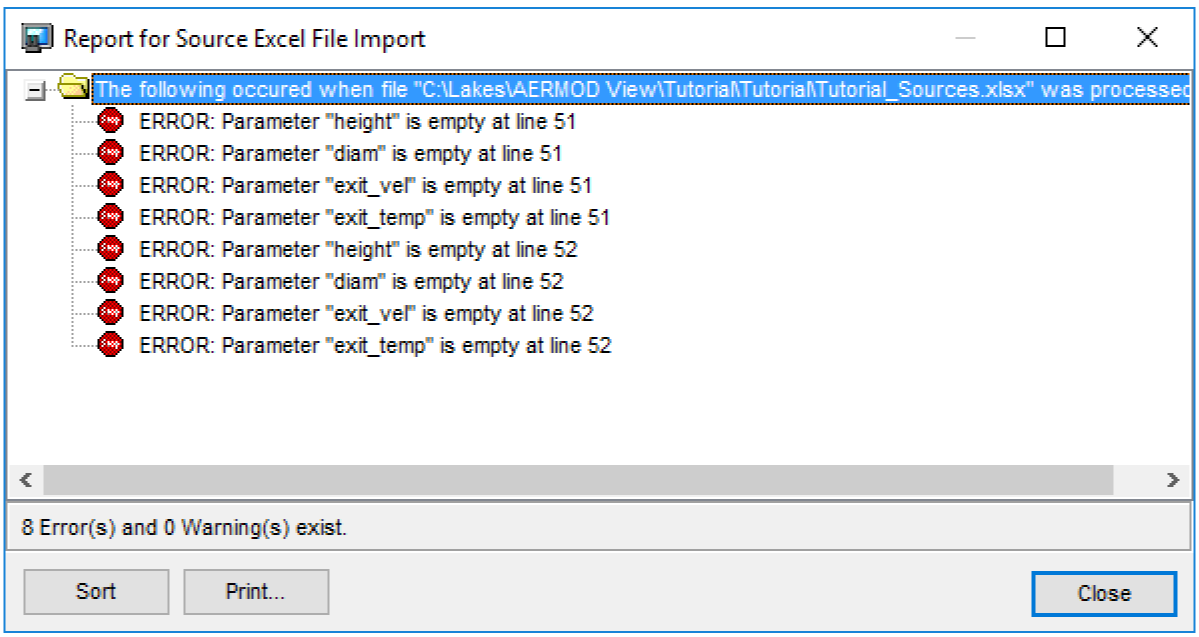
You can import sources into your project by clicking on the 

button in the Source Summary screen or by selecting **Import | Sources...** from the menu. The following file types can be used to import sources:

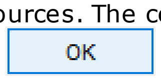
- Input Files

- [Lakes Format](#) (.XLS and .XLSX)
- [SO Pathway Partial Input File](#)
- [BPIP Input Files](#)
- [AERMOD/BPIP projects](#) (You can import sources from other AERMOD View (*.isc) and BPIP View (*.bpv) projects.)
- [Space or Tab Delimited Format](#)
- [Comma Separated Values (CSV) Format](#)

Once you've specified the name and location of the file you wish to import sources from, click on the **Open** button. If any source IDs in the imported file conflict with existing sources in your project, then the [Conflicting Source IDs](#) dialog is displayed. From this dialog you can specify if a source should be renamed or ignored (not imported). If there is data missing from the import file, the following dialog will be displayed:



Import Errors report

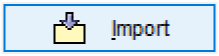
The sources that have missing information will not be imported. Click **Close** button to continue importing valid sources. The confirmation dialog telling you how many sources were imported will appear. Click the 

button to finish the import process.

Import AERMOD/BPIP Project

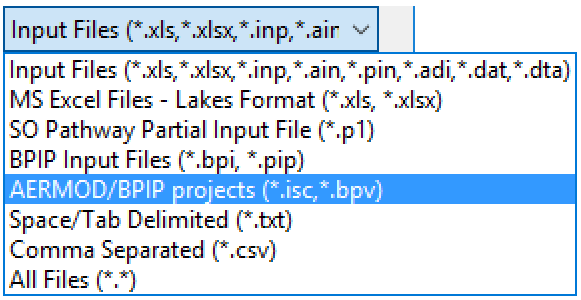
You can import sources directly from any other AERMOD (.isc) View or BPIP (.bpv) project.

To import sources from a project follow the directions below:

1. Click the 

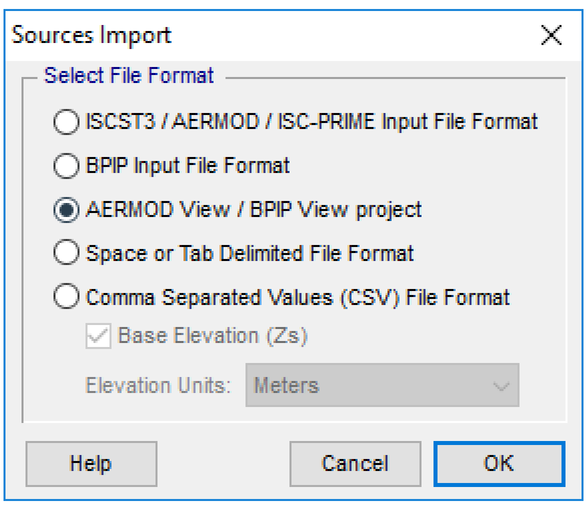
button in the **Source Pathway - Source Summary** dialog. **Import Sources** dialog will open.

2. In the **Import Sources** dialog select **AERMOD/PBIP projects (*.isc,*.bpv)**.



3. Browse to the project that contains the sources you wish to import and click **Open**.

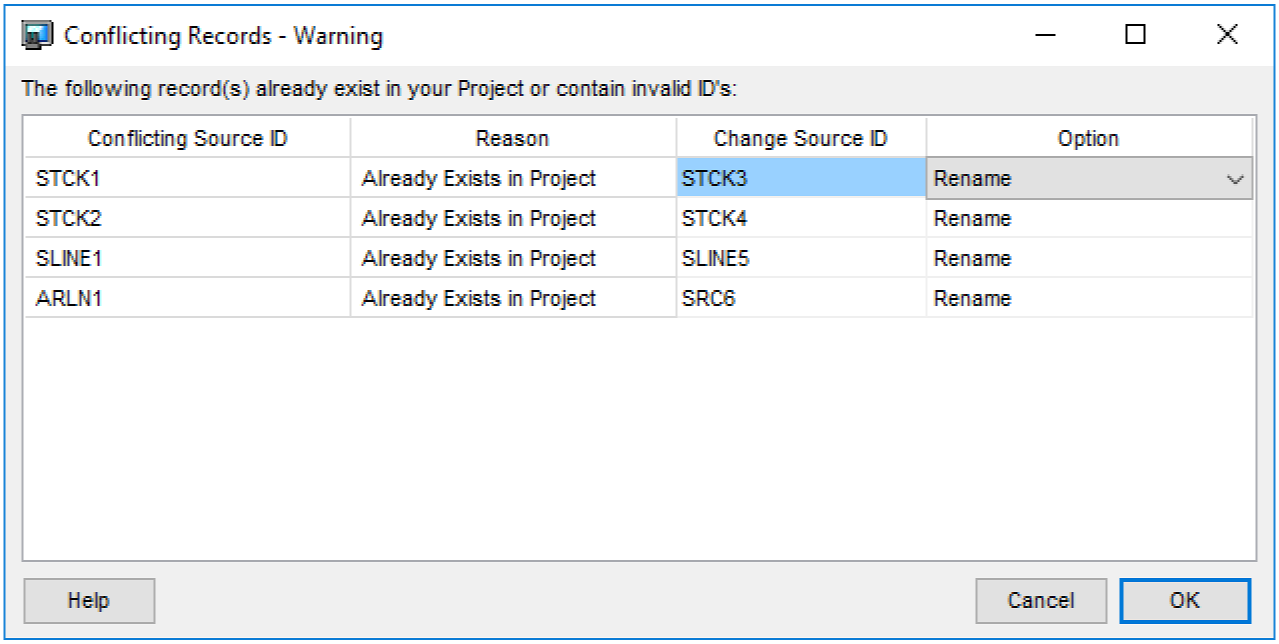
4. Select **AERMOD View/ BPIP View project** as file format in **Sources Import** dialog and click **OK**.



The sources will be added to your current project.

Conflicting Source IDs

When you are importing sources into an existing project, some of the sources you are importing may have source IDs that conflict with the existing source IDs in your project. AERMOD View detects these sources and lists them in the **Conflicting Source IDs** dialog.



Conflicting Source IDs dialog

In the **Change Source ID** column, AERMOD View places default replacement source IDs for each conflicting source. You may change this source ID to another one if you wish. In the **Option** column, you may select from the drop-down list what you wish to do for each conflicting source ID -

- **Rename:** This option will import the source with the alternate source ID specified in the **Change Source ID** column.
- **Do Not Include:** Select this option if you do not wish to import the source.

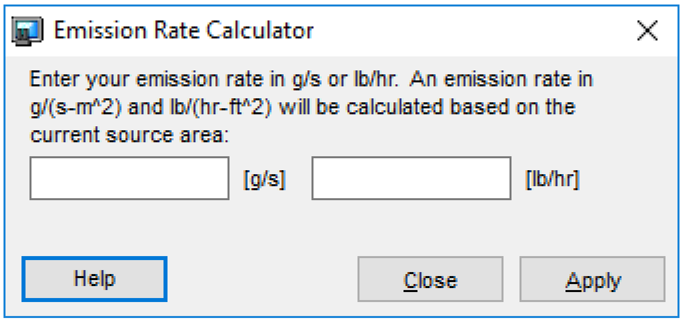
AERMOD View will then import the sources according to the selected options.

Source Pathway Accessory Dialogs

This section describes accessory dialogs used by the Source Pathway.

Auto-Calculate Area Emissions Tool

The **Auto-Calculate Area Emissions** tool allows you to enter an emission rate for Area Sources and Open Pit Sources in grams per second [g/s] or in pounds per hour [lb/hr]. AERMOD View will use that parameter to calculate the emission rate in grams per second per meter squared g/(s-m^2) or in pounds per hour per foot squared lb/(hr-ft^2).



Convert Specified Emission Calculator dialog

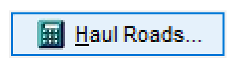
Haul Road Calculator

The **Haul Road Calculator** can be accessed from the following dialogs:

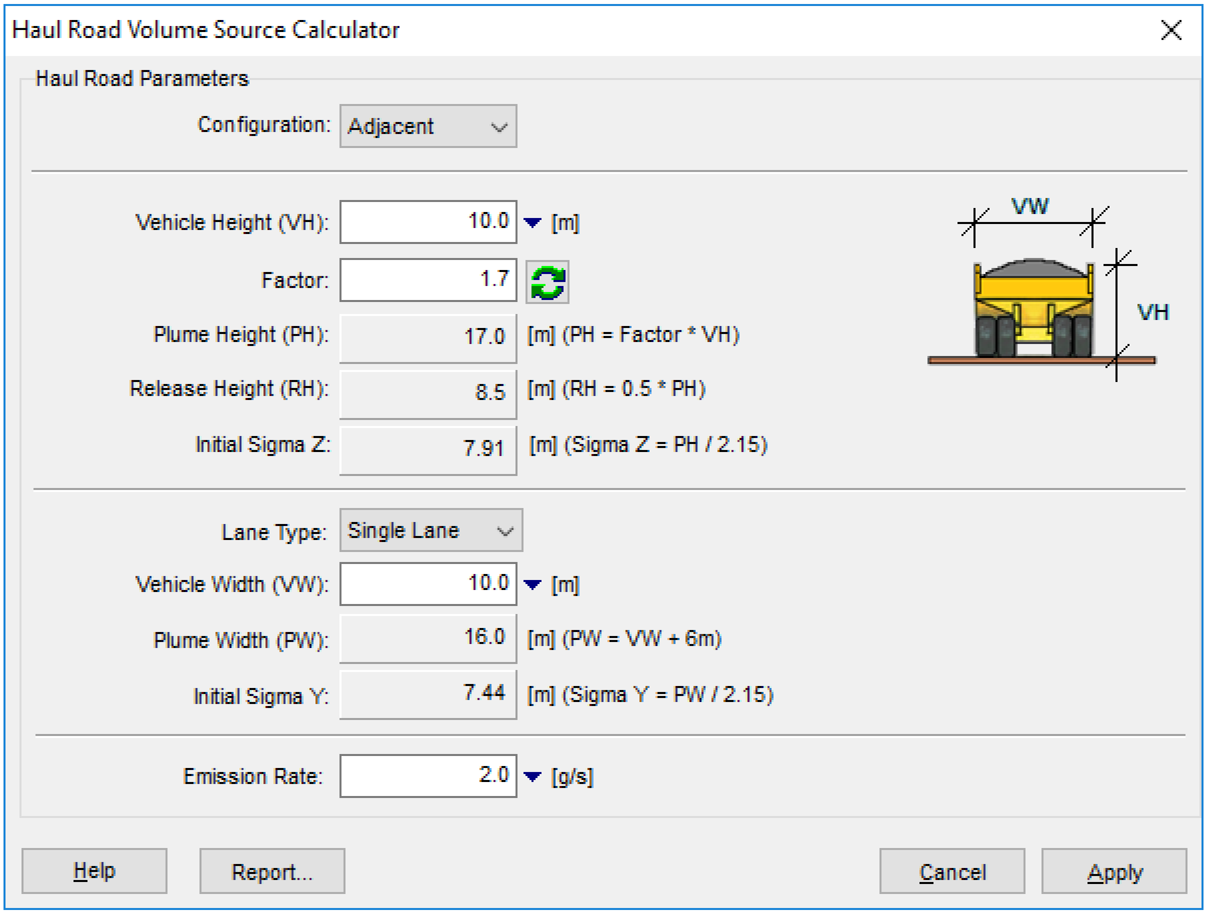
- LINE VOLUME Sources
- LINE AREA Sources

The **Haul Road Calculator** is based on the US EPA Haul Road Workgroup Final Report released on March 2, 2012 which suggests the use of Adjacent Volume Sources to represent the haul road.

Haul Road Volume Source Calculator

The **Haul Road Volume Source Calculator** is displayed when you press the 

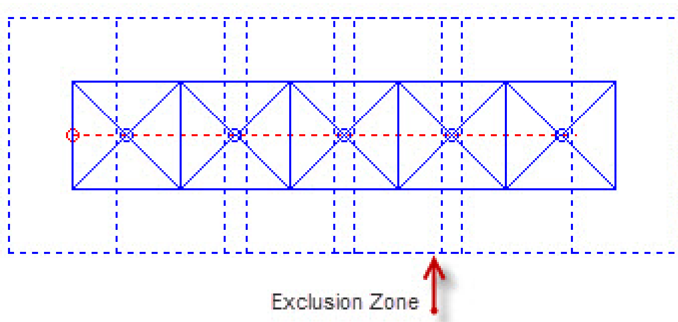
button from the Source Inputs dialog when the source type **LINE VOLUME** is selected.



The **US EPA Haul Road Workgroup Report of March 2, 2012** recommends:

- Adjacent Volume Sources
- Top of Plume Height = 1.7 x Vehicle Height (VH)
- Release Height = 0.5 x Top of Plume Height
- Plume Width (PW) = Vehicle Width (VW) + 6m for single lane roadways or Road Width + 6m for two-lane roadways
- Initial Sigma Z = Top of Plume / 2.15
- Initial Sigma Y = Plume Width (PW) / 2.15
- Two-lane roadways are for cases with heavy two-way traffic where the combined plume needs to be approximated.

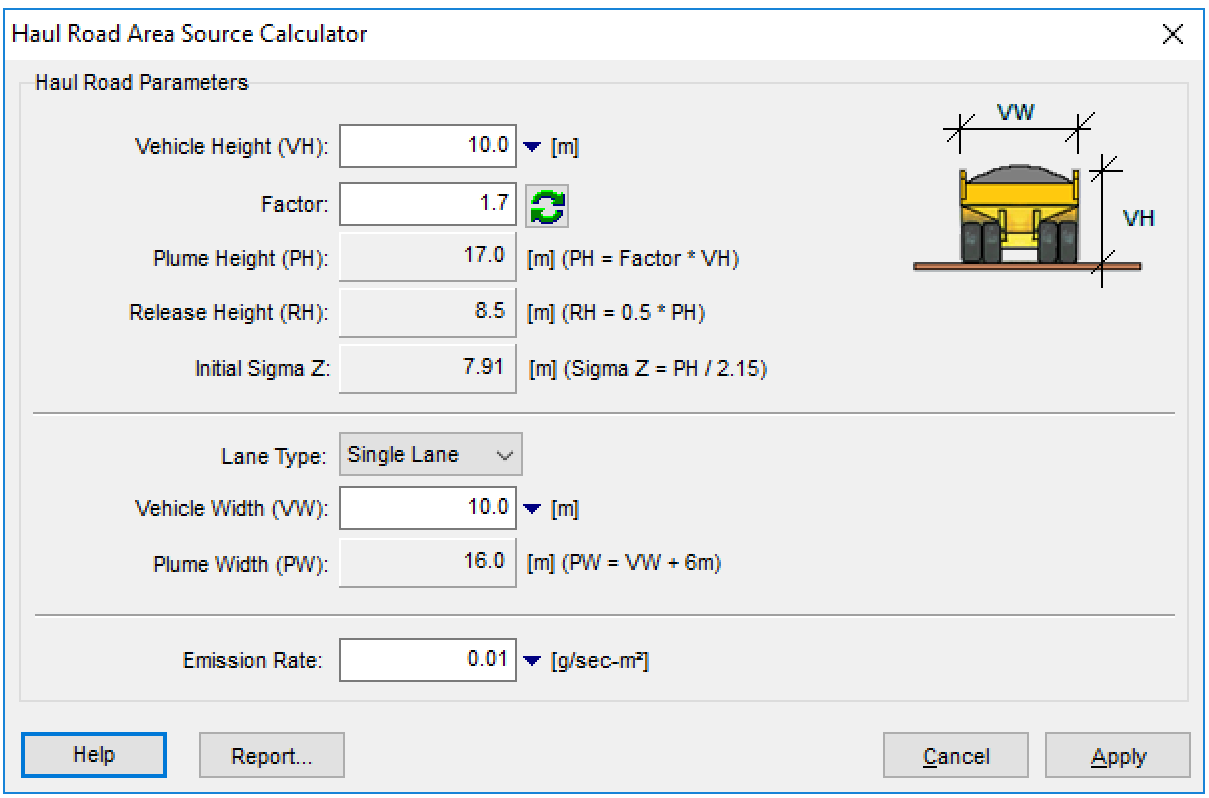
- Do not locate receptors within the Volume source exclusion zone. The exclusion zone is calculate as [(2.15 x Sigma Y) + 1 m]. An example of the exclusion zone for a LINE VOLUME sources can be seen below. See how to visualize the exclusion zone.



Haul Road Area Source Calculator

The **Haul Road Area Source Calculator** is displayed when you press the 

button from the Source Inputs dialog when the source type **LINE AREA** is selected.



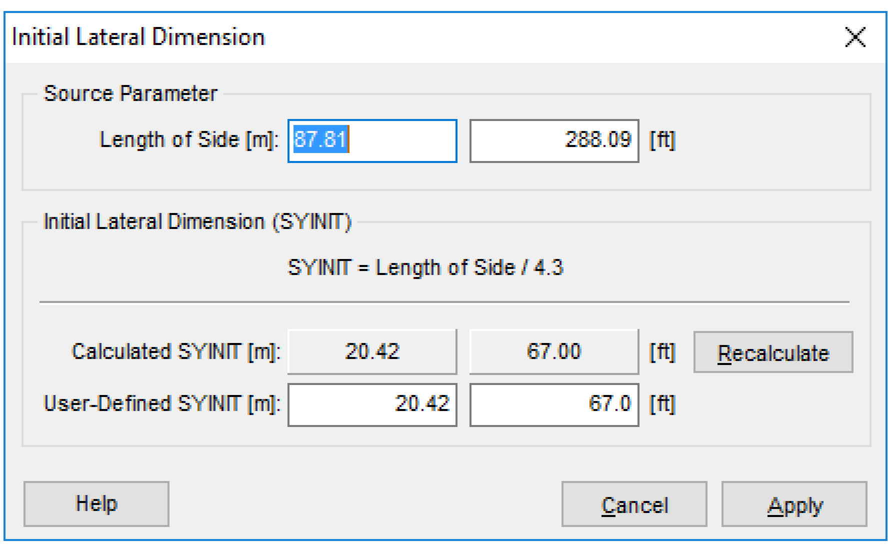
Based on the **US EPA Haul Road Workgroup Report of March 2, 2012**, the LINE AREA Sources configuration is recommend to be used for haul roads **only if receptors are located within LINE VOLUME source exclusion area**.

- Top of Plume Height = 1.7 x Vehicle Height (VH)
- Release Height = 0.5 x Top of Plume Height
- Plume Width (PW) = Vehicle Width (VW) + 6m for single lane roadways or Road Width + 6m for two-lane roadways
- Initial Sigma Z = Top of Plume / 2.15
- Two-lane roadways are for cases with heavy two-way traffic where the combined plume needs to be approximated.

Initial Lateral Dimension

You have access to the **Initial Lateral Dimension** dialog by pressing the Tip... button located in either the Source Inputs dialog for Volume Sources or the Generated Volume Sources dialog.

The **Volume Source - Initial Lateral Dimension** dialog enables you to utilize standard computed values or specify custom values for the initial lateral dimension.



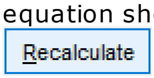
Volume Source - Initial Lateral Dimension dialog

The following options are available:

- **Length of Side:** Here you can view and edit the user-specified **Length of Side** parameter for the volume source. The **Length of Side** parameter can be entered in meters [m] or in feet [ft].

- **Initial Lateral Dimension (SYINIT):** The first section of this panel displays the equation being used to compute the **Initial Lateral Dimension**. The value in the SYINIT (Calculated) field is based on this equation.

  SYINIT = Length of Side/4.3

- **Calculated SYINIT:** This is the computed value based on the **Length of Side** value and the equation shown above. If you change the Length of Side value you must click the 

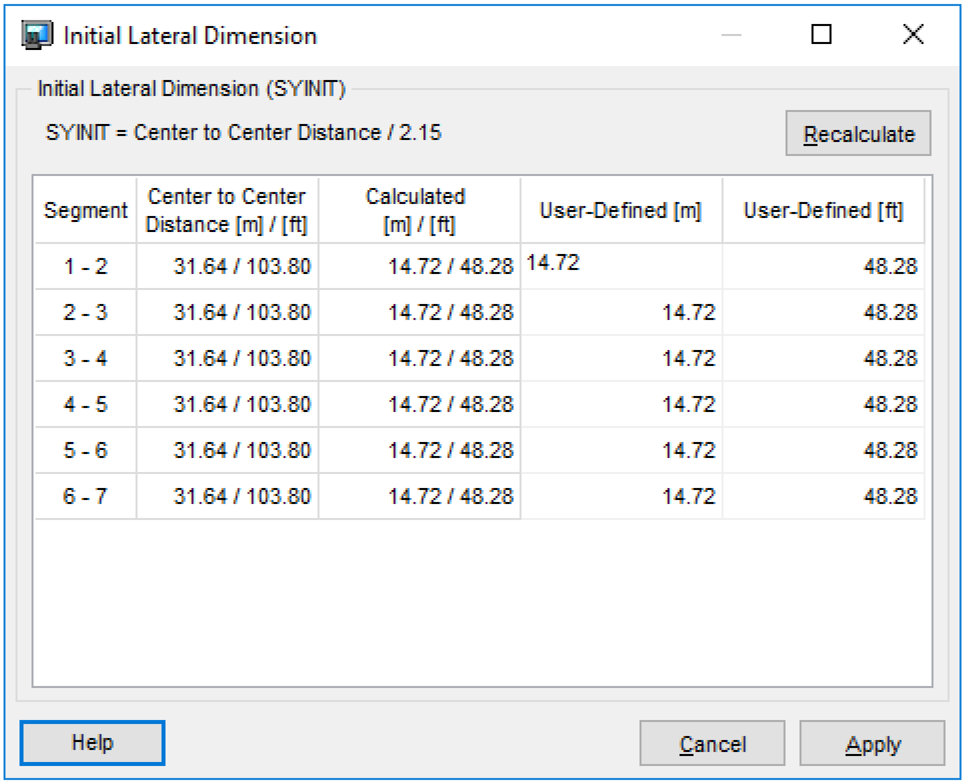
button to update the calculated value. Enter the Calculated SYINIT in meters [m] or in feet [ft].

- **User-Defined SYINIT:** The default value displayed here is the Calculated value. If you wish to use a different value during modeling, you can specify an alternate user-defined value here. Enter the User-Defined SYINIT in meters [m] or in feet [ft].

> **See Also:** Procedures for Obtaining Initial Dimensions for US EPA suggested procedures and equations.

Line Volume Source - Initial Lateral Dimension dialog

Line sources are made up of multiple volume sources, and as a result the **Initial Lateral Dimension** options are similar.



Line Source - Initial Lateral Dimension dialog

The equation on which the auto-calculated values are based is displayed at the top of the dialog. Clicking the Recalculate button will apply this equation to all segments in the table. The following parameters are displayed in the table:

- **Segment:** This is the segment of the line source as defined by 2 nodes. For example, Segment 1-2 is the line segment between nodes 1 and 2.

- **Center to Center Distance [m]/[ft]:** This is the straight-line distance from the center to center of the volume sources created to represent the line source segment. This parameter is displayed in meters [m] and in feet [ft].

- **Calculated [m]/[ft]:** This column contains the calculated values for the initial lateral dimension based on the above equation. This parameter is displayed in meters [m] and in feet [ft].

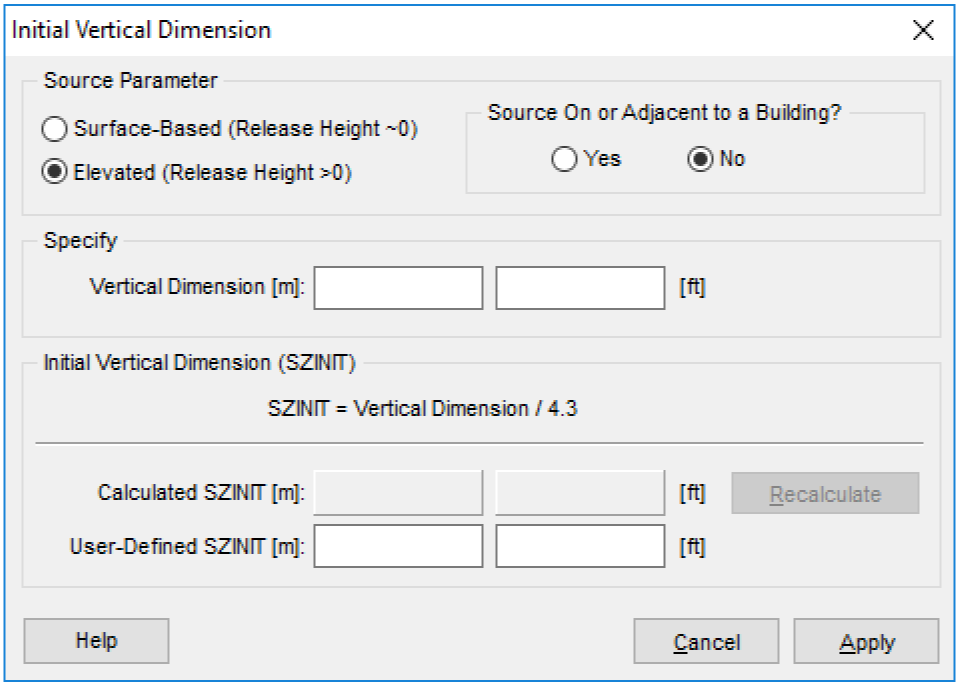
- **User-Defined:** The default value displayed in this column is the Calculated value. If you wish to use a different value during modeling, you can specify an alternate user-defined value here. This User-Defined value is displayed in meters [m] and in feet [ft].

See Also: Procedures for Obtaining Initial Dimensions for US EPA suggested procedures and equations.

Initial Vertical Dimension

You have access to the **Initial Vertical Dimension** dialog by pressing the 

button located in either the Source Inputs dialog for Volume Sources or the Generated Volume Sources dialog.



Initial Vertical Dimension dialog

The Initial Vertical Dimension dialog enables you to utilize standard computed values or specify custom values for the initial vertical dimension. The following options are available:

- **Source Parameter:** Here you can specify whether the source is Surface-Based or Elevated. This will affect the equation used to compute the Initial Vertical Dimension.

- **Surface-Based (He ~ 0):** Select this option if the effective emission height (release height above ground) is approximately zero. An example of a surface-based volume source is a surface rail line. See below for details of the procedure used to estimate the Initial Vertical Dimension of the source, if this option is selected.

  Initial Vertical Dimension = Vertical Dimension of Source / 2.15

- **Elevated (He > 0):** Select this option if the effective emission height (Release Height above Ground) is greater than zero. An example of an elevated line/volume source is an elevated rail line. See below the procedure used to estimate the Initial Vertical Dimension of the source, if this option is selected.

If **On or Adjacent to a Building** then

Initial Vertical Dimension = Building Height / 2.15

If NOT **On or Adjacent to a Building** then

Initial Vertical Dimension = Vertical Dimension of the Source / 4.3

- **Specify:** Here you can view and edit the user-specified Vertical Dimension of the Source parameter for the volume source. If the source is adjacent to a building, then the Building Height parameter can be defined. The Vertical Dimension can be specified in meters [m] or in feet [ft].

- **Initial Vertical Dimension (SZINIT):** The first section of this panel displays the equation being used to compute the Initial Vertical Dimension. The value in the SZINIT (Calculated) field is based on one of the equations, depending on you selections in the Source Parameters panel -

  SZINIT = Vertical Dimension/4.3

  OR

  SZINIT = Vertical Dimension/2.15

  OR

  SZINIT = Building Height / 2.15

- **Calculated SZINIT:** This is the computed value based on the Vertical Dimension of the Source or Building Height and the equation shown above. If you change either value you must click the Recalculate button to update the calculated value. Enter the Calculated SZINIT in meters [m] or in feet [ft].

- **User-Defined SZINIT:** The default value displayed here is the Calculated value. If you wish to use a different value during modeling, you can specify an alternate user-defined value here. Enter the User-Defined SZINIT in meters [m] or in feet [ft].

See Also: Procedures for Obtaining Initial Dimensions for US EPA suggested procedures and equations.

Procedures for Obtaining Initial Dimension

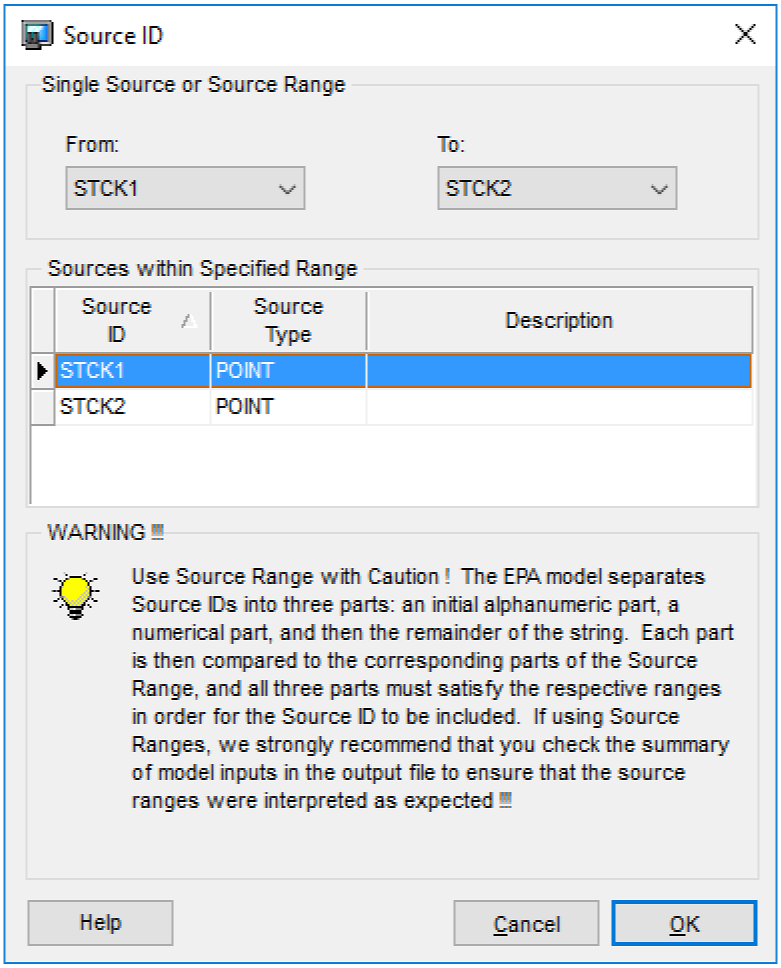
The following tables describes the summary of suggested procedures for estimating initial lateral dimension (syo) and initial vertical dimension (szo) for Volume and Line Sources (U.S. EPA 1995e)

Initial Lateral Dimension for:	Procedure for Obtaining Initial Dimension
Single Volume Source	$s_{yo} = \frac{\text{side length}}{4.3}$
Line Source Represented by Adjacent Volume Sources	$s_{yo} = \frac{\text{side length}}{2.15}$
Line Source Represented by Separated Volume Sources	$s_{yo} = \frac{\text{center to center distance}}{2.15}$

Initial Vertical Dimension for:	Procedure for Obtaining Initial Dimension
Surface-Based Source (he ~ 0)	$s_{zo} = \frac{\text{vertical dimension of source}}{2.15}$
Elevated Source (he > 0) on or Adjacent to a Building	$s_{zo} = \frac{\text{building height}}{2.15}$
Elevated Source (he > 0) not on or Adjacent to a Building	$s_{zo} = \frac{\text{vertical dimension of source}}{4.3}$

Source ID (Range)

The **Source ID** dialog can be accessed from several screens in the Source Pathway dialog. From the **Source ID** dialog, specify the source or the range of sources you wish to use.



Source ID dialog

The **Source ID** dialog has the following fields:

- **From:** Select the source you want to include from the drop-down list.
- **To:** From this drop-down list, you can select:
- **(None)** - this specifies a single source; only the source listed in the **From** drop-down list will display in the **Sources within specified Range** table.
- any source other than the source selected in the **From** drop-down list - this specifies a source range; all sources in the range will display in the Sources within Specified Range table.

- **Sources within Specified Range:** This table displays a list of all the sources included within the source range that you specified using the **To/From** drop-down lists.

> The US EPA models interpret source ranges using a special method (see link below). Therefore, we recommend that you check the Sources within Specified Range table to see if sources considered within your specified source range are what you expected.

> See Also: How EPA Interprets Source Ranges.

How EPA Interprets Source Ranges

When comparing a source ID to the range limits, the model separates the source IDs into three parts:

Part 1: an initial alphabetical part

Part 2: a numerical part

Part 3: the remainder of the string

Each part is then compared to the corresponding parts of the source range, and all three parts must satisfy the respective ranges in order for the source ID to be included.

If there is no numeric part, then the ID consists of only one alphabetical part. If the ID begins with a numeric character, then the initial alphabetical part defaults to a single blank. If there is no trailing alphabetical part, then the third part also defaults to a single blank part. If the trailing part consists of more than one alphabetical or numeric field, it is all lumped into one character field.

Two-Part Example:

The source ID 'STACK2' consists of the parts '**STACK**' + '**2**' + ' ' (a single blank). By comparing the separate parts of the source IDs, it can be seen that **STACK2** falls between the range '**STACK1**-**STACK10**'

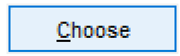
Three-Part Example:

VENT1B falls within the range of **VENT1A**-**VENT1C**. However, **VENT2** does not fall within the range of **VENT1A** to **VENT3C**, since the third part of **VENT2** is a single blank, which does not fall within the range of **A** to **C**. This is because a blank character will precede a normal alphabetical character.

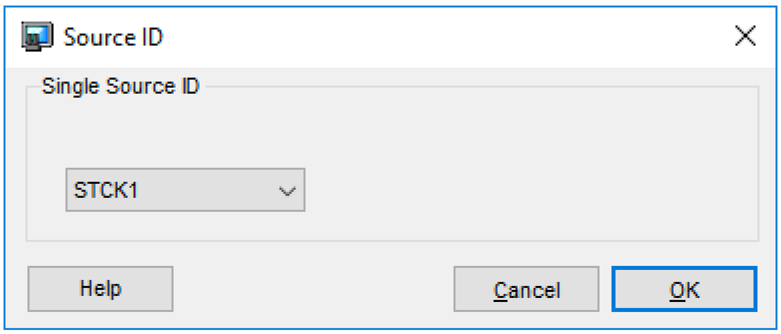
Normally, the source ranges will work as one would intuitively expect for simple source names. Most importantly, for names that are made up entirely of numeric characters, the source ranges will be based simply on the relative numerical values.

> The user is strongly encouraged to check the summary of model inputs to ensure that the source ranges were interpreted as expected, and also to avoid using complex source names in ranges, such as AA1B2C-AB3A3C.

Source ID

The **Source ID** dialog is displayed by clicking on the 

button in the **Source Pathway**. Here you may select the source you wish to use.



Source ID dialog

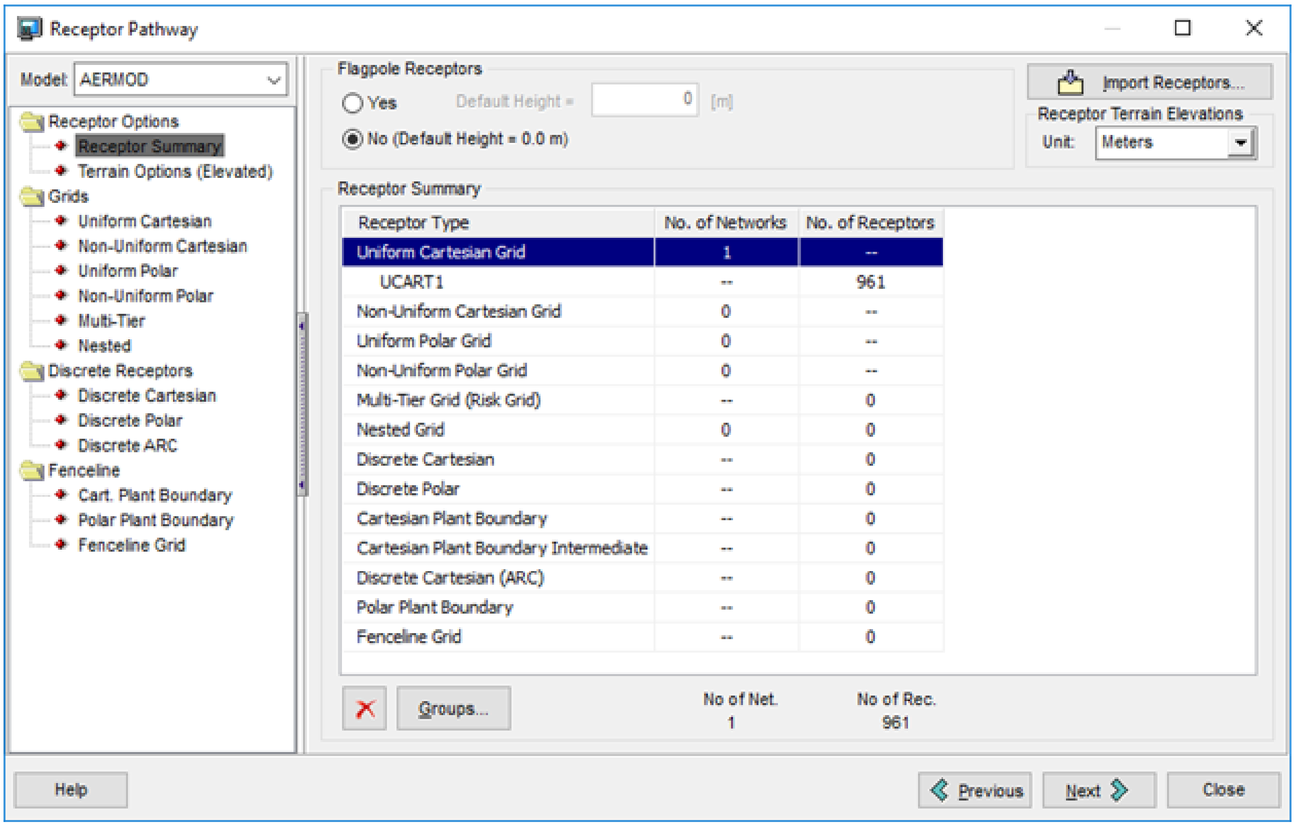
Select the source you wish to use from the drop-down list and click **OK** when finished.

Receptor Pathway

The **Receptor Pathway** allows you to specify the receptor locations for a particular run, define the number and type of receptors in your project, delete selected receptors, define receptor groups, and flagpole options. You have access to the **Receptor Pathway** dialog by select **Data | Receptor Pathway...** from the menu or by clicking on the **Receptor** 

menu toolbar button.

The **Receptor Pathway** dialog uses a two-pane view. The tree located on the left side of the dialog is used for navigation and item selection. Select an item (marked as ✦) in the tree to display the available options on the right panel.



Receptor Pathway dialog

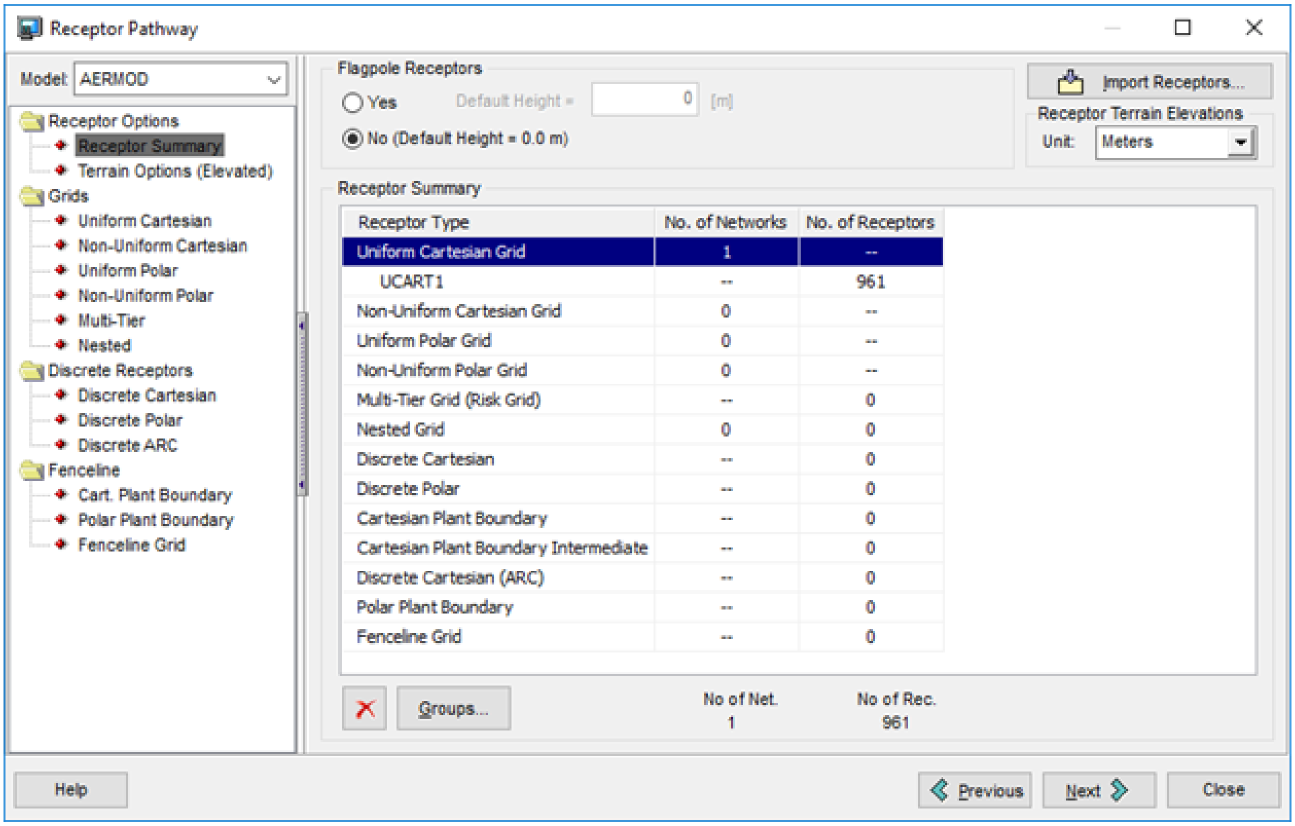
In the **Receptor Pathway** you can define the following:

- Receptor Summary
- Terrain Options
- Uniform Cartesian Grid

- [Non-Uniform Cartesian Grid]
- [Uniform Polar Grid]
- [Non-Uniform Polar Grid]
- [Multi-Tier Grid]
- [Nested Grid]
- [Discrete Cartesian Receptors]
- [Discrete Polar Receptors]
- [Discrete ARC Receptors]
- [Cartesian Plant Boundary]
- [Polar Plant Boundary]
- [Fenceline Grid]
- [Terrain Grid Options]

Receptor Summary

In the **Receptor Summary** page, you can view a summary of the type of receptors, number of grids, and total number of receptors already specified for the current project. Select **Receptor Pathway - Receptor Summary** from the tree located on the left side of the Receptor Pathway dialog, to display the available options.



Receptor Pathway dialog - Receptor Summary

Flagpole Receptors

In this panel you can select whether or not you wish to use flagpole receptors. A flagpole receptor is defined as any receptor located above ground level and can be used, for example, to represent the roof or balcony of a building. Two options are available:

- **No:** Select this option if no flagpole receptors are to be considered. This is the default option, and assumes a default flagpole receptor height of 0.0 m (i.e., ground-level receptors). Any flagpole heights that are entered in the Receptor Pathway are ignored if the No option is selected.

- **Yes:** This option indicates that flagpole receptors are to be used. When this option is selected, a text box is displayed where you can input a default flagpole height (height above ground level) to be applied to any unspecified flagpole heights at each receptor location. If the default height text filed is left blank, a default value of 0.0 meters will be used.

Receptor Terrain Elevation Units

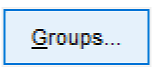
If you select **Elevated** for the terrain height option in the Terrain Options window, then you have the choice of selecting the units for the receptor terrain elevations. Select the elevation units from the drop down list, either meters or feet.

Click on this button to Import Receptors into your project.

Receptor Summary

You can select a particular receptor type from the **Receptor Summary Table**; double-click on it and the screen for that particular receptor type will be displayed. The following buttons are also found in this window:


Click on this button to delete selected receptors from the **Receptor Summary Table**. Click with the left mouse button on the row for the receptor you want to delete and press this button. For multiple selections in sequence press the Shift key while selecting the receptors or press the Ctrl key to make disjoint selections.

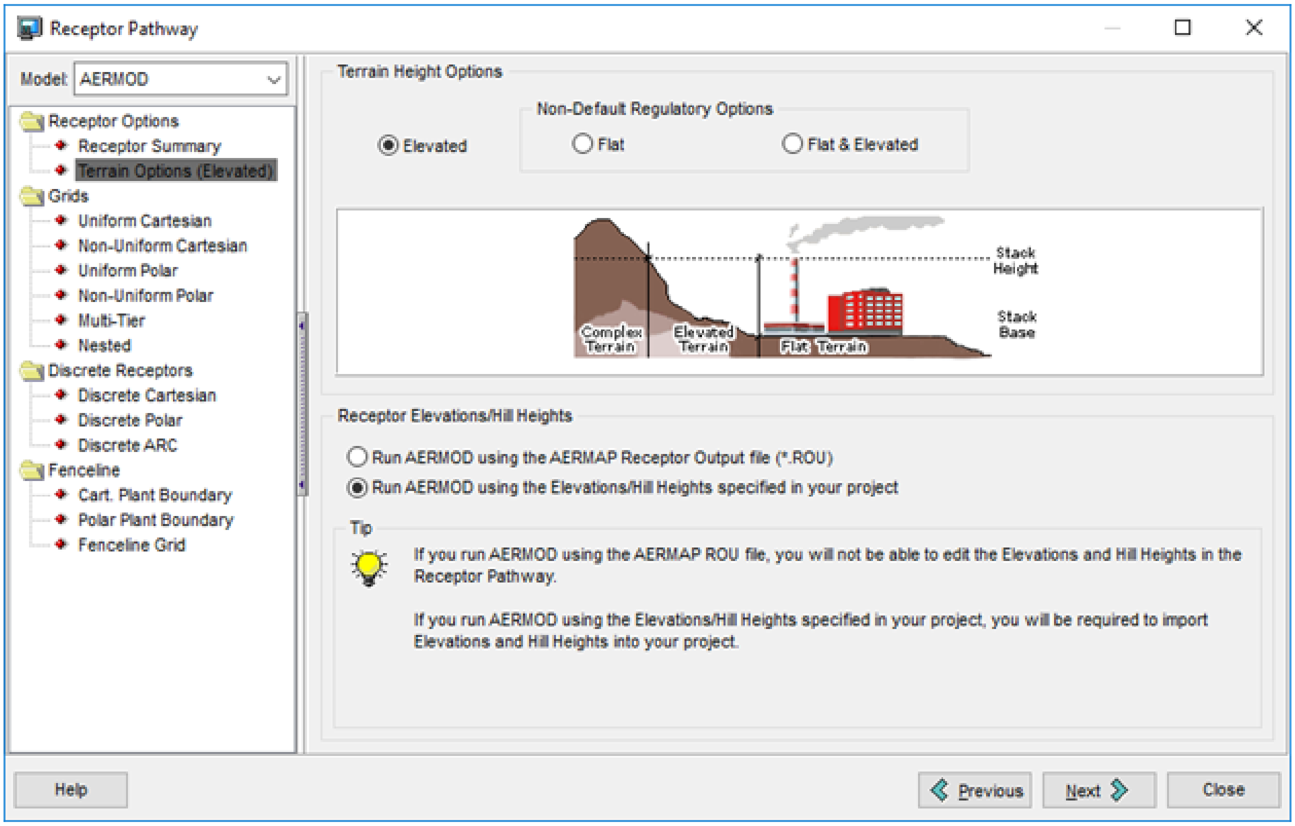


Click on this button to display the Receptor Groups dialog from where you can define receptor groups.

Terrain Options

In the **Terrain Options** page, you can specify whether you will be modeling the effects of terrain above stack base, in order to be able to input elevated receptor heights and also specify receptor elevations above ground level to model flagpole receptors. Select **Receptor Pathway - Terrain Options** from the tree located on the left side of the Receptor Pathway dialog, to display the available terrain height and flagpole receptor options.

AERMOD Terrain Options

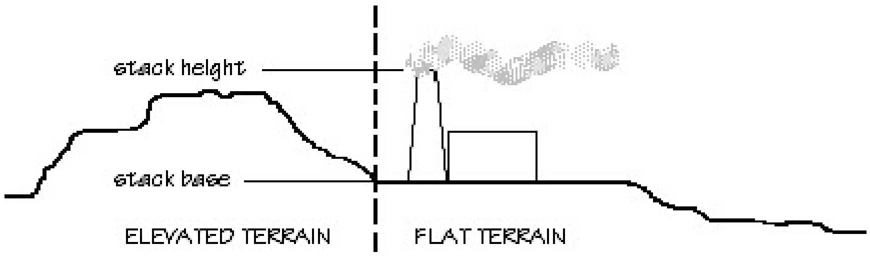


Receptor Pathway dialog - Terrain Options - AERMOD

You may select from two terrain height options:

- **Flat:** This option assumes that the terrain height does not exceed stack base elevation, that is, terrain height assumed to be 0.0 m. All receptors are assumed to be at the same elevation as the base elevation for the source as the default mode of operation. Flat Terrain calculations are used throughout, and any terrain elevations entered in the Receptor Pathway will be ignored.

- **Elevated:** This options assumes terrain height exceeds stack base elevation. Select this option if you wish to model receptors on elevated terrain. You must then enter terrain elevations for your receptors in the Receptor Pathway. If elevated terrain is selected and receptor heights are not specified, then it is assumed to have a value of 0.0 meters.

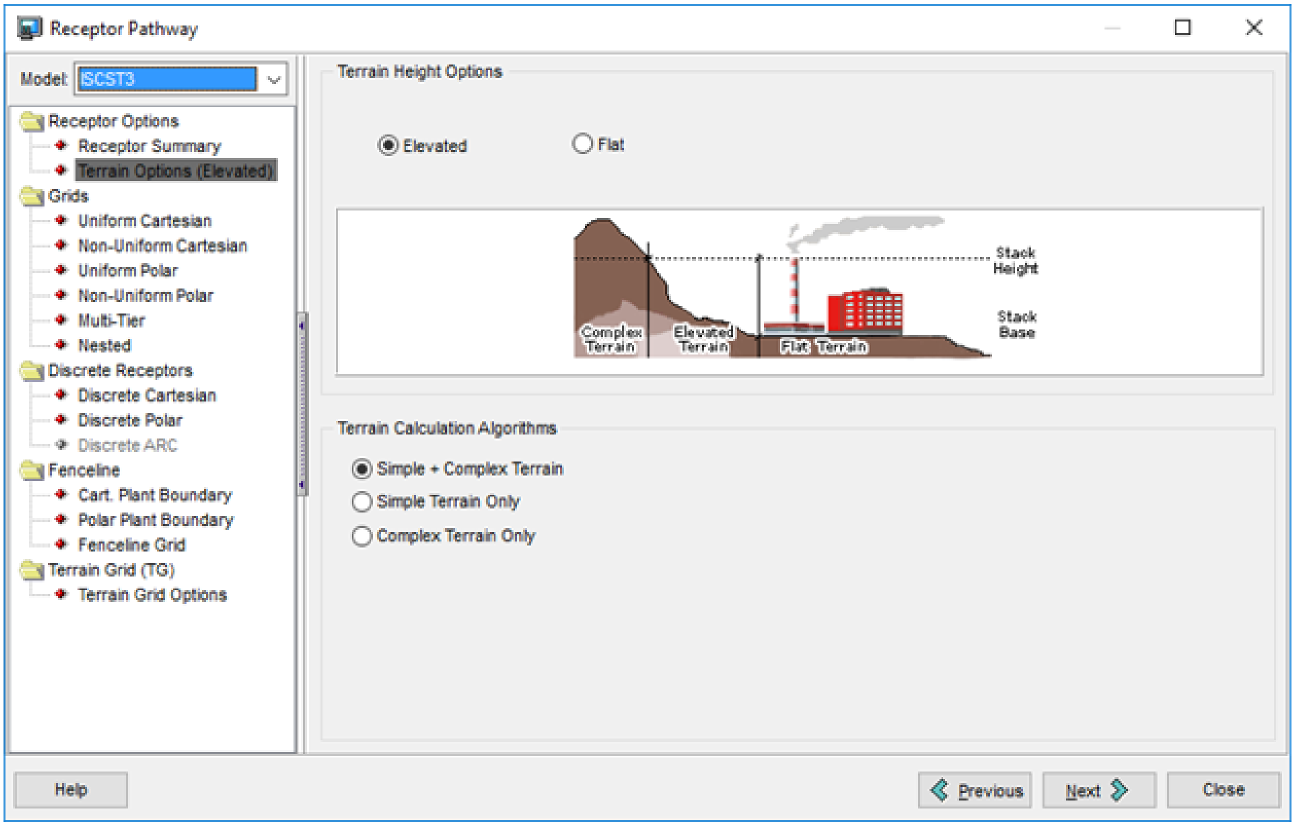


Receptor Elevations/Hill Heights - AERMOD Only

In this panel you can select whether or not you wish to use the AERMAP ROU file.

- **Run AERMOD using the AERMAP Receptor Output file (*.ROU):** With this option, AERMOD View reads the receptors elevations directly from the *.ROU file generated once AERMAP has run. You cannot modify receptor elevations within the interface using this option.
- **Run AERMOD using the Elevations/Hill Heights specified in your project:** With this option, AERMOD View copies the receptors elevations directly from the *.ROU file into the project input file. You will then be able to modify receptor elevations directly within the interface.

ISCST3/ISC-PRIME Terrain Options



Receptor Pathway dialog - Terrain Options - ISC

Terrain Calculation Algorithms - ISCST3 and ISC-PRIME Only

In this panel you may select one of the following terrain calculation algorithms:

- **Simple + Complex Terrain:** This is the default option where the model implements both simple and complex terrain algorithms and also applies intermediate terrain processing. In this case, the model will select the higher of the simple and complex terrain calculations on an hour-by-hour, source-by-source and receptor-by-receptor basis for receptors in intermediate terrain, that is, terrain between release height and plume height.

- **Simple Terrain Only:** This option specifies that no complex terrain calculations will be made and only ISCST algorithms are used. You should not use this option if you are modeling with complex terrain (terrain above the release height of the source).

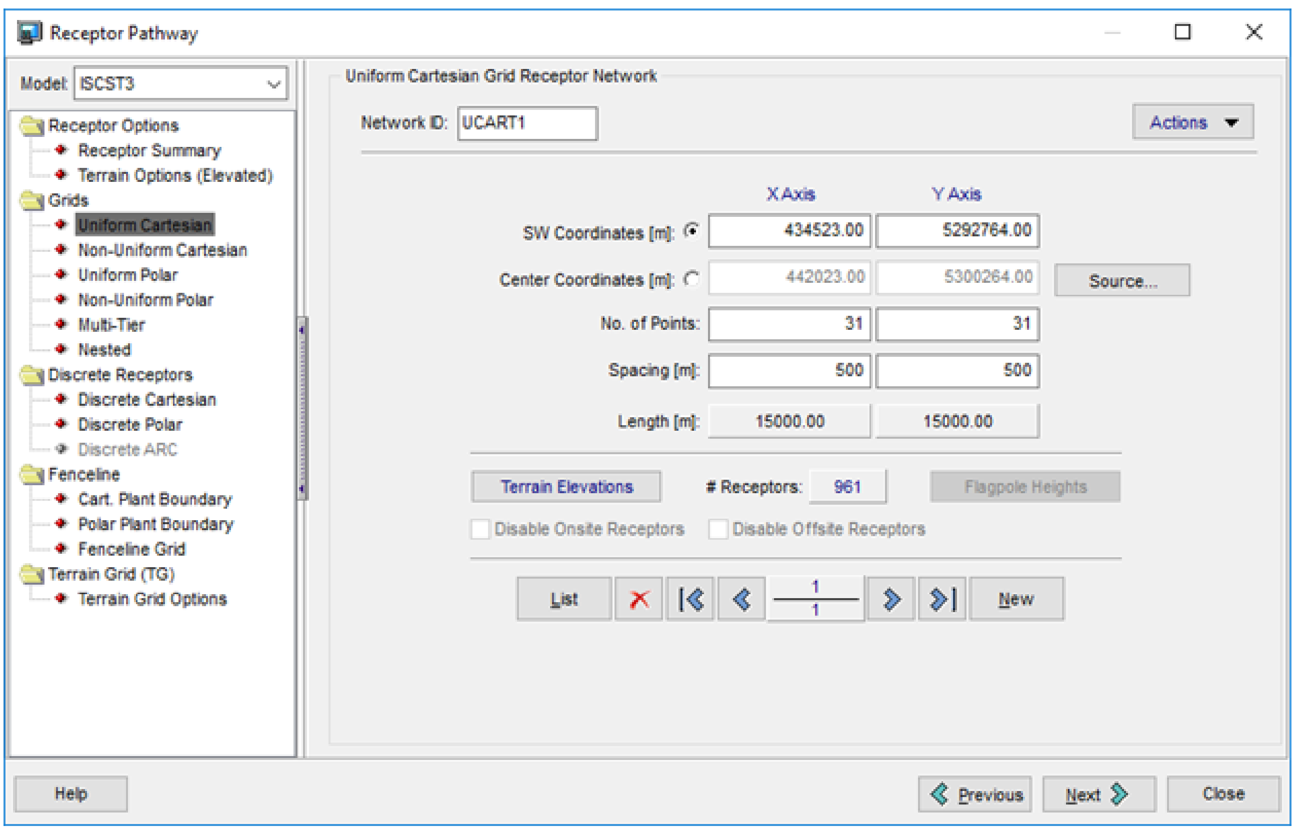
- **Complex Terrain Only:** This option is only available for the Elevated terrain height option. This option specifies that no simple terrain calculations will be made, and only COMPLEX1 algorithms are used.

For terrain above the release height of the source, the model automatically truncates or chops the terrain to the physical release height when modeling impacts at those receptors using the simple terrain algorithm (Simple Terrain Only option). Terrain above the release height is not truncated when the COMPLEX1 algorithm is used in ISCST3 (Complex Terrain Only option).

There is no distinction in AERMOD between elevated terrain below release height and terrain above release height, as with ISCST3 and ISC-PRIME that distinguish between simple terrain and complex terrain.

Uniform Cartesian Grid

In the **Uniform Cartesian Grid** page, you can define Cartesian grid receptor networks with uniform grid spacing. Select **Grids - Uniform Cartesian** from the tree located on the left side of the Receptor Pathway dialog, to display the available options.



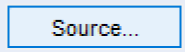
Receptor Pathway dialog - Uniform Cartesian Grid

The following parameters are necessary to define a Uniform Cartesian Grid Receptor Network:

- **Network ID:** This is the ID for the uniform Cartesian grid and can be up to eight alphanumeric characters. AERMOD View automatically creates default IDs (UCART1, UCART2, etc.) but these IDs can be changed at any time.

- **SW Coordinates:** Select this radio button and specify the X (east-west) and Y (north-south) coordinates of the grid origin (Southwest corner) in meters.

- **Center Coordinates:** Select this radio button and specify the X (east-west) and Y (north-south) coordinates of the grid origin (center) in meters.

To select a source as the center of the grid, click the 

button and select the source from the drop-down list.

- **No. of Points (Px, Py):** This is the number of points on the X-axis and Y-axis.

- **Spacing (Dx, Dy):** This is the spacing, in meters, between X-axis receptors and between Y-axis receptors.

- **Length:** The final dimensions of the grid in the X direction and Y direction are automatically calculated by AERMOD View and displayed here.

If you have graphically defined the uniform Cartesian grid using the Uniform Cartesian Grid Tool, the above fields will be automatically filled. You may modify the values if you wish.

- **# Receptors:** This field displays the total number of receptors defined for the current grid, and is automatically filled. Receptor are defined at each grid node.

The following buttons are available:

Click this button to select from two options:

- **Convert to Discrete** - converts the current grid into discrete Cartesian receptors. This option is used when there is a need to eliminate one or more receptors from the receptor network grid, for example, eliminate receptors within a plant boundary.

- **Export to CSV File** - exports the location of each grid node as discrete receptors, with each receptor's parameters, such as X coordinate (x), Y coordinate (y), terrain elevations (ELEV), hill heights (HILL), and flagpole heights (FLAG) to a file.

> See Also: Ordered Discrete Receptors for more information on the methodology for how receptors are ordered when converted from a grid to discrete.

> See Also: Receptor Import/Export Format.

Terrain Elevations Press this button to open the Receptor Terrain Elevations dialog where you can specify terrain elevations for each receptor. This button is available only is you have selected the **Elevated** terrain option in the Terrain Options page. If no terrain elevations are defined when you are modeling with elevated terrain, then the elevations will default to 0.0 meters.

Flagpole Heights Press this button to open the Flagpole Heights dialog where you can specify flagpole heights for each receptor. This is the receptor height above ground in meters. This button is available only if you have selected Yes for the Flagpole Receptors option in the Receptor Summary page.

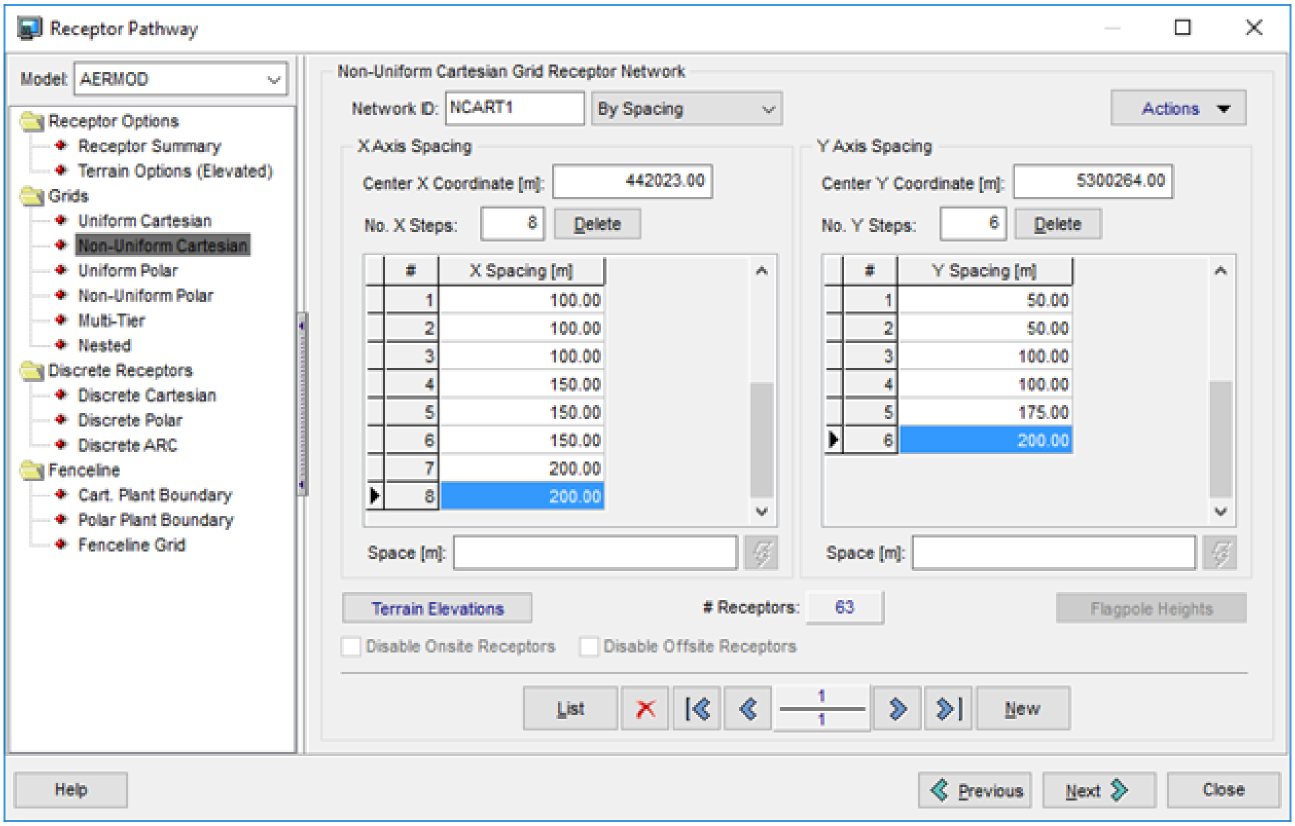
If no flagpole receptor heights are defined when you are modeling with Flagpole Receptors, then the flagpole receptor heights will default to the Default Height value. If no default height was specified, then the flagpole height will default to 0.0 meters.

The **Disable Onsite Receptors** / **Disable Offsite Receptors** option allows the interface to either automatically disable any receptors inside the boundary (onsite) or outside the boundary (offsite) where the gridded receptor network intersects with a Plant Boundary.

The buttons at the bottom of the **Uniform Cartesian Grid** window, called the Record Navigator buttons, will help you in managing the information that is defined for your grids.

Non-Uniform Cartesian Grid

In the **Non-Uniform Cartesian Grid** page, you can define Cartesian grid receptor networks with non-uniform grid spacing. Select **Grids - Non-Uniform Cartesian** from the tree located on the left side of the Receptor Pathway dialog, to display the available options.



Receptor Pathway dialog - Non-Uniform Cartesian Grid

The following parameters are necessary to define a Non-Uniform Cartesian Grid Receptor Network:

- **Network ID**: This is the ID for the grid and can be up to eight alphanumeric characters. AERMOD View automatically creates default IDs (NCART1, NCART2, etc.) but these IDs can be changed at any time.

You may select from the drop-down list the method of input: by Coordinates or by Spacing. The inputs required for each method type differ slightly.

By Coordinates

- **X Axis Coordinates:** These are the location values in meters for the first x-coordinate to the 'n-th' x-coordinate. To input the location values in the table cells, you must first specify the number of x-coordinate points on the No. X Points field. The table will automatically be divided with the number of rows equal to the number of X points. You must then input all the x-coordinate location values on the table. You may enter a value for the spacing in the Space text box to automatically calculate the coordinates.

- **Y Axis Coordinates:** These are the location values in meters for the first y-coordinate to the ‘n-th’ y-coordinate. To input the location values in the table cells, you must first specify the number of y-coordinate points on the No. Y Points field. The table will automatically be divided with the number of rows equal to the number of Y points. You must then input all the y-coordinate location values on the table. You may enter a value for the spacing in the **Space** text box to automatically calculate the coordinates.

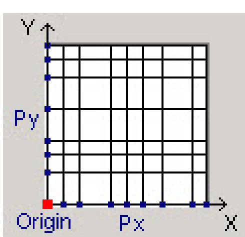
By Spacing

- **X Axis Spacing:** Here you must specify the X-coordinate for the origin and the No. of X Steps is requested. Once you enter the number of steps the table will be automatically divided with the number of rows equal to the number of X steps. You must then input all the spacing in meters between two consecutive x-axis points. You may enter a value for the spacing in the Space text box to be applied to all points.

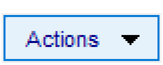
- **Y Axis Spacing:** Here you must specify the Y-coordinate for the origin and the No. of Y Steps is requested. Once you enter the number of steps the table will be automatically divided with the number of rows equal to the number of Y steps. You must then input all the spacing in meters between two consecutive y-axis points. You may enter a value for the spacing in the Space text box to be applied to all points.

> If you have graphically defined the non-uniform Cartesian grid using the Non-Uniform Cartesian Grid Tool, the above fields will be automatically filled. You may modify the values if you wish.

- **# Receptors:** This field displays the total number of receptors defined for the current grid.



The following buttons are available:



Click this button to select from two options:

- **Convert to Discrete** - coverts the current grid into discrete Cartesian receptors. This option is used when there is a need to eliminate one or more receptors from the receptor network grid, for example, eliminate receptors within a plant boundary.

- **Export to CSV File** - exports the location of each grid node as discrete receptors, with each receptor's parameters, such as X coordinate (x), Y coordinate (y), terrain elevations (ELEV), hill heights (HILL), and flagpole heights (FLAG) to a file.

> Ordered Discrete Receptors for more information on the methodology for how receptors are ordered when converted from a grid to discrete.

> See also: Receptor Import/Export Format.

Terrain Elevations Press this button to open the Receptor Terrain Elevations dialog where you can specify terrain elevations for each receptor. This button is available only is you have selected the **Elevated** terrain option in the Terrain Options page. If no terrain elevations are defined when you are modeling with elevated terrain, then the elevations will default to 0.0 meters.

Flagpole Heights Press this button to open the Flagpole Heights dialog where you can specify flagpole heights for each receptor. This is the receptor height above ground in meters. This button is available only if you have selected Yes for the Flagpole Receptors option in the Receptor Summary page.

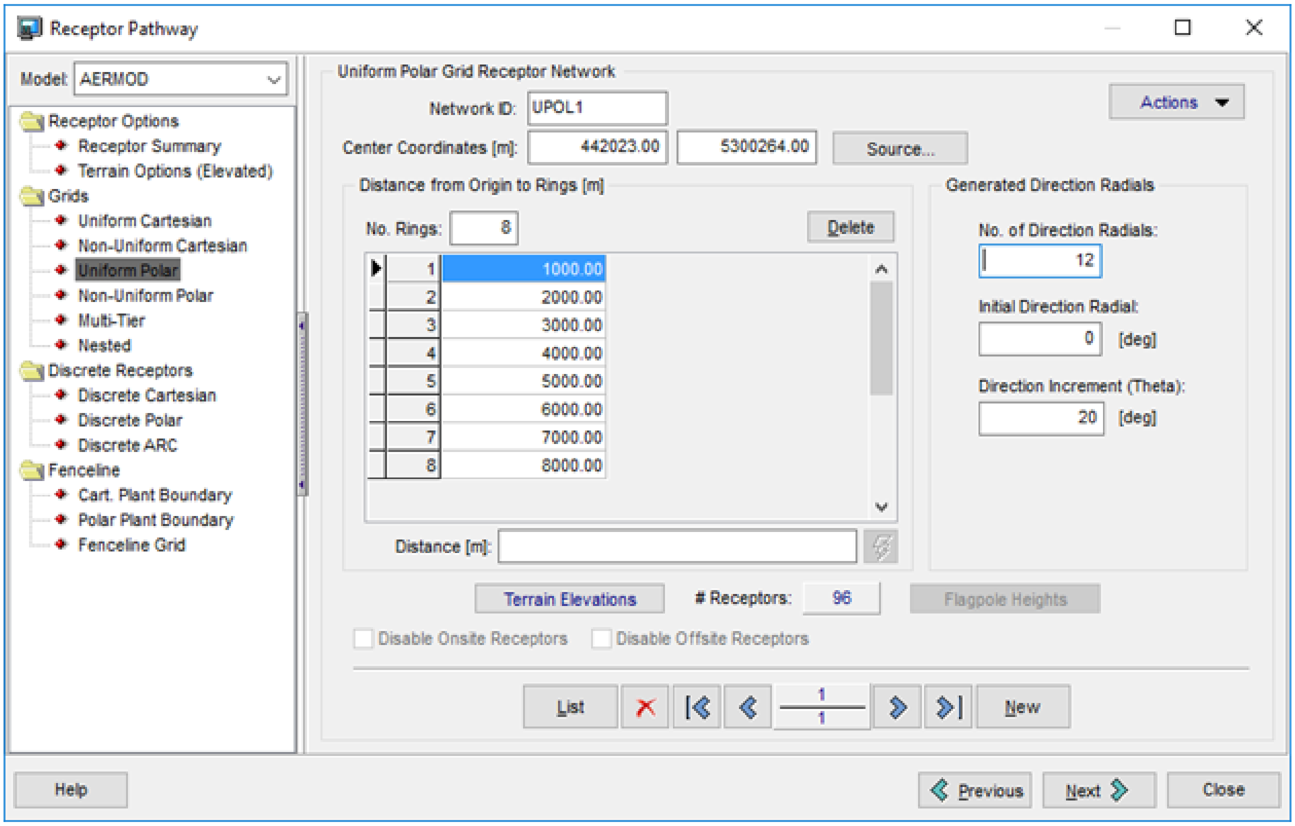
> If no flagpole receptor heights are defined when you are modeling with Flagpole Receptors, then the flagpole receptor heights will default to the Default Height value. If no default height was specified, then the flagpole height will default to 0.0 meters.

The **Disable Onsite Receptors** / **Disable Offsite Receptors** option allows the interface to either automatically disable any receptors inside the boundary (onsite) or outside the boundary (offsite) where the gridded receptor network intersects with a Plant Boundary.

The buttons at the bottom of the **Non-Uniform Cartesian Grid** window, called the Record Navigator buttons, will help you in managing the information that is defined for your grids.

Uniform Polar Grid

The **Uniform Polar Grid** screen allows you to define polar grid receptor networks with a uniform grid spacing. Select **Grids - Uniform Polar** from the tree located on the left side of the Receptor Pathway dialog, to display the available options.

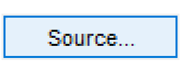


Receptor Pathway dialog - Uniform Polar Grid

The following parameters are necessary to define a Uniform Polar Grid Receptor Network:

- **Network ID:** This is the ID for the grid and can be up to eight alphanumeric characters. AERMOD View automatically creates default IDs (UPOL01, UPOL02, etc.) but these IDs can be changed at any time.

- **Origin (Center):** Specify here the X and Y coordinates in meters for the origin (center) of the grid.



Click on this button to display the Origin Coordinates dialog. Here you can select a source already defined in your project from which to obtain the origin coordinates for the grid.

Distance from Origin to Rings

In this panel you must specify the distances in meters from the grid origin to each ring. To input the distances, you must first specify the **No. of Rings**. The table will automatically be divided with the number of rows equal to the number of rings. You can then input all the distances from the origin to the rings. You may enter a value for the distance in the **Distance** text box to automatically assign the same distance between each ring.

Generated Direction Radials

In this panel you must specify the direction radials for the grid by supplying the following information -

- **No. of Directions Radials:** Enter here the number of directions used for the polar grid.

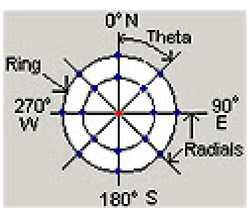
- **Initial Direction Radial:** This is the starting direction of the polar grid in degrees.

- **Direction Increment (Theta):** Specify here the increment in degrees for defining directions.

> **No. of Direction Radials** times **Direction Increment (Theta)** must be equal to or less than 360. A combination that results in a number greater than 360 indicates overlapping radials, which is not permitted by the model.

> If you have graphically defined the grid using the Uniform Polar Grid Tool, the above fields will be automatically filled. You may modify the values if you wish.

- **# Receptors:** This field displays the total number of receptors defined for the current grid.



The following buttons are available:

Click this button to select from two options:

- **Convert to Discrete** - coverts the current grid into discrete Cartesian receptors. This option is used when there is a need to eliminate one or more receptors from the receptor network grid, for example, eliminate receptors within a plant boundary.

- **Export to CSV File** - exports the location of each grid node as discrete receptors, with each receptor's parameters, such as X coordinate (x), Y coordinate (y), terrain elevations (ELEV), hill heights (HILL), and flagpole heights (FLAG) to a file.

> Ordered Discrete Receptors for more information on the methodology for how receptors are ordered when converted from a grid to discrete.

> See also: Receptor Import/Export Format.

Terrain Elevations Press this button to open the Receptor Terrain Elevations dialog where you can specify terrain elevations for each receptor. This button is available only is you have selected the **Elevated** terrain option in the Terrain Options page. If no terrain elevations are defined when you are modeling with elevated terrain, then the elevations will default to 0.0 meters.

Flagpole Heights Press this button to open the Flagpole Heights dialog where you can specify flagpole heights for each receptor. This is the receptor height above ground in meters. This button is available only if you have selected **Yes** for the **Flagpole Receptors** option in the Receptor Summary page.

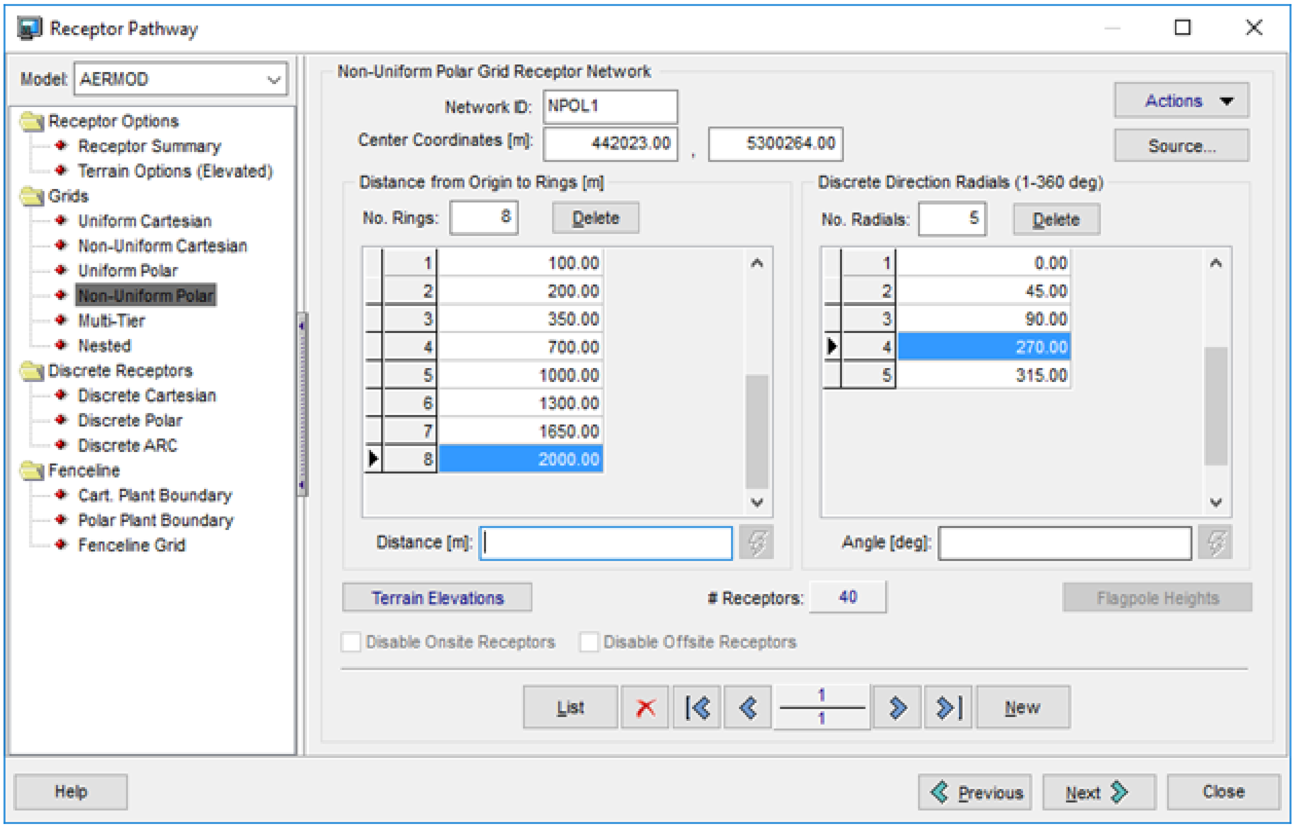
> If no flagpole receptor heights are defined when you are modeling with Flagpole Receptors, then the flagpole receptor heights will default to the **Default Height** value. If no default height was specified, then the flagpole height will default to 0.0 meters.

The **Disable Onsite Receptors** / **Disable Offsite Receptors** option allows the interface to either automatically disable any receptors inside the boundary (onsite) or outside the boundary (offsite) where the gridded receptor network intersects with a Plant Boundary.

The buttons at the bottom of the Uniform Polar Grid window, called the Record Navigator buttons, will help you in managing the information that is defined for your grids.

Non-Uniform Polar Grid

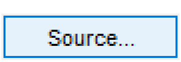
The **Non-Uniform Polar Grid** screen allows you to define polar grid receptor networks with a non-uniform grid spacing. Select **Grids - Non-Uniform Polar** from the tree located on the left side of the Receptor Pathway dialog, to display the available options.



Receptor Pathway dialog - Non-Uniform Polar Grid

The following parameters are necessary to define a Non-Uniform Polar Grid Receptor Network:

- **Network ID**: This is the ID for the grid and can be up to eight alphanumeric characters. AERMOD View automatically creates default IDs (NPOL01, NPOL02, etc.) but these IDs can be changed at any time.
- **Origin (Center):** Specify here the X and Y coordinates in meters for the origin (center) of the grid.



Click on this button to display the **Origin Coordinates** dialog. Here you can select a source already defined in your project from which to obtain the origin coordinates for the grid.

Distance from Origin to Rings

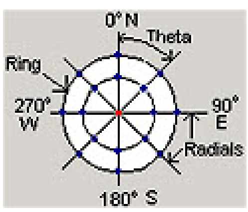
In this panel you must specify the distances in meters from the network origin to each ring. To input the distances, you must first specify the **No. of Rings**. The table will automatically be divided with the number of rows equal to the number of rings. You can then input all the distances from the origin to the rings. You may enter a value for the distance in the **Distance** text box and click on the button to automatically assign the same distance between each ring.

Discrete Direction Radials

In this panel you must specify the discrete direction radials (1 to 360 degrees) for the polar network. To input the directions radials in the table cells, you must first specify the **No. Radials**. The table will automatically be divided with the number of rows equal to the number of radials you have specified. You must then input the direction in degrees for each radial. You may enter a value for the angle in the **Angle** text box to automatically assign the same angle between each ring

> If you have graphically defined the non-uniform polar grid using the Non-Uniform Polar Grid Tool, the above fields will be automatically filled. You may modify the values if you wish.

- **Total # Receptors:** This field displays the total number of receptors defined for the current grid.



The following buttons are available -

Click this button to select from two options:

- **Convert to Discrete** - coverts the current grid into discrete Cartesian receptors. This option is used when there is a need to eliminate one or more receptors from the receptor network grid, for example, eliminate receptors within a plant boundary.

- **Export to CSV File** - exports the location of each grid node as discrete receptors, with each receptor's parameters, such as X coordinate (x), Y coordinate (y), terrain elevations (ELEV), hill heights (HILL), and flagpole heights (FLAG) to a file.

Ordered Discrete Receptors for more information on the methodology for how receptors are ordered when converted from a grid to discrete.

See also: Receptor Import/Export Format.

Terrain Elevations Press this button to open the Receptor Terrain Elevations dialog where you can specify terrain elevations for each receptor. This button is available only is you have selected the **Elevated** terrain option in the Terrain Options page. If no terrain elevations are defined when you are modeling with elevated terrain, then the elevations will default to 0.0 meters.

Flagpole Heights Press this button to open the Flagpole Heights dialog where you can specify flagpole heights for each receptor. This is the receptor height above ground in meters. This button is available only if you have selected **Yes** for the **Flagpole Receptors** option in the Receptor Summary page.

If no flagpole receptor heights are defined when you are modeling with Flagpole Receptors, then the flagpole receptor heights will default to the Default Height value. If no default height was specified, then the flagpole height will default to 0.0 meters.

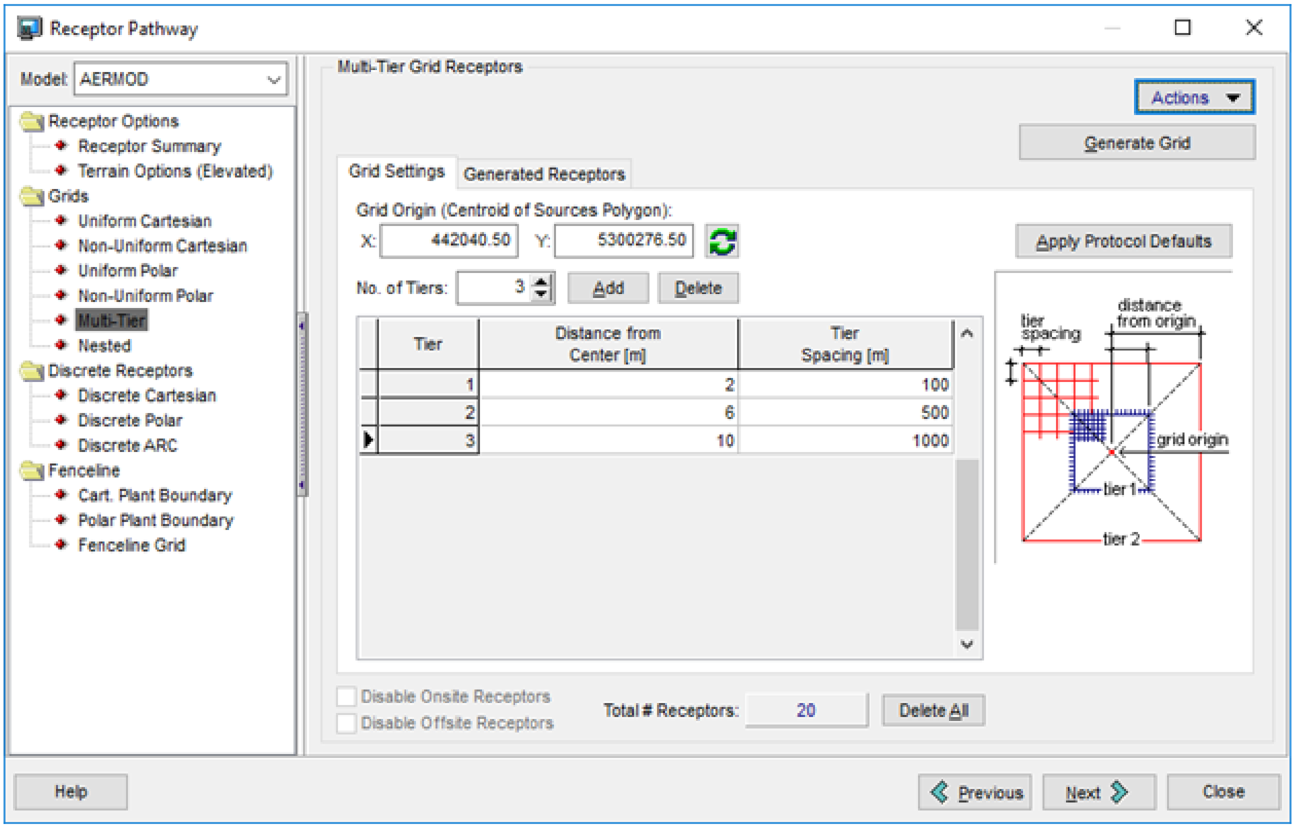
The **Disable Onsite Receptors** / **Disable Offsite Receptors** option allows the interface to either automatically disable any receptors inside the boundary (onsite) or outside the boundary (offsite) where the gridded receptor network intersects with a Plant Boundary.

The buttons at the bottom of the **Non-Uniform Polar Grid** window, called the Record Navigator buttons, will help you in managing the information that is defined for your grids.

Multi-Tier Grid

In the **Multi-Tier Grid** window, you can define a multi-tier grid. The multi-tier grid is defined by discrete Cartesian receptors, square in shape, with the origin at the center of the grid and can be defined with different tier spacing. The Multi-Tier grid can also be called a Risk Grid because it can be used to define the receptor grid according to the 1998 U.S. EPA-OSW Human Health Risk Assessment Protocol (HHRAP) and the 1999 U.S. EPA-OSW Screening Level Ecological Risk Assessment Protocol (SLERAP).

Select **Receptor Pathway - Multi-Tier Grid** from the tree located on the left side of the Receptor Pathway dialog, to display the available options.



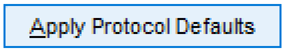
Receptor Pathway dialog - Multi-Tier Grid

Two tabs are available in the **Multi-Tier Grid** screen -

Grid Settings tab

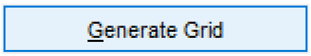
In this tab you must specify the following parameters to define a Multi-Tier Grid:

- **Grid Origin (Centroid of Sources Polygon):** You must define the X and Y coordinates for the origin (center) of the grid. Press the  button to obtain the centroid of the polygon formed by linking the origin of all the sources already defined in your project.
- **No. of Tiers:** Specify here the number of tiers for the grid. Tiers are the segments with different grid spacing.
- **Distance from Center (Origin):** You must specify the distance in meters from the center of the grid to each tier.
- **Tier Spacing:** For each tier, you must specify the spacing between receptors in the X and Y directions.



Press this button to get the default grid settings according to the HHRAP and SLERAP protocols. According to these two U.S. EPA-OSW protocols, the risk grid should be:

- A 100-meter spaced grid from the centroid of the emission sources out to a radius of 3 km.
- A 500-meter spaced grid extending from 3 km to 10 km.
- Origin of the grid should be the centroid of the polygon formed by all sources.



Press this button to generate the Risk Grid. When AERMOD finishes generating the grid, the total number of receptors created is automatically displayed. If you wish to delete the generated receptors click on the **Delete All** button.

Click this button to select from two options:

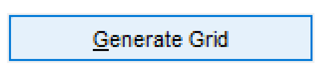
- **Remove Plant Receptors** - Eliminates receptors that are within the plant boundary.
- **Convert to Discrete** - coverts the current grid into discrete Cartesian receptors. This option is used when there is a need to eliminate one or more receptors from the receptor network grid, for example, eliminate receptors within a plant boundary.
- **Export to CSV File** - exports the location of each grid node as discrete receptors, with each receptor's parameters, such as X coordinate (x), Y coordinate (y), terrain elevations (ELEV), hill heights (HILL), and flagpole heights (FLAG) to a file.

The **Disable Onsite Receptors** / **Disable Offsite Receptors** option allows the interface to either automatically disable any receptors inside the boundary (onsite) or outside the boundary (offsite) where the gridded receptor network intersects with a Plant Boundary.

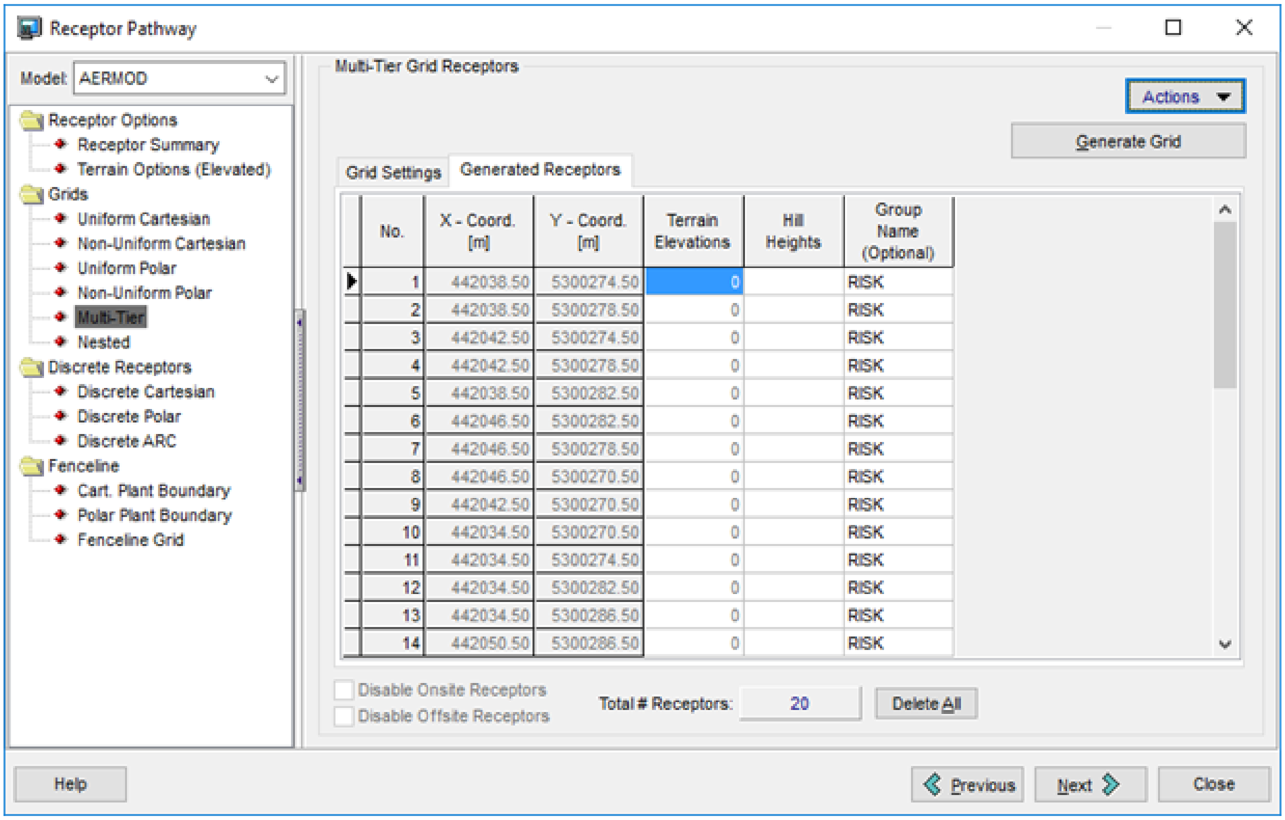
Ordered Discrete Receptors for more information on the methodology for how receptors are ordered when converted from a grid to discrete.

See also: Receptor Import/Export Format.

Generated Receptors tab

This tab is automatically displayed once you have clicked on the


button. The X and Y coordinates for all Multi-Tier receptors are automatically calculated and displayed here.



Multi-Tier Grid - Generated Discrete Receptors tab

Depending on the modeling options you have selected, the following additional parameters must be specified:

- **Terrain Elevations:** This column is displayed only if the **Elevated** terrain option was selected in the Terrain Options page. Here you can specify elevations for the multi-tier receptors. If no terrain elevations are defined when you are modeling with elevated terrain, then the elevations will default to 0.0 meters. If you have processed and imported terrain elevation files, the elevations will be automatically displayed here.

- **Hill Heights: AERMOD Only** This column is displayed only if the Elevated terrain option was selected in the Terrain Options page. Here you can view the hill heights generated from AERMAP once you've processed your terrain data.

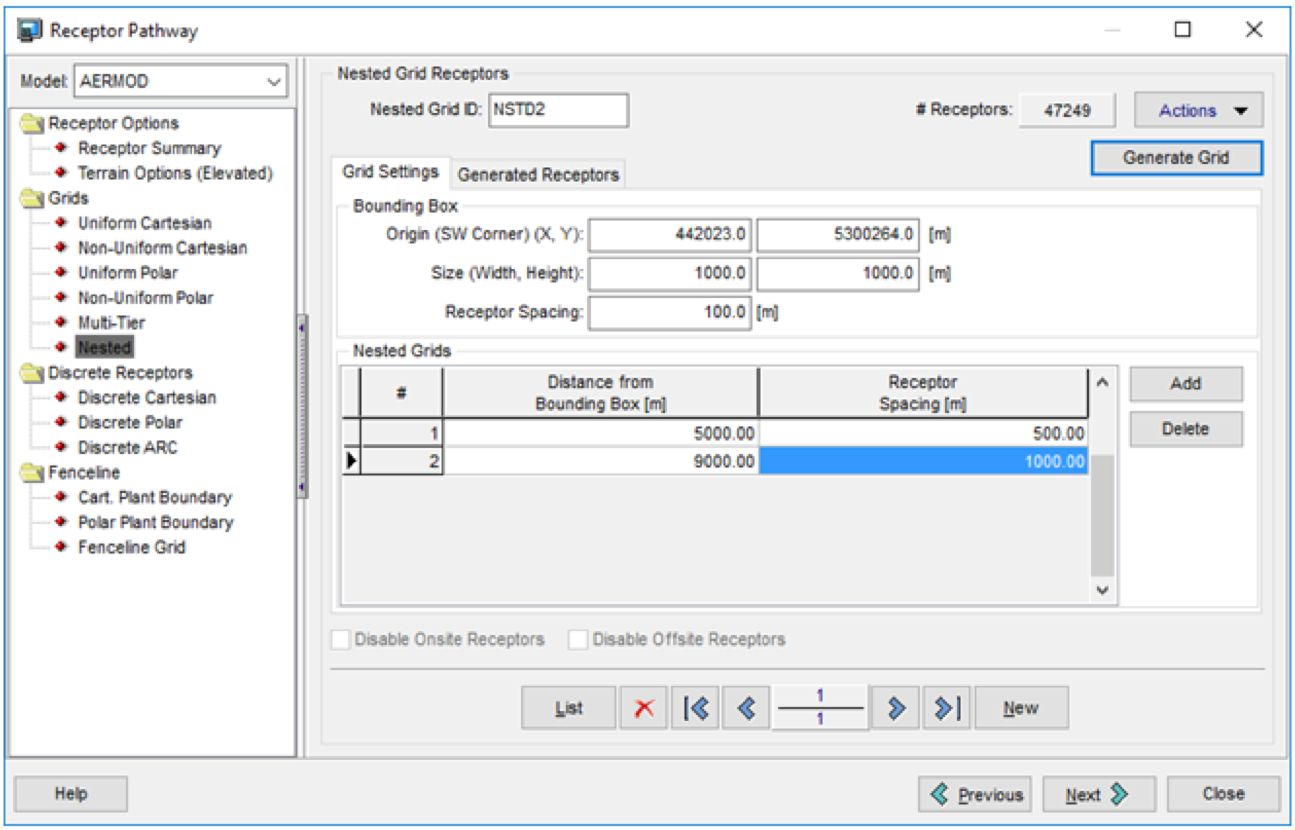
> You may modify the hill heights, but this is only recommended for advanced users.

- **Flagpole Heights:** This column is displayed only if the Flagpole Receptors terrain option was selected in the Receptor Summary page. Here you can specify the receptor height above ground in meters. Any missing values in the Flagpole Heights column will be interpreted by the U.S. EPA models as being equal to the default value specified for the Flagpole Receptors option. If no default height was specified, then the flagpole height will default to 0.0 meters.

- **Group Name:** The group name, RISK, is automatically assigned to every multi-tier grid receptor. To change this group name click within the cell to display the ... button. You can then click on this button to open the Receptor Groups dialog.

Nested Grid

The **Nested Grid Receptors** screen allows you to define a nested grid. Nested grids are grids with varying mesh size which are embedded in each other (nested). Each individual grid in a nested grid is called a tier grid. Generally, the grid with the smaller mesh size is used near the sources while further away from the sources, a larger mesh size grid is used. Select **Grids - Nested** from the tree located on the left side of the Receptor Pathway dialog, to display the available options.



Receptor Pathway dialog - Nested Grid

Two tabs are available in the **Nested Grid Receptors** screen -

Grid Settings tab

In this tab you must specify the following parameters to define the Nested Grid:

Bounding Box: In this panel you need to define the parameters for your bounding box. This is the box that indicates the extent of the location of sources in your project and is the smallest box in the grid -

- **Origin (SW Corner):** You must define the X and Y coordinates for the origin (south-west corner) of the bounding box.

- **Size:** Specify here the width and the height of the bounding box.

- **Receptor Spacing:** Specify the spacing between receptors in the X and Y directions within the bounding box.

Nested Grids: In this panel you can define the spacing for the individual grids -

- **Distance from Bounding Box:** You must specify the distance in meters from the bounding box to each tier.

- **Receptor Spacing:** For each tier or nested grid, you must specify the spacing between receptors in the X and Y directions.

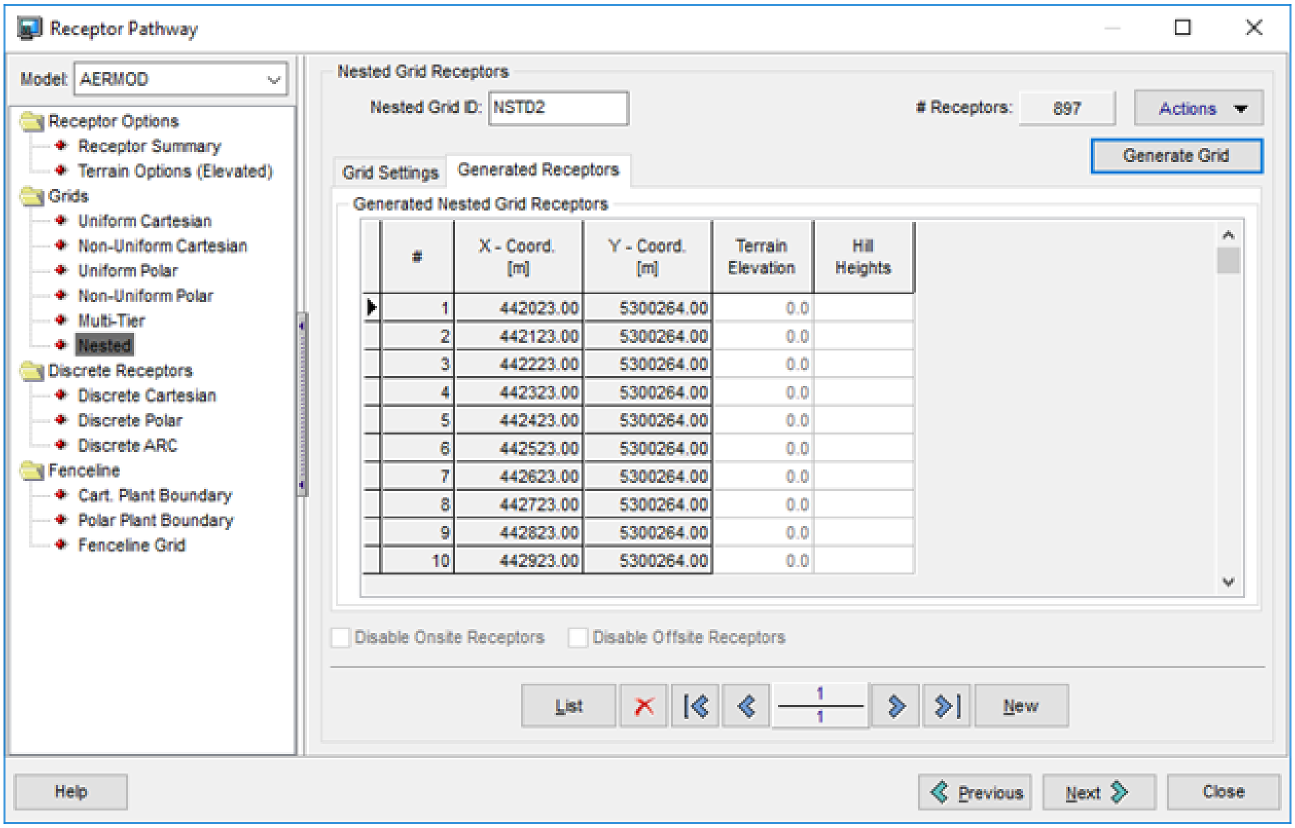
> If you have graphically defined the grid using the Nested Grid Receptors Tool, the above fields will be automatically filled. You may modify the values if you wish.

> The Nested Grid Receptors Tool automatically defines the grid extents based on the location of your sources, according to Ontario Provincial MOE specifications outlined in the Air Dispersion Modelling Guideline for Ontario (ADMGO).

- **# Receptors:** This field displays the total number of receptors defined for the current receptor grid network, and is automatically filled. Receptors are defined at each grid node.

Generated Receptors tab

The **Generated Receptors** tab displays the X and Y coordinates for the generated grid receptors. The **Terrain Elevations** column is displayed only if the **Elevated** terrain option was selected in the Terrain Options page. Here you can specify elevations for the nested grid receptors. If no terrain elevations are defined when you are modeling with elevated terrain, then the elevations will default to 0.0 meters. If you have processed and imported terrain elevation files, the elevations will be automatically displayed here. The Hill Heights column is displayed only if the Elevated terrain option was selected in the Terrain Options page. Here you can view the hill heights generated from AERMAP once you've processed your terrain data.



Nested Grid - Generated Discrete Receptors tab

The following buttons are available:

Click this button to select from two options:

- **Remove Plant Receptors** - Eliminates receptors that are within the plant boundary.
- **Convert to Discrete**- converts the grid into discrete cartesian receptors. This option is used when there is a need to eliminate one or more receptors from the receptor grid, for example, eliminate receptors within a plant boundary.
- **Export to CSV File** - exports the location of each grid node as discrete receptors, with each receptor's parameters, such as X coordinate (x), Y coordinate (y), terrain elevations (ELEV), hill heights (HILL), and flagpole heights (FLAG) to a file.

See also Receptor Import/Export Format

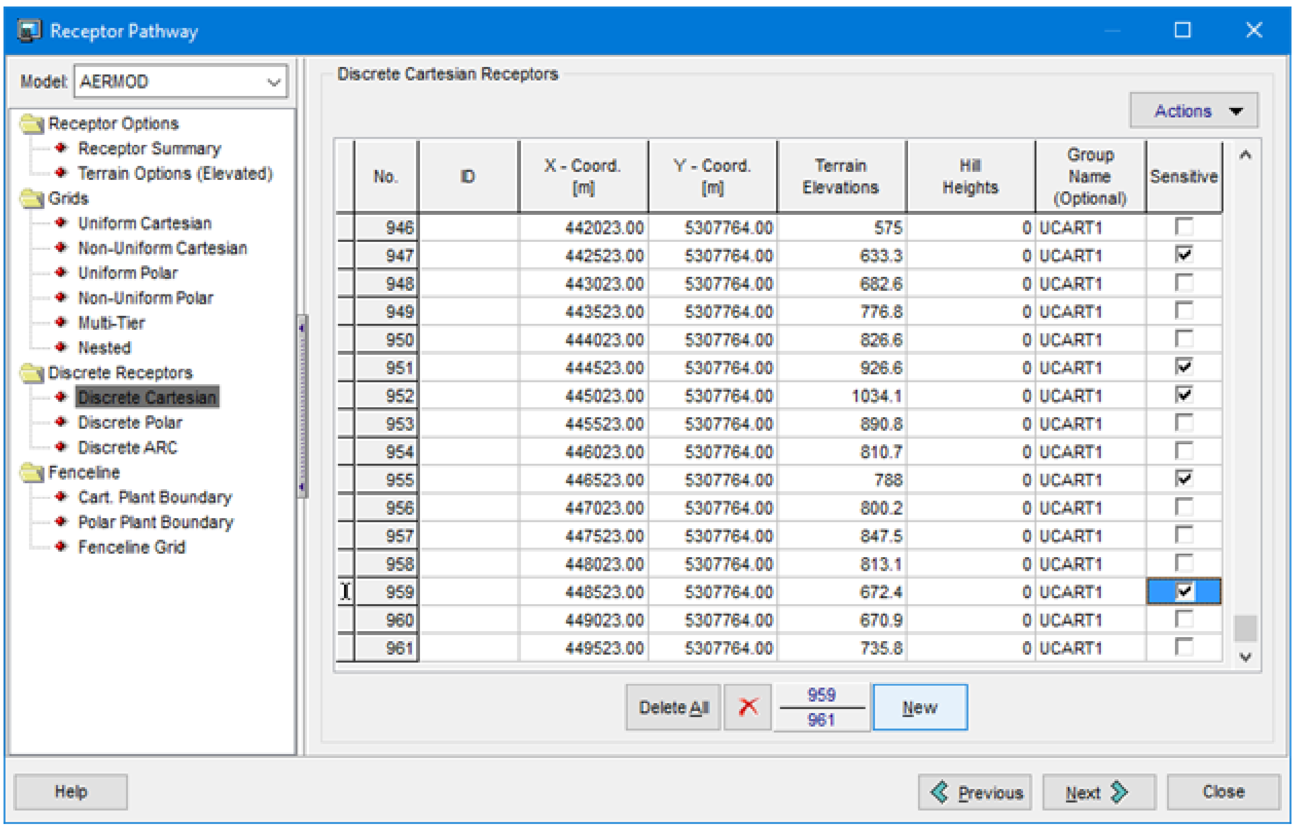
Generate Grid Click on this button if you wish to retrieve the discarded receptors within the plant boundary, or to update the Generated Receptors list after a modification.

The **Disable Onsite Receptors** / **Disable Offsite Receptors** option allows the interface to either automatically disable any receptors inside the boundary (onsite) or outside the boundary (offsite) where the gridded receptor network intersects with a Plant Boundary.

The buttons at the bottom of the **Nested Grid** window, called the Record Navigator buttons, will help you in managing the information that is defined for your grids.

Discrete Cartesian Receptors

The **Discrete Cartesian Receptors** screen allows you to define one or more discrete Cartesian receptors. Select **Discrete Receptors - Discrete Cartesian** from the tree located on the left side of the Receptor Pathway dialog, to display the available options.



Receptor Pathway dialog - Discrete Cartesian

The following parameters are necessary to define a Discrete Cartesian Receptor:

- **No.:** This column displays the entry number or record number for each receptor you have defined.

- **X Coord.:** This is the X coordinate, in meters, for the receptor location.

- **Y Coord.:** This is the Y coordinate, in meters, for the receptor location.

> If you have graphically defined a receptor using the Discrete Cartesian Receptor Tool, the above fields will be automatically filled. You may modify the values if you wish.

- **Terrain Elevations:** This column is displayed only if the Elevated terrain option was selected in the Terrain Options page. Here you can specify elevations for the receptors. If you have processed and imported terrain elevation files, the elevations will be automatically displayed here.

> If you are modeling with Elevated terrain and no value is input in the Terrain Elevations column, then the missing terrain elevations will default to 0.0 meters.

- **Hill Heights:** AERMOD Only—This column is displayed only if the Elevated terrain option was selected in the Terrain Options page. Here you can view the hill heights generated from AERMAP once you've processed your terrain data.

> You may modify the hill heights, but this is only recommended for advanced users.

- **Flagpole Heights:** This column is displayed only if the Flagpole Receptors terrain option was selected in the Receptor Summary page. Here you can specify the receptor height above ground in meters. Any missing values in the Flagpole Heights column will be interpreted by the U.S. EPA models as being equal to the default value specified for the Flagpole Receptors option. If no default height was specified, then the flagpole height will default to 0.0 meters.

- **Group Name:** Here, you can specify a group name for your receptors. To change this group name click within the cell to display the ... button. You can then click on this button to open the Receptor Groups dialog.

- **Sensitive:** Here, you can specify if the receptor is considered sensitive.

> It is possible to display discrete receptors in the drawing area with different colors and markers. For example, if you have sensitive receptors that you would like displayed differently from the rest of the receptors in your project you can do

> so in the Graphical Options - Color Mappings page. The only requirement for this option is that the receptor/s must be categorized in a separate group.

The following buttons are available:

Click this button to select from the following options:

- **Apply Group Name** - You can select multiple receptors from the list by pressing down the **Shift** key while you select the items in sequence or the **Ctrl** key to make disjointed selections. Once selected, use this Apply Group Name option to open the Receptor Groups dialog to apply a Group Name to the selected receptors.

- **Remove Plant Receptors** - Eliminates receptors that are within the plant boundary.

- **Import from File** - Imports receptor parameters, such as X coordinate (x), Y coordinate (y), terrain elevations (ELEV), hill heights (HILL), and flagpole heights (FLAG) from a file.

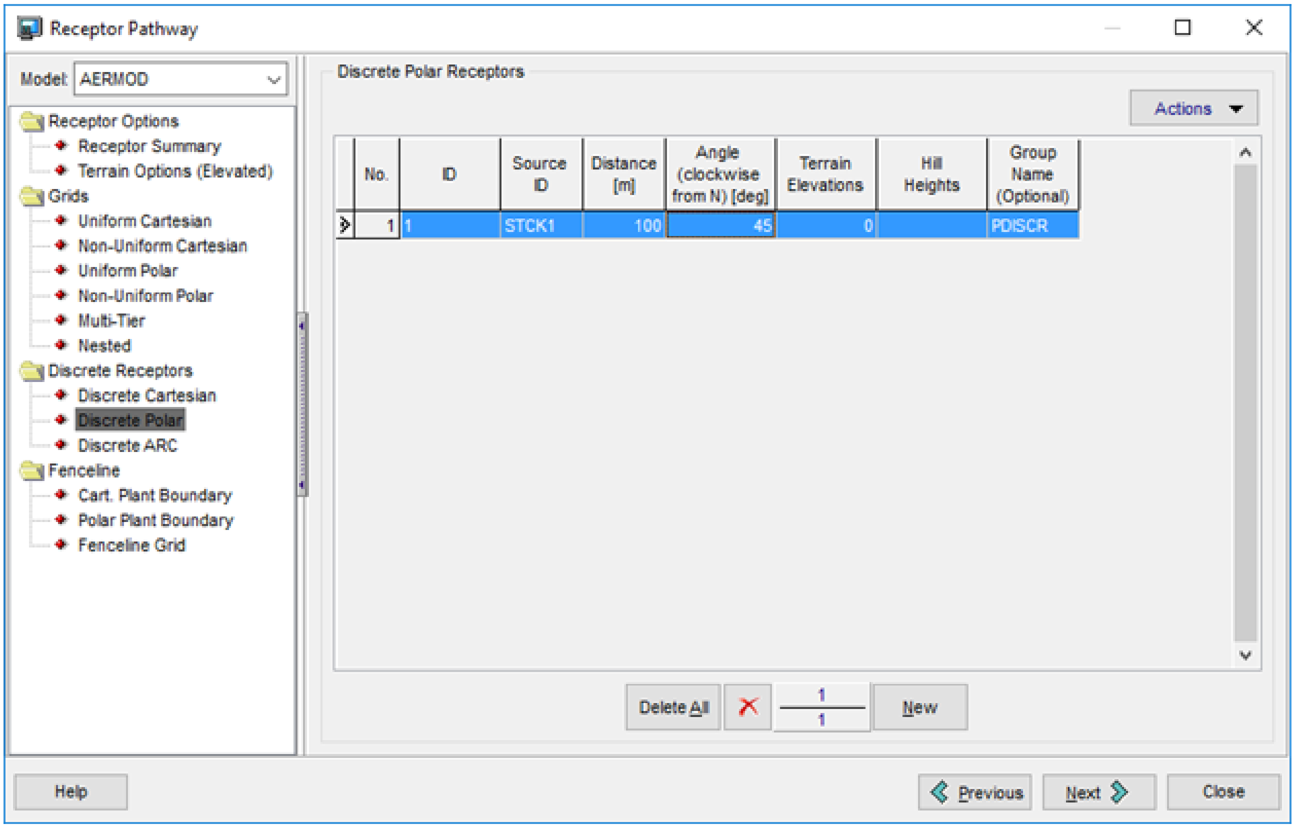
- **Export to CSV File** - exports the receptor parameters, such as X coordinate (x), Y coordinate (y), terrain elevations (ELEV), hill heights (HILL), and flagpole heights (FLAG) to a file.

> See Also: Receptor Import/Export Format.

The buttons at the bottom of the Discrete Cartesian Receptors window, called the Record Navigator buttons, will help you in managing the information that is defined for your receptors.

Discrete Polar Receptors

The **Discrete Polar** screen allows you to define one or more discrete polar receptors. Discrete Polar Receptors are defined by specifying a distance and a direction (angle) from an existing source. Select **Discrete Receptors - Discrete Polar** from the tree located on the left side of the Receptor Pathway dialog, to display the available options.



Receptor Pathway dialog - Discrete Polar Receptors

The following parameters are necessary to define a discrete polar receptor:

- **No.:** This column displays the record number for each discrete polar receptor you have defined.

- **Source ID:** You must define which source is the origin for the polar receptor location. AERMOD View automatically supplies you with the list of the Source IDs for all sources defined for your project. Click the cell to display a drop-down list containing all the available sources.

- **Distance:** This is the distance, in meters, from the source to the discrete polar receptor.

- **Angle:** This is the direction, in degrees, measured clockwise from north for the discrete polar receptor location.

> If you have graphically defined a receptor using the Discrete Polar Receptor Tool, the above fields will be automatically filled. You may modify the values if you wish.

- **Terrain Elevations:** This column is displayed only if the Elevated terrain option was selected in the Terrain Options page. Here you can specify elevations for the receptors. If you are modeling with Elevated terrain and no value is input in the Terrain Elevations column, then the missing terrain elevations will default to 0.0 meters If you have processed and imported terrain elevation files, the elevations will be automatically displayed here.

- **Hill Heights: AERMOD Only** This column is displayed only if the Elevated terrain option was selected in the Terrain Options page. Here you can view the hill heights generated from AERMAP once you've processed your terrain data.

> You may modify the hill heights, but this is only recommended for advanced users.

- **Flagpole Heights:** This column is displayed only if the Flagpole Receptors terrain option was selected in the Receptor Summary page. Here you can specify the receptor height above ground in meters. Any missing values in the Flagpole Heights column will be interpreted by the U.S. EPA models as being equal to the default value specified for the Flagpole Receptors option. If no default height was specified, then the flagpole height will default to 0.0 meters.

- **Group Name:** Here may specify a group name. Click on the Group Name cell to display the ... button. You can then click on this button to open the Receptor Groups dialog.

The following buttons are available:

Actions ▼ Click this button to select the following option:

- **Export to CSV File** - exports the receptor parameters, such as X coordinate (x), Y coordinate (y), terrain elevations (ELEV), hill heights (HILL), and flagpole heights (FLAG) to a file.

> See Also: Receptor Import/Export Format.

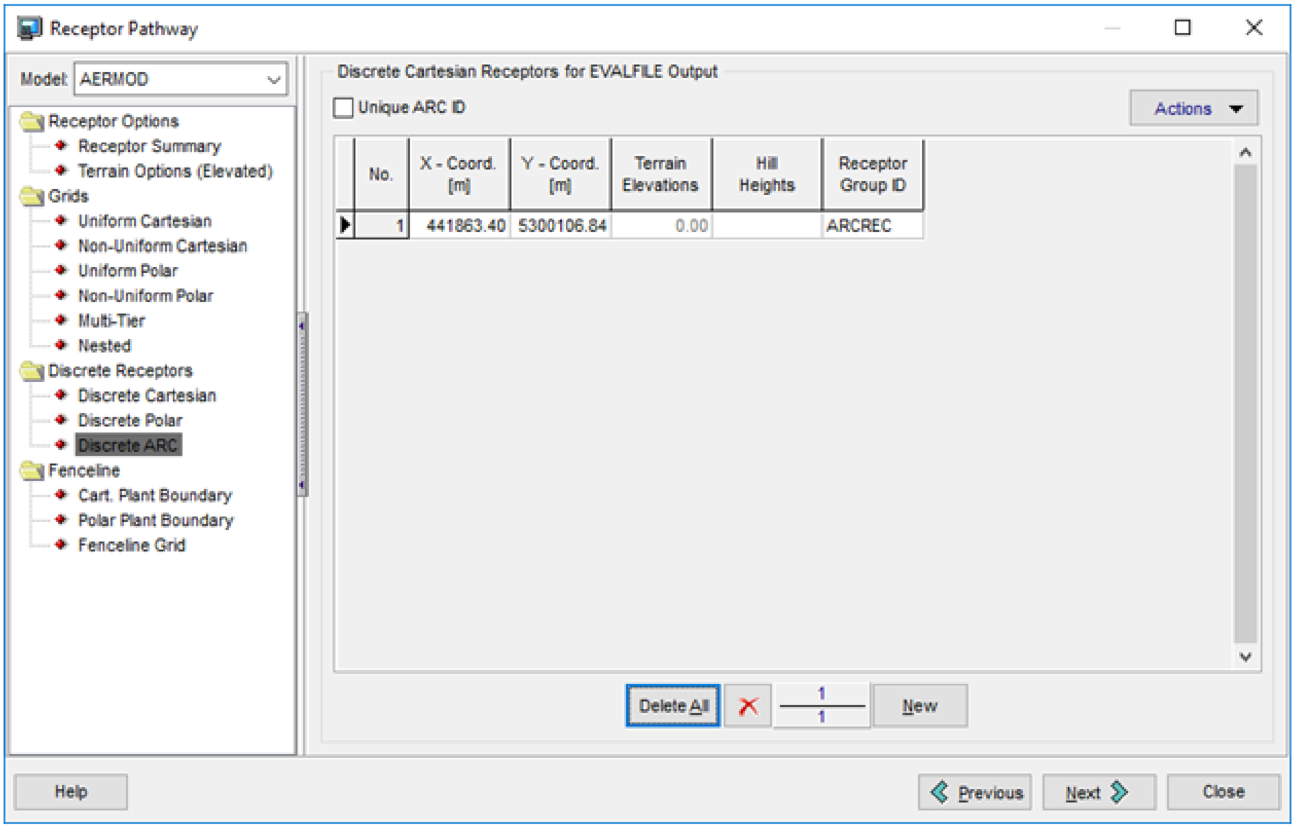
The buttons at the bottom of the Discrete Polar Receptors window, called the Record Navigator buttons, will help you in managing the information that is defined for your receptors.

Discrete ARC Receptors

AERMOD Only

In the **Discrete ARC** page, you can define one or more Discrete Cartesian (ARC) Receptors. The Discrete Cartesian (ARC) Receptor option is only applicable to AERMOD and is the only option that the grouping of receptors is recognized by the U.S. EPA model. This Discrete Cartesian (ARC) Receptors option is designed to be used with the EVALFILE option described in the Evaluation Files window. If the Discrete Cartesian (ARC) Receptors option is used without the use of the EVALFILE option, then the receptor grouping is ignored.

Select **Discrete Receptors - Discrete ARC** from the tree located on the left side of the Receptor Pathway dialog, to display the available options.



Receptor Pathway dialog - Discrete ARC Receptors

The following parameters are necessary to define Discrete Cartesian (ARC) Receptors:

- **Unique ARC ID.:** Check this box if you wish to have a unique ID for each receptor.
- **No.:** This column displays the entry number or record number for each receptor you have defined.
- **ARC ID:** This column is displayed instead of the **Receptor Group ID** column if you have checked the Unique ARC ID box. A unique, default ID will be assigned and displayed for each receptor.
- **X Coord.:** This is the X coordinate, in meters, for the receptor location.
- **Y Coord.:** This is the Y coordinate, in meters, for the receptor location.

> If you have graphically defined a receptor using the [Discrete Cartesian Receptor (ARC) Tool](), the above fields will be automatically filled. You may modify the values if you wish.

- **Terrain Elevations:** This column is displayed only if the Elevated terrain option was selected in the Terrain Options page. Here you can specify elevations for the receptors. If you have processed and imported terrain elevation files, the elevations will be automatically displayed here.

> If you are modeling with Elevated terrain and no value is input in the Terrain Elevations column, then the missing terrain elevations will default to 0.0 meters.

- **Hill Heights: AERMOD Only**—This column is displayed only if the Elevated terrain option was selected in the Terrain Options page. Here you can view the hill heights generated from AERMAP once you've processed your terrain data.

> You may modify the hill heights, but this is only recommended for advanced users.

- **Flagpole Heights:** This column is displayed only if the Flagpole Receptors terrain option was selected in the Receptor Summary page. Here you can specify the receptor height above ground in meters. Any missing values in the Flagpole Heights column will be interpreted by the U.S. EPA models as being equal to the default value specified for the Flagpole Receptors option. If no default height was specified, then the flagpole height will default to 0.0 meters.

- **Receptor Group ID:** The Receptor Group ID can be used to group receptors by ARC. This parameter is mandatory for Discrete Cartesian (ARC) Receptors. The group name, ARCREC, is automatically assigned every time you add a new receptor. This group name can be changed at any time by clicking within the cell to display the ... button. You can then click on this button to open the Receptor Groups dialog.

The following buttons are available:

Actions ▼ Click this button to select from three options:

- **Remove Plant Receptors** - Eliminates receptors that are within the plant boundary.

- **Import from File** - Imports receptor parameters, such as X coordinate (x), Y coordinate (y), terrain elevations (ELEV), hill heights (HILL), and flagpole heights (FLAG) from a file.

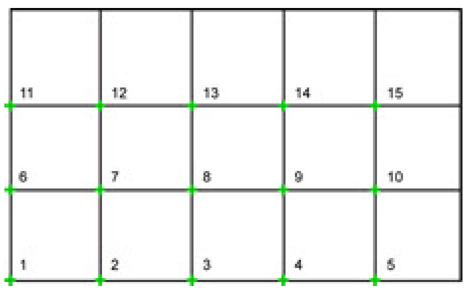
- **Export to CSV File** - exports the receptor parameters, such as X coordinate (x), Y coordinate (y), terrain elevations (ELEV), hill heights (HILL), and flagpole heights (FLAG) to a file.

See Also: Receptor Import/Export Format.

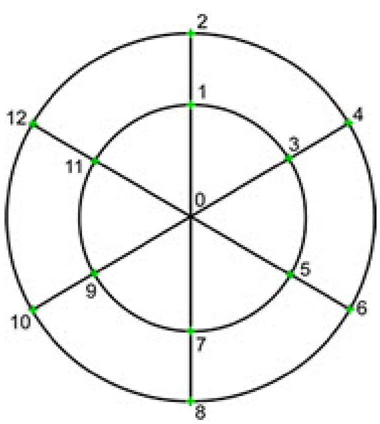
The buttons at the bottom of the Discrete ARC Receptors window, called the Record Navigator buttons, will help you in managing the information that is defined for your receptors.

Ordered Discrete Receptors

This is the methodology by which the receptors are ordered when they are converted from grid to discrete when the PLOTFILE is written. This is the same methodology adopted by the U.S. EPA



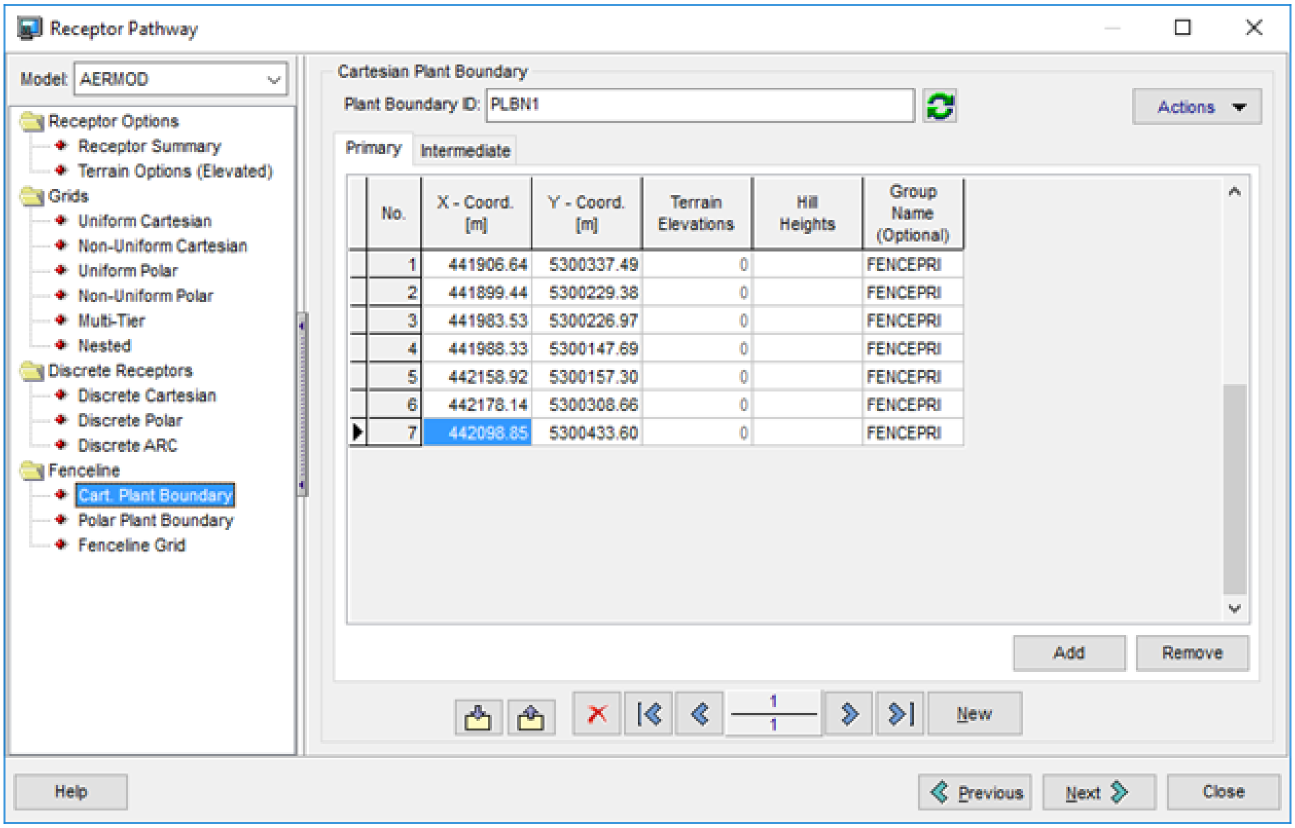
Uniform and Non-Uniform Cartesian Grids



Uniform and Non-Uniform Polar Grids

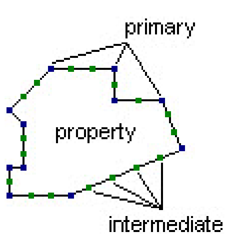
Cartesian Plant Boundary

In the **Cartesian Plant Boundary** page, you can define the boundaries of your plant using discrete Cartesian receptors. Select **Fenceline - Cart. Plant Boundary** from the tree located on the left side of the Receptor Pathway dialog, to display the available options.



Receptor Pathway dialog - Cartesian Plant Boundary

You can define your plant boundary or fenceline by assigning discrete Cartesian receptors for each node of the fenceline polygon. The fenceline nodes, also called primary fenceline receptors, must be defined in a clockwise direction or counter clockwise direction. Intermediate receptors are assigned between primary fenceline receptors, equally spaced.



At the top of the list you can enter an identification name for the plant boundary being defined. The ID can be up to 8 characters long and are always in upper case. Press the button to get default IDs automatically set up for you.

Two tabs are available in the **Cartesian Plant Boundary** window:

Primary tab

In this tab you can define the primary fenceline receptors. The following parameters must be defined -

- **No.:** This column displays the record number for each fenceline node (discrete Cartesian receptor) you have defined.
- **X Coord.:** This is the X coordinate, in meters, for the node location.
- **Y Coord.:** This is the Y coordinate, in meters, for the node location.
- **Terrain Elevations:** This column is displayed only if the Elevated terrain option was selected in the Terrain Options page. Here you can specify elevations for the receptors. If you have processed and imported terrain elevation files, the elevations will be automatically displayed here.

> If you are modeling with **Elevated** terrain and no value is input in the **Terrain Elevations** column, then the missing terrain elevations will default to 0.0 meters.

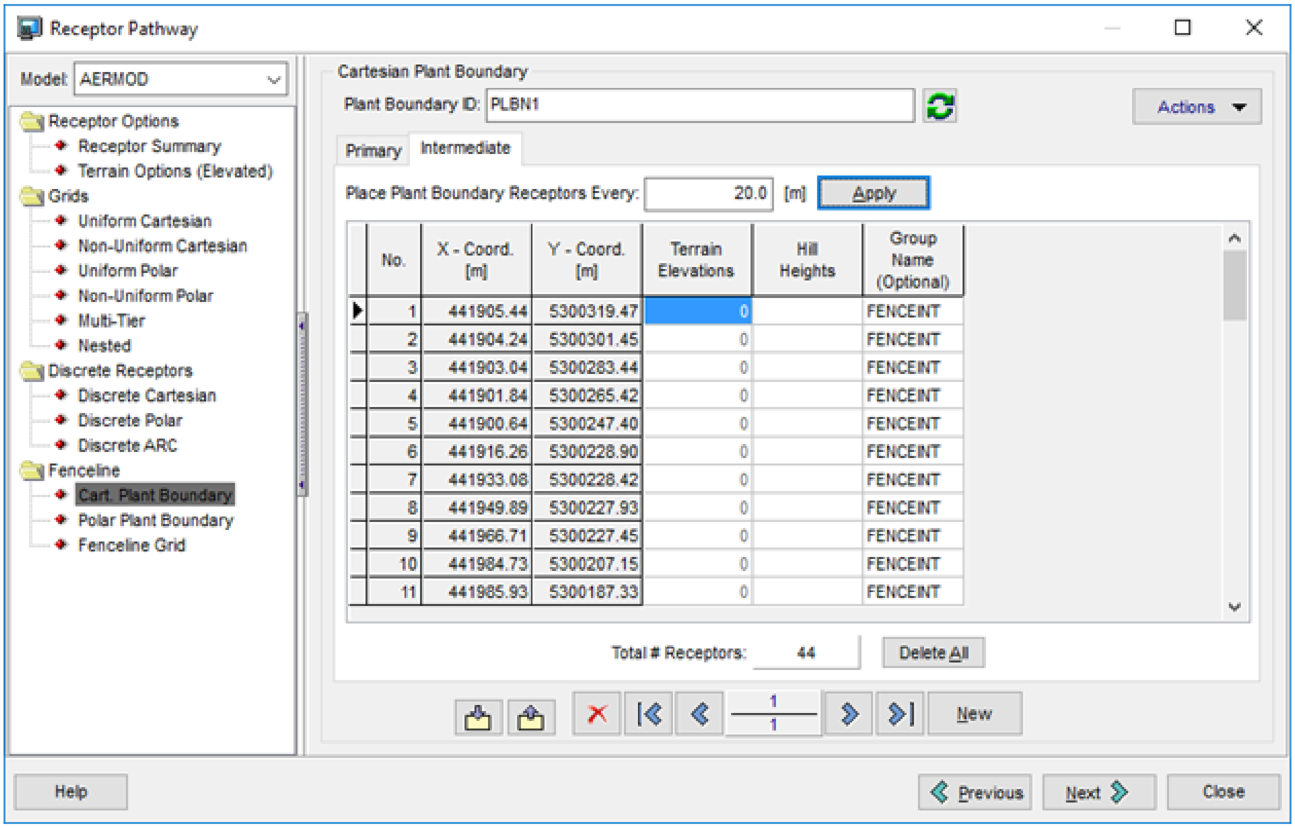
- **Hill Heights: AERMOD Only**—This column is displayed only if the Elevated terrain option was selected in the Terrain Options page. Here you can view the hill heights generated from AERMAP once you've processed your terrain data.

> You may modify the hill heights, but this is only recommended for advanced users.

- **Flagpole Heights:** This column is displayed only if the Flagpole Receptors terrain option was selected in the Receptor Summary page. Here you can specify the receptor height above ground in meters. Any missing values in the Flagpole Heights column will be interpreted by the U.S. EPA models as being equal to the default value specified for the Flagpole Receptors option. If no default height was specified, then the flagpole height will default to 0.0 meters.
- **Group Name:** The group name, FENCEPRI, is automatically assigned every time you add a new fenceline node. To change this group name click within the cell to display the ... button. You can then click on this button to open the Receptor Groups dialog.

Intermediate tab

In this tab you can easily assign intermediate receptors to your fenceline.



Cartesian Plant Boundary - Intermediate tab

In the **Place Plant Boundary Receptors Every** text box specify the spacing between intermediate receptors in meters and click on the 

button. Discrete receptors will be automatically placed between nodes, equally spaced. The distance between receptors will be rounded to an even multiple of the spacing you have specified. As an example, if between two fenceline nodes you have a distance of 100 meters and the specified spacing is 30 meters, then 3 intermediate receptors will be placed between the two primary nodes spaced every 25 meters.

The following buttons are available:

Click this button to select from these options:

- **Convert to Discrete** - coverts the current grid into discrete Cartesian receptors. This option is used when there is a need to eliminate one or more receptors from the grid, for example, eliminate receptors within a plant boundary.

- **Remove Plant Receptors** - Eliminates receptors that are within the plant boundary.

- **Import from File** - Imports discrete ARC Cartesian receptor parameters, such as X coordinate (x), Y coordinate (y), terrain elevations (ELEV), hill heights (HILL), and flagpole heights (FLAG) from a file.

- **Export to CSV File** - exports the receptor parameters, such as X coordinate (x), Y coordinate (y), terrain elevations (ELEV), hill heights (HILL), and flagpole heights (FLAG) to a file.

See Also: Receptor Import/Export Format.

- **Export Blanking File:** Select this option to export the plant boundary as a blanking file which allows you to mask a specified portion of your plot file.

If you defined a plant boundary in your AERMOD View project, a plant boundary blanking file is automatically generated which masks the contour plot lines inside your plant boundary. This Plant Blanking Boundary layer is placed in the Tree View - Overlays tab, you can chose whether or not to display this layer.

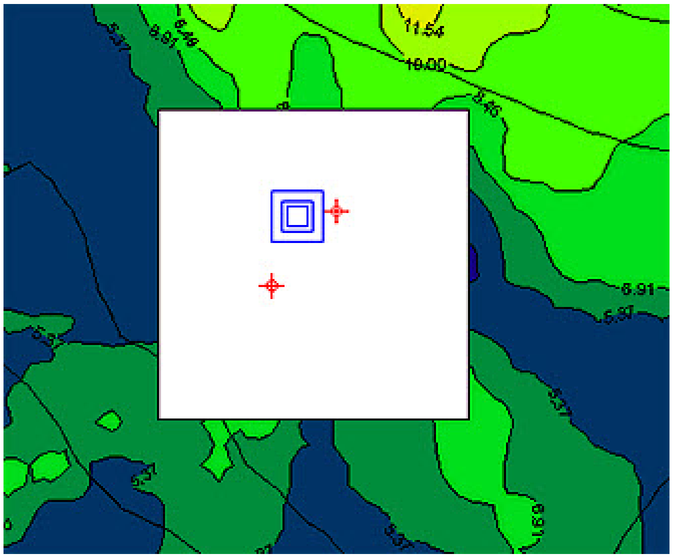
See Also: Blanking File Format.

The buttons at the bottom of the Cartesian Plant Boundary window, called the Record Navigator buttons, will help you in managing the information that is defined for your receptors.

Import Blanking Files

A blanking file allows you to mask a specified portion of your plot file. You can, for example, mask the area within your plant boundary and view only the contours beyond the plant boundary.

See Also: Cartesian Plant Boundary.



Imported Blanking File

How to Import a Blanking File

1. Select **Import - Blanking File...** from the menu. This option allows you to select a blanking file (*.rpb) to import into your project. This could be a plant boundary file generated from another project or a file you created yourself.

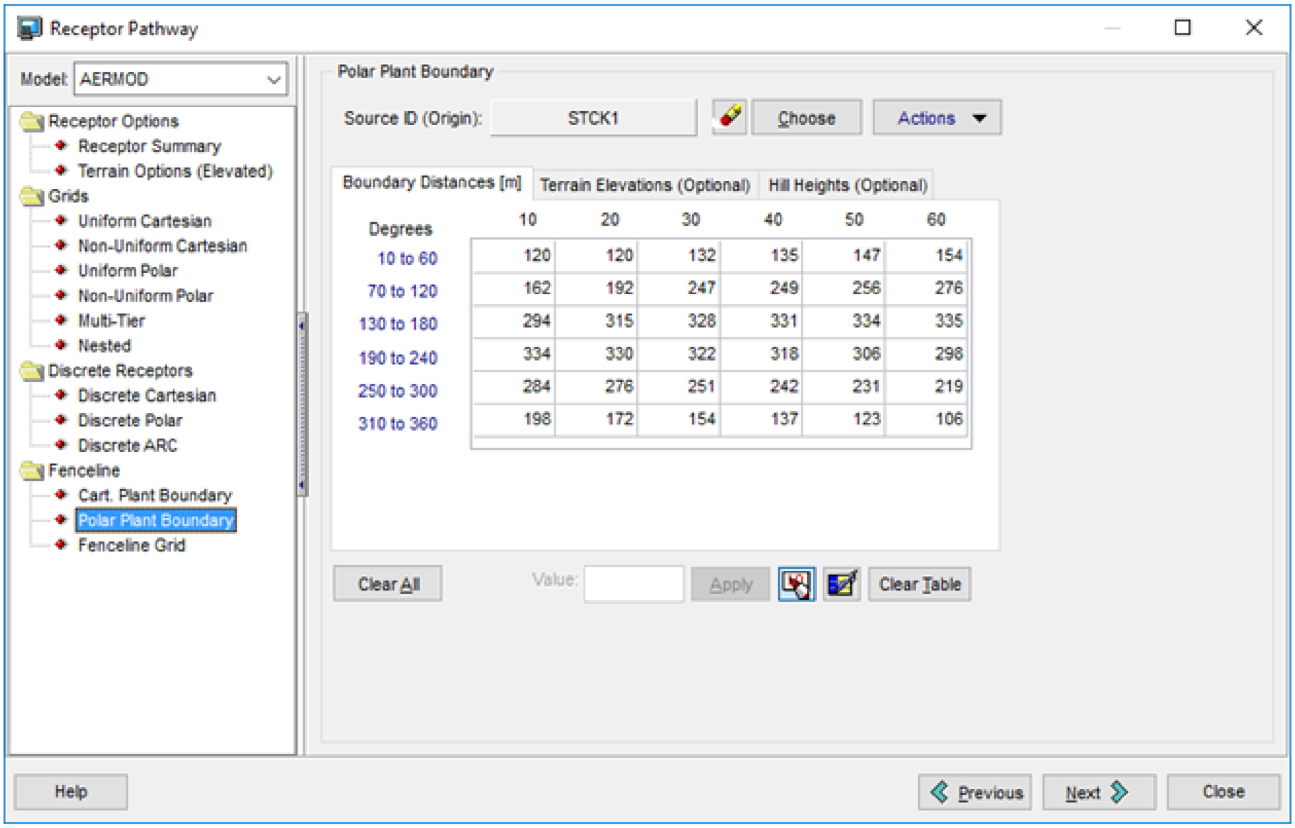
> See Also: Blanking File Format

2. The **Import Blanking File** dialog is displayed. Specify the name and path of the blanking file and click the Open button. The blanking file will be imported into your project.

> You can use the Overlay Control option to move layers above or below the masked area of the blanking file.

Polar Plant Boundary

In the **Polar Plant Boundary** page, you can define the boundaries of your plant using polar plant boundary receptors. For the polar plant boundary, you must define the distance from a specific source to the fenceline every 10 degrees for 36 directions. Select **Fenceline - Polar Plant Boundary** from the tree located on the left side of the Receptor Pathway dialog, to display the available options.

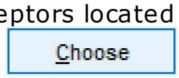


Receptor Pathway dialog - Polar Plant Boundary

Three tabs are available in the **Polar Plant Boundary** screen-

Boundary Distances tab

To define polar plant boundary distances, you must specify the following -

- **Source ID (Origin):** The location of the source will serve as the origin for 36 discrete polar receptors located at every 10 degrees around the source. To specify the Source ID, click on the  button to display the Source ID dialog from where you can select a source.
- **Boundary Distances:** These are the distances in meters for each of the directions, beginning with the 10 degree radial and incrementing every 10 degrees clockwise. You must input these values in the table paying attention to the exact direction of each cell. As you move from one cell to the other, AERMOD View displays the degrees for each direction on the table heading.

Terrain Elevations tab

In this optional tab you may define boundary receptor elevations. Terrain elevations are only needed if you have selected the **Elevated** terrain option in the Terrain Options page. Here you can input into the table the terrain elevations for each of the 36 boundary receptor points. The default unit is meters, however you can use feet if these were the units defined as the Terrain Elevations Unit. If you have processed and imported terrain elevation files, the elevations will be automatically displayed here.

> There is no option for inputting boundary receptor flagpole heights. The easiest way to input boundary receptors with flagpole receptor heights are to define them as Cartesian Plant Boundary Receptors.

> For applications where a uniform flagpole receptor height is used for all receptors, those flagpole receptor heights will also apply to any boundary receptors defined as **Polar Plant Boundary Receptors**.

Hill Heights tab

AERMOD Only In this optional tab you may view the hill heights generated from AERMAP once you've processed your terrain data.

> You may modify the hill heights, but this is only recommended for advanced users.

The following buttons are available:

Click this button to select from two options:

- **Convert to Cart. Plant Boundary** - converts Polar Plant Boundary Receptors into Cartesian Plant Boundary Receptors. Use this option when you need to eliminate receptors within plant boundary. AERMOD View only eliminates receptors within a plant boundary that was specified as Cartesian Plant Boundary.

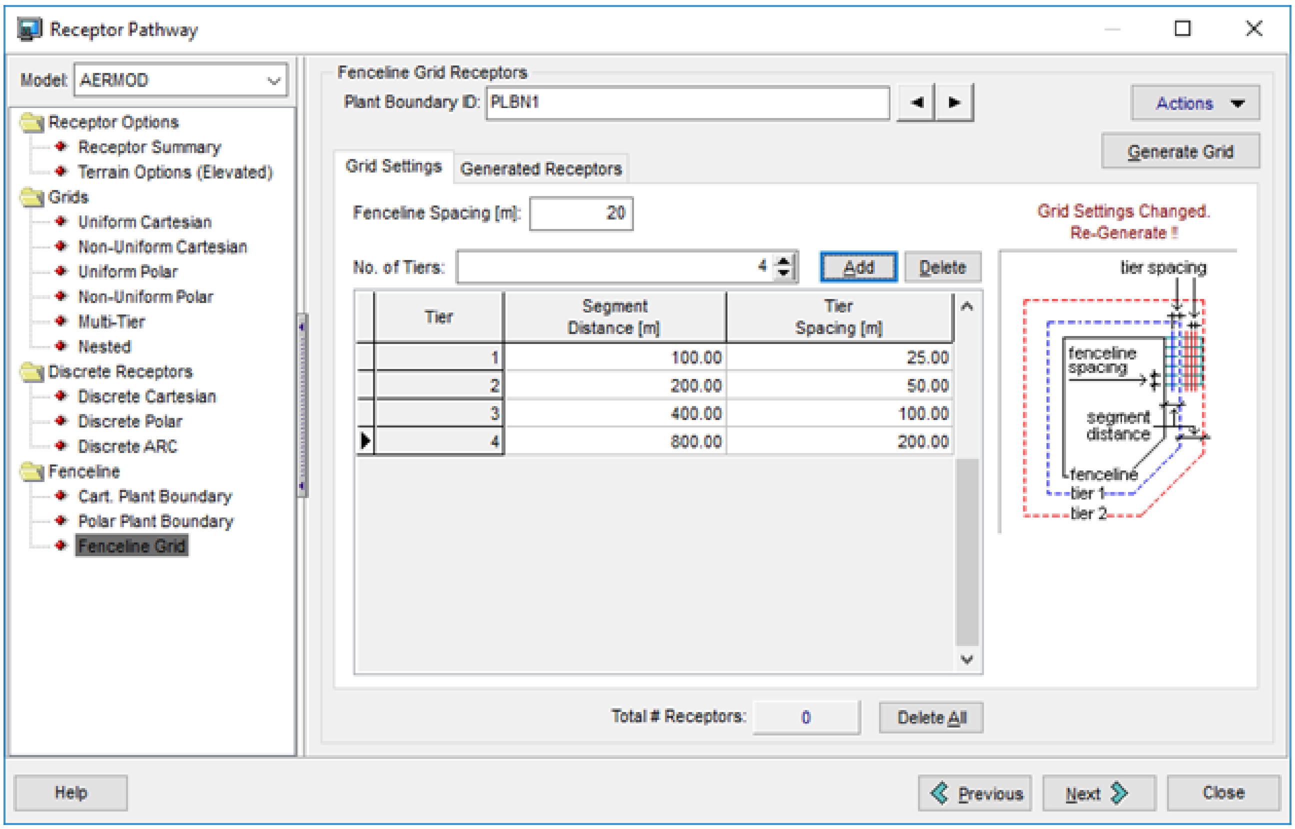
- **Export to CSV File** - exports the receptor parameters, such as X coordinate (x), Y coordinate (y), terrain elevations (ELEV), hill heights (HILL), and flagpole heights (FLAG) to a file.

> See Also: Receptor Import/Export Format.

Fenceline Grid

The **Fenceline Grid** screen allows you to define a grid around your [Cartesian Plant Boundary] or fenceline. The fenceline grid can have more than one tier with different grid spacings. Fenceline grid receptors are modeled in ISCST3, ISC-PRIME, and AERMOD as Discrete Cartesian Receptors.

Select **Fenceline - Fenceline Grid** from the tree located on the left side of the [Receptor Pathway] dialog, to display the available options. These options are available only if you have defined a Cartesian Plant Boundary.



Fenceline Grid

> The Fenceline Grid dialog is available only if you have created a [Cartesian Plant Boundary].

At the top of the dialog you can enter an identification name for the fenceline being defined. The ID can be up to 8 characters long and are always in upper case. Press the button to get a default ID automatically set up for you.

Two tabs are available in the Fenceline Grid dialog -

Grid Settings tab

To define the Fenceline Grid, you must specify the following:

- **Fenceline Spacing:** Here you must define the spacing between receptors along the fenceline.

- **No. of Tiers:** Specify here the number of tiers. Tiers are the segments with different grid spacing.

Add Adds a new tier to the table.

Delete Deletes the selected tier from the table.

- **Segment Distance:** You must specify the distance between tiers. AERMOD View places some default values for each tier you create. These values however can be changed to suit your modeling needs.

- **Tier Spacing:** For each tier, you must specify the spacing between receptors perpendicular to the fenceline. AERMOD View places some default values for each tier you create. These values however can be changed to suit your modeling needs.

The following buttons are available:

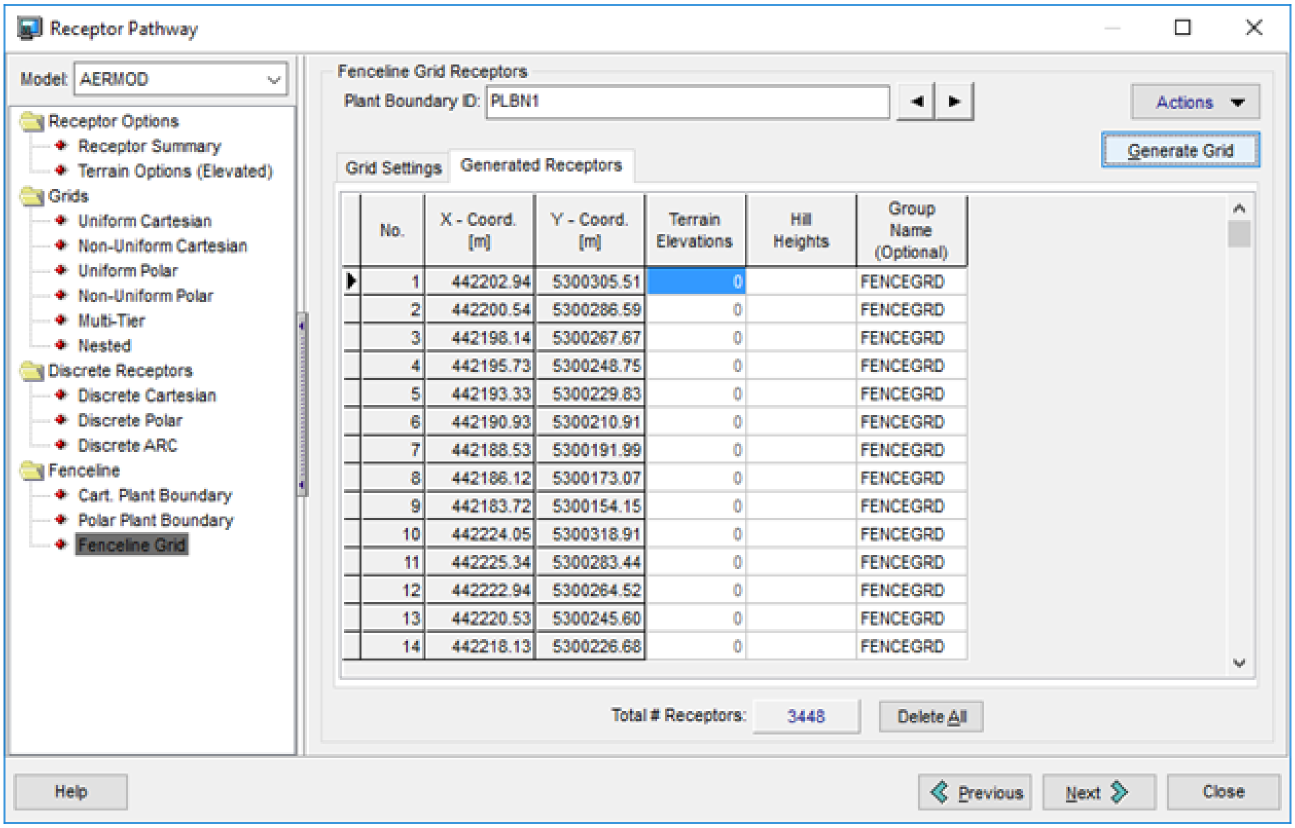
Actions Click this button to select from two options:

- **Convert to Discrete** - converts the fenceline grid into discrete receptors.

Generate Grid Press this button to generate the Fenceline Grid. When AERMOD finishes generating the grid, the total number of receptors created is automatically displayed. If you wish to delete the generated receptors click on the Delete All button.

Generated Receptors tab

This tab is automatically displayed once you have clicked on the Generate Grid button. The X and Y coordinates for all fenceline grid receptors are automatically calculated and displayed here.



Depending on the modeling options you have selected, the following additional parameters must be specified:

- **Terrain Elevations:** This column is displayed only if the Elevated terrain option was selected in the Terrain Options page. Here you can specify elevations for the fenceline grid receptors. If no terrain elevations are defined when you are modeling with Elevated terrain, then the elevations will default to 0.0 meters. If you have processed and imported terrain elevation files, the elevations will be automatically displayed here.

- **Hill Heights: AERMOD Only** This column is displayed only if the **Elevated** terrain option was selected in the Terrain Options page. Here you can view the hill heights generated from AERMAP once you've processed your terrain data.

> You may modify the hill heights, but this is only recommended for advanced users.

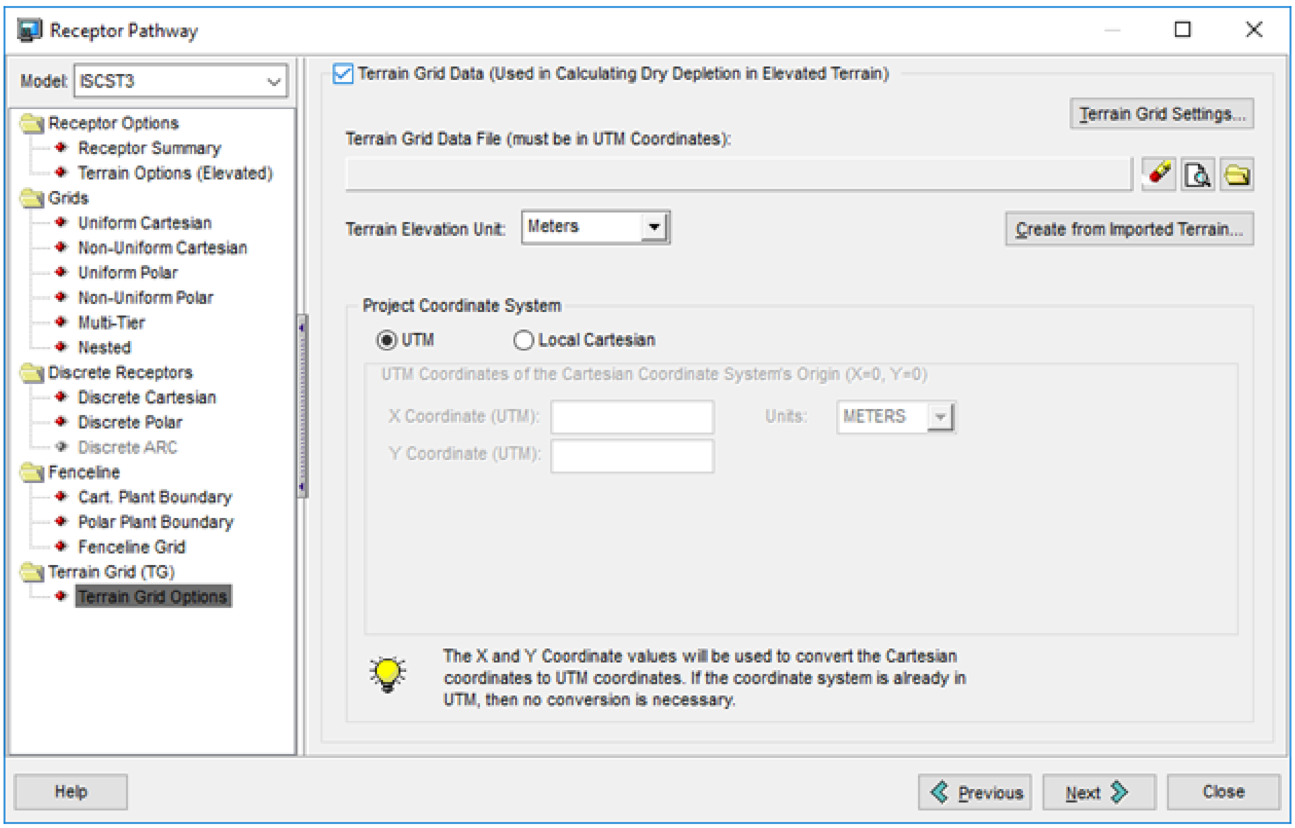
- **Flagpole Heights:** This column is displayed only if the Flagpole Receptors terrain option was selected in the Receptor Summary page. Here you can specify the receptor height above ground in meters. Any missing values in the Flagpole Heights column will be interpreted by the U.S. EPA models as being equal to the default value specified for the Flagpole Receptors option. If no default height was specified, then the flagpole height will default to 0.0 meters.

- **Group Name:** The group name, FENCEGRD, is automatically assigned to every fenceline grid receptor. To change this group name click within the cell to display the ... button. You can then click on this button to open the Receptor Groups dialog.

Terrain Grid Options

ISCST3 & ISC-PRIME Only

The **Terrain Grid Options** screen allows you to define the input terrain grid data used in calculating dry depletion in elevated or complex terrain. Select **Terrain Grid (TG) - Terrain Grid Options** from the tree located on the left side of the Receptor Pathway dialog, to display the available terrain grid options or select **Data | Terrain Grid Pathway...** from the menu.



Terrain Grid Options

The following options are found in this window:

- **Terrain Grid Data:** Check this box if you chose to use the terrain grid option and specify a terrain grid data input file.

The **Terrain Grid** option should only be used when modeling Dry Depletion in Elevated Terrain. If the Dry Removal option was selected in the Control Pathway and the Terrain Grid option is omitted, then the model will linearly interpolate between the source base elevation and the receptor elevation when calculating dry depletion. In all other cases when no dry depletion is calculated the model will neglect the terrain grid data provided in Terrain Grid Pathway.

- **Terrain Grid Data File:** Specify here the name and location of the file containing the terrain grid data by clicking on the button. Please note that the terrain grid data file must be in UTM coordinates. Grid information for the specified file will be displayed in the panel.

See Also: Terrain Grid Data File Format.

- **Terrain Elevation Unit:** Here you can select from the drop-down list the units of the elevations in your terrain grid file. The default units for terrain elevations are meters above MSL (Mean Sea Level).

Create from Imported Terrain... Click on this button to automatically generate a terrain grid file using the pre-processed terrain elevations for the area you are modeling. By default the terrain grid data file will be placed in your project folder with the project name (*.tgf). Grid information for the created file will also now be displayed.

Terrain Grid Settings... Click on this button to display the Terrain Grid Settings dialog where you can specify the grid settings for the generated terrain grid file.

Project Coordinate System

In this panel you must specify the **Project Coordinate System**. The following options are available -

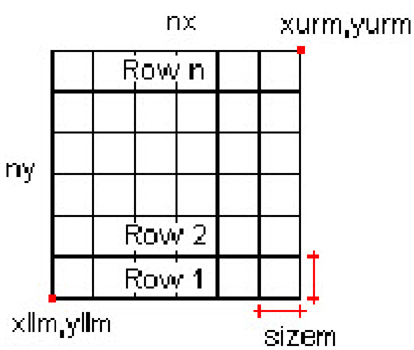
- **UTM:** Select UTM if you defined the source and receptor locations in UTM coordinates.
- **Local Cartesian:** Select this option if you defined source and receptor locations in another coordinate system. In this case, you also have to define the UTM coordinates of the Cartesian Coordinate System's Origin by specifying the following -
- **X Coordinate (UTM):** Enter here the value to be added to the x coordinate locations of the sources and receptors to translate them to UTM coordinates. The X Coordinate is the UTM coordinate for the origin of the source/receptor coordinate system.
- **Y Coordinate (UTM):** Enter here the value to be added to the y coordinate locations of the sources and receptors, to translate them to UTM coordinates. The Y Coordinate is the UTM coordinate for the origin of the source/receptor coordinate system.

- **Units:** Select from the drop-down list the unit used for the value input in the X Coordinate and Y Coordinate text boxes, either meters, feet, or kilometers. The default unit is meters.

Terrain Grid Data File Format

The terrain grid file contains one header record, followed by any number of data records. The file is read as a free-format ASCII file. The header record contains the following information:

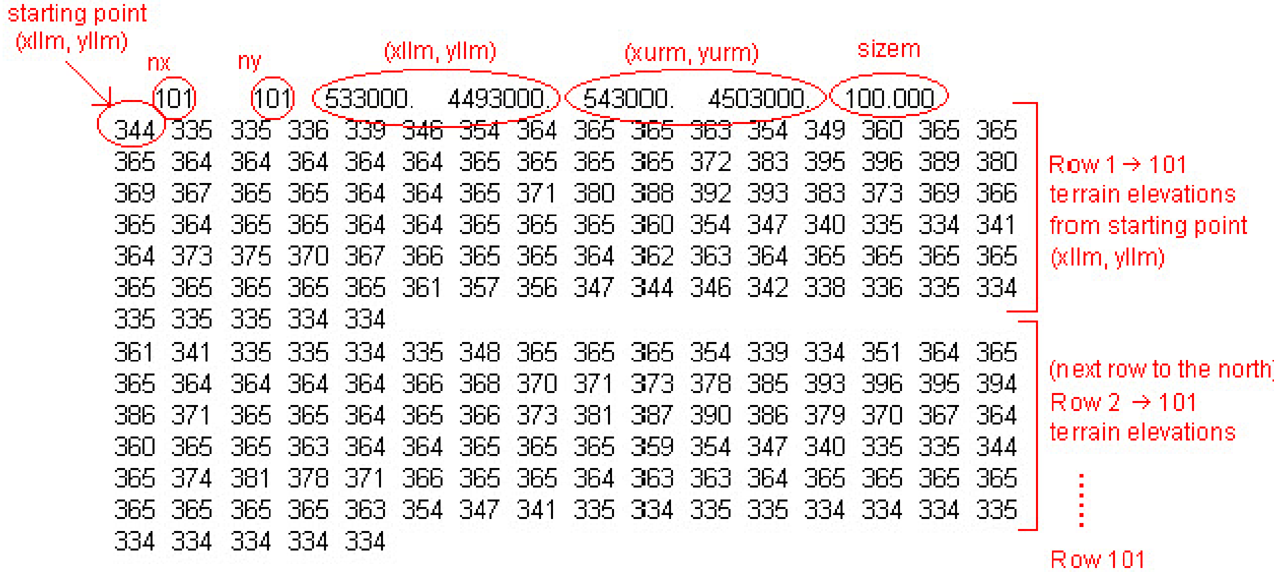
nx, ny, xllm, yllm, xurm, yurm, sizem



Where:

- **nx, ny:** Number of data points in x (Easting) and Y (Northing) directions.
- **xllm, yllm:** UTM coordinates (in meters) of the point at the lower left corner of the grid.
- **xurm, yurm:** UTM coordinates (in meters) of the point at the upper right corner of the grid.
- **sizem:** Spacing between grid points in both the x and y directions, in meters.

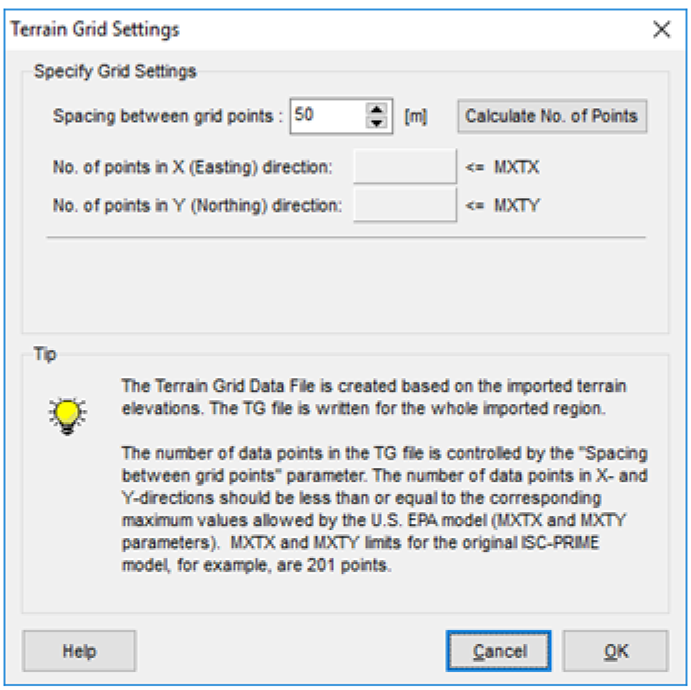
The data records are ordered by rows. The first row contains nx terrain elevations ordered from west to east, starting at point (xllm, yllm). Row 2 contains the data for the next row to the north of the grid. There are a total of ny rows of data in the terrain grid file. The maximum number of points in the terrain grid file is controlled by the MXTX and MXTY parameters. See below a terrain grid data file that contains terrain elevations for a 101x101 grid.



```
101 101 533000. 4493000. 543000. 4503000. 100.000
344 335 335 336 339 346 354 364 365 365 363 354 349 360 365 365
365 364 364 364 364 364 365 365 365 365 372 383 395 396 389 380
369 367 365 365 364 364 365 371 380 388 392 393 383 373 369 366
365 364 365 365 364 364 365 365 365 360 354 347 340 335 334 341
364 373 375 370 367 366 365 365 364 362 363 364 365 365 365 365
365 365 365 365 365 361 357 356 347 344 346 342 338 336 335 334
335 335 335 334 334
361 341 335 335 334 335 348 365 365 365 354 339 334 351 364 365
365 364 364 364 364 366 368 370 371 373 378 385 393 396 395 394
386 371 365 365 364 365 366 373 381 387 390 386 379 370 367 364
360 365 365 363 364 364 365 365 365 359 354 347 340 335 335 344
365 374 381 378 371 366 365 365 364 363 363 364 365 365 365 365
365 365 365 365 363 354 347 341 335 334 335 335 334 334 334 335
334 334 334 334 334
```


Terrain Grid Settings

The **Terrain Grid Settings** dialog allows you to change the default terrain grid spacing of 50 meters.



TG Grid Settings dialog

In the **Specify Grid Settings** panel, enter a value for the spacing between grid points and click on the Calculate No. of Points button. AERMOD View calculates the number of data points in the X and Y directions that will be created when the terrain grid is generated. It is important that you verify the calculated number of points in the X and Y directions against the parameters MXTX and MXTY. These are the parameters in the U.S. EPA models that control the maximum number of data points allowed in the terrain grid file. For the original U.S. EPA ISC-PRIME model, these parameters have a limit of 201 points. In the ISCST3 model, on the other hand, the MXTX and MXTY parameter limits are allocated dynamically at run time based on the number of grid points specified in your terrain grid file.

> The creation of the terrain grid file may take hours of processing depending on the number of data points and the domain extents of the pre-processed terrain elevations. Please make sure you have the right spacing and domain extents before creating the terrain grid file.

Create from Imported Terrain

When processing imported terrain elevations, such as USGS DEM, UK DTM, UK NTF, or XYZ files, AERMOD View will create an intermediate database table where it stores terrain elevations for the selected domain. When you press the Create from Imported Terrain... located in the Terrain Grid Options window, AERMOD View creates a Terrain Grid Data File using these pre-processed terrain elevations. AERMOD View will read the database table containing the pre-processed terrain elevation and resamples the data according to the spacing defined in the Terrain Grid Settings dialog. By default, AERMOD View calculates terrain elevations for the specified domain (the one specified when pre-processing the terrain elevations) for a grid spacing of 50 meters in the X and Y directions. You can change this grid spacing in the Terrain Grid Settings dialog.

Importing Receptors

You may import receptors into your project by clicking on the Import Receptors... button found in the Receptor Summary dialog or by selecting **Import | Receptors...** from the menu. The **Import Receptors From File** dialog is displayed allowing you to specify the name and location of the file containing the receptors to be imported.

AERMOD View can import receptors from any of the following file types:

- **ISCST3 Input Files (*.inp):** This file is generated when you run the ISCST3 model or when you select **Run | View Input File | ISCST3** from the menu.
- **ISC-PRIME Input Files (*.pin):** This file is generated when you run the ISC-PRIME model or when you select **Run | View Input File | ISC-PRIME** from the menu.
- **AERMOD Input Files (*.ain):** This file is generated when you run the AERMOD model or when you select **Run | View Input File | AERMOD** from the menu.

- **RE Pathway Partial Input Files (*.p2):** This file can be generated by selecting **Export | Pathway to File | RE Pathway to File** from the menu.

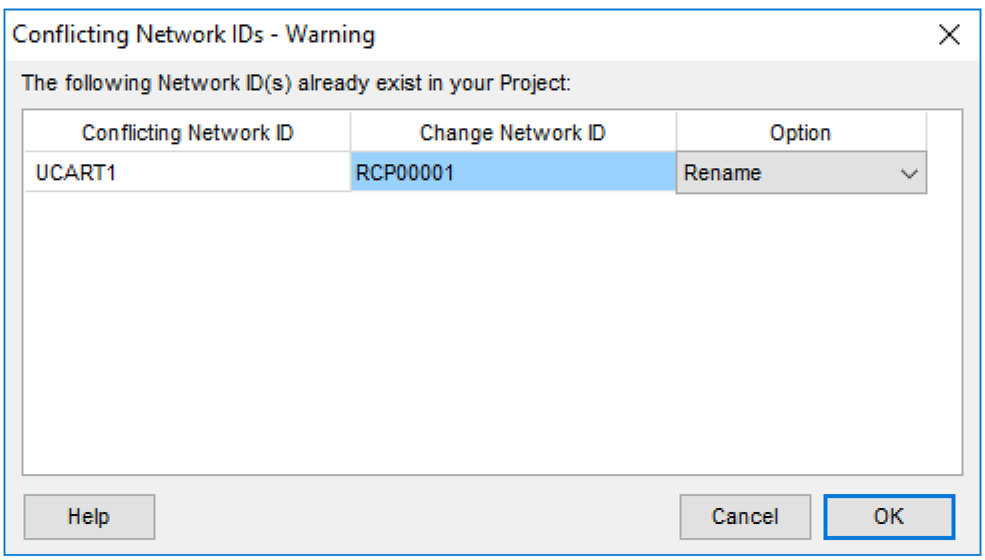
- **RE Included Files (*.rou):** This file is generated by AERMAP and contains all the receptor information for your AERMOD run in elevated terrain.

> For a detailed description of the standard ISCST3, ISC-PRIME, or AERMOD input file format see the U.S. EPA User's Guide for ISCST3 and AMS/EPA User's Guide for AERMOD.

During the receptor import process, if any Network IDs in the imported file conflict with existing Network IDs in your project, then the Conflicting Network IDs dialog is displayed. From this dialog you can specify if a network should be renamed or ignored.

Conflicting Network IDs

When you are importing receptors into an existing project, some of the receptor networks you are importing may have network IDs that conflict with the existing network IDs in your project. AERMOD View detects these receptor networks and lists them in the **Conflicting Network IDs** dialog.



Conflicting Network IDs dialog

In the **Change Network ID** column, AERMOD View places default replacement network IDs for each conflicting receptor network. You may change this network ID to another one if you wish. In the **Option** column, you may select from the drop-down list what you wish to do for each conflicting network ID -

- **Rename:** This option will import the receptor network with the alternate network ID specified in the **Change Network ID** column.

- **Do not Include:** Select this option if you do not wish to import the receptor network.

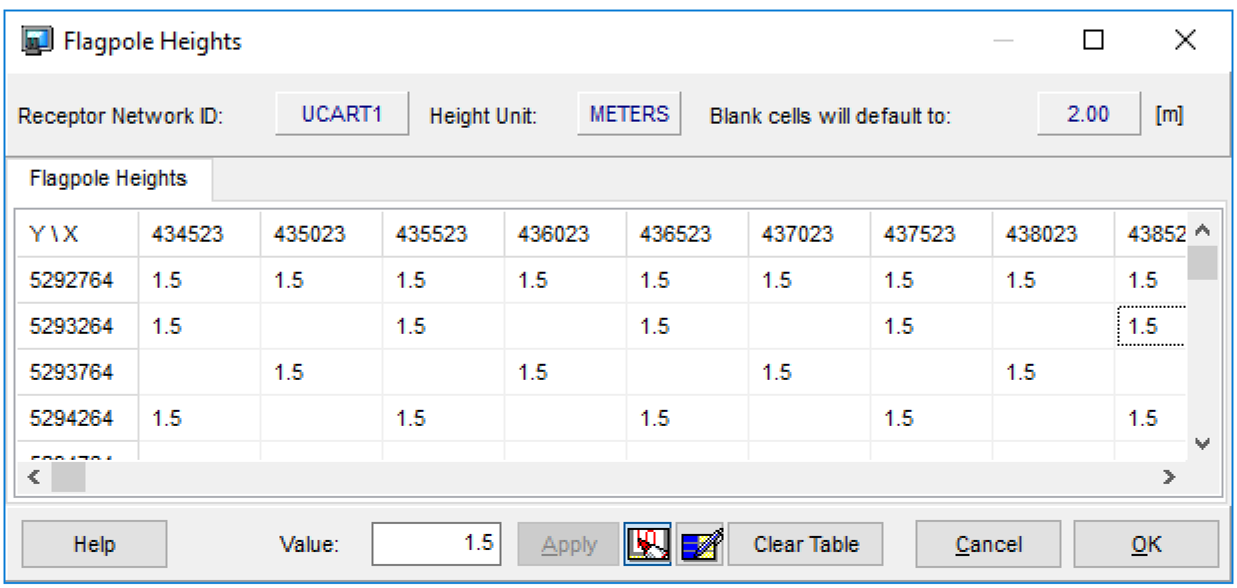
AERMOD View will then import the receptors according to the selected options.

Receptor Pathway Accessory Dialogs

This section describes accessory dialogs used by the Receptor Pathway.

Flagpole Heights

The **Flagpole Heights** dialog allows you to specify flagpole receptor heights for each grid point of the receptor network. You have access to this dialog by clicking on the Flagpole Heights button in the Uniform Cartesian Grid, Non-Uniform Cartesian Grid, Uniform Polar Grid or Non-Uniform Polar Grid windows.



Flagpole Heights dialog

At the top of the **Flagpole Heights** dialog the receptor network ID for which flagpole heights are to be specified is displayed along with the flagpole height unit and the default flagpole height that will be used for the cells (receptor locations) that are left blank.

The flagpole heights are entered in tabular format. The column headings contain the X-axis coordinate value for each point (node) and the number of columns is defined by the number of X-axis receptors you have defined for the current grid. The row headings contain the Y-axis coordinate value for each point (node) and the number of rows is defined by the number of Y-axis receptors you have defined for the current grid.

Each cell of the table corresponds to one grid point (node). You should input the flagpole height value for each grid point location.

You can assign the values in the table using the following controls:

Value: Type in the value you wish to assign to selected cells.

- Click to apply the contents of the **Value** field to the selected cells.

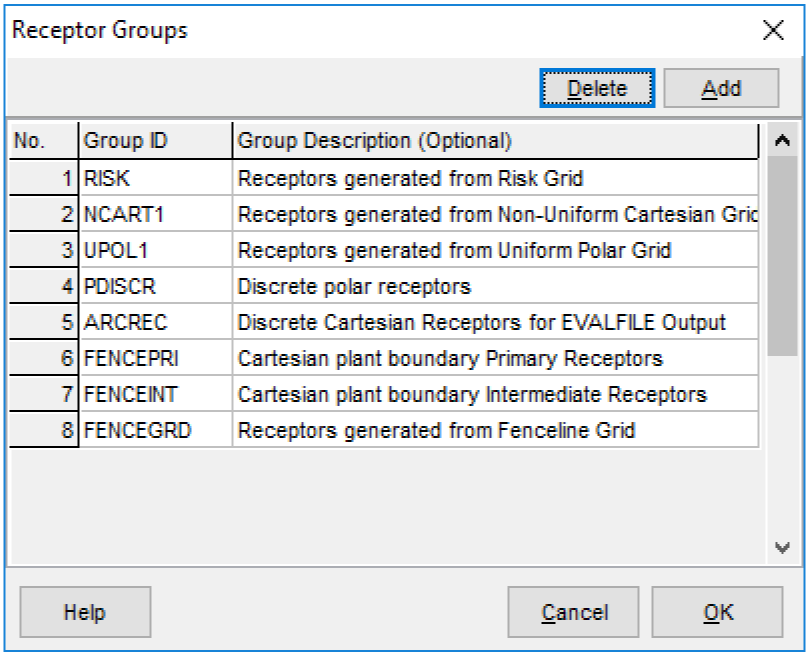
 - Click this button to be able to manually edit values in individual cells.

 - Click this button to be able to highlight/select entire rows of cells. Clicking on an already selected row will clear that selection.

- Click to clear all the values in the table.

Receptor Groups

The **Receptor Groups** dialog allows you to arrange receptors in groups. Some groups are created automatically, as is the case of the Multi-Tier Grid, Fenceline Grid, and Cartesian Plant Boundary receptors. Also, when converting network grids to discrete Cartesian receptors, the Network ID becomes the Group ID for the converted discrete receptor. This way, it is easier at a later time to distinguish which receptors were converted or not. Once the receptors are in the same group, they can be deleted as a group.

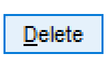


Receptor Groups dialog

> Receptors belonging to a group can be displayed in the drawing area with different colors and markers from other receptors within your project. For example, if you have sensitive receptors that you would like displayed differently from the rest of the receptors in your project you can do so in the Graphical Options - Color Mappings screen as long as a group has been created for these receptors.

The **Receptor Groups** dialog displays the **Group ID** and a description of the group. The following buttons are found in this dialog -

Click on this button to add a Receptor Group. Enter a group ID and if desired a group description.

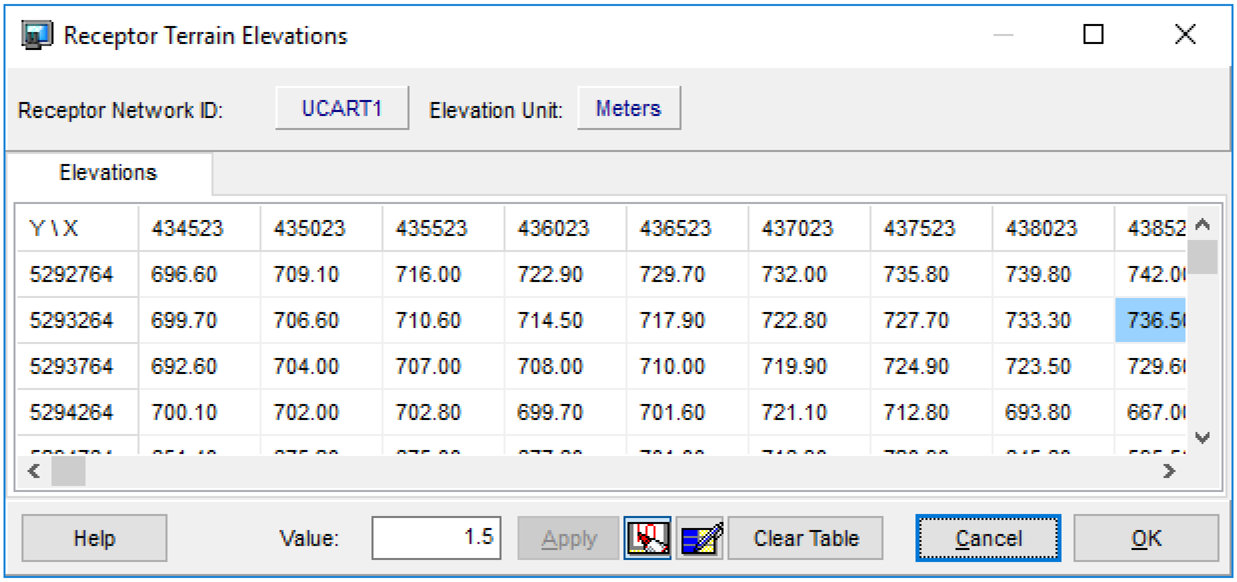


Click on this button to delete the selected Receptor Group. All receptors in the group will be deleted from the project.

Receptor Terrain Elevations

The **Receptor Terrain Elevations** dialog allows you to specify terrain elevations for each grid point of the receptor network. You have access to this dialog by clicking on the 

button. This dialog is available only if you are modeling with elevated terrain.



Receptor Terrain Elevations dialog

At the top of the **Receptor Terrain Elevations** dialog the receptor network ID for which terrain heights are to be specified is displayed along with the elevation unit that should be used when specifying terrain elevation for each receptor location. This unit is the one selected in the Terrain Options page.

If you have imported and processed digital elevation files in the Terrain Processor, the terrain elevations will be automatically displayed here.

> If you do import terrain elevations, the units will always be meters. This is due to the fact that AERMOD View always converts the USGS DEMs from feet to meters.

The terrain heights are entered in tabular format. The column headings contain the X-axis coordinate value for each point (node) and the number of columns is defined by the number of X-axis receptors you have defined for the current grid. The row headings contain the Y-axis coordinate value for each point (node) and the number of rows is defined by the number of Y-axis receptors you have defined for the current grid. Each cell of the table corresponds to one grid point (node). You should input the terrain elevation value for each grid point location.

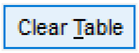
You can assign the values in the table using the following controls:

Value: Type in the value you wish to assign to selected cells.

- Click to apply the contents of the **Value** field to the selected cells.

 - Click this button to be able to manually edit values in individual cells.

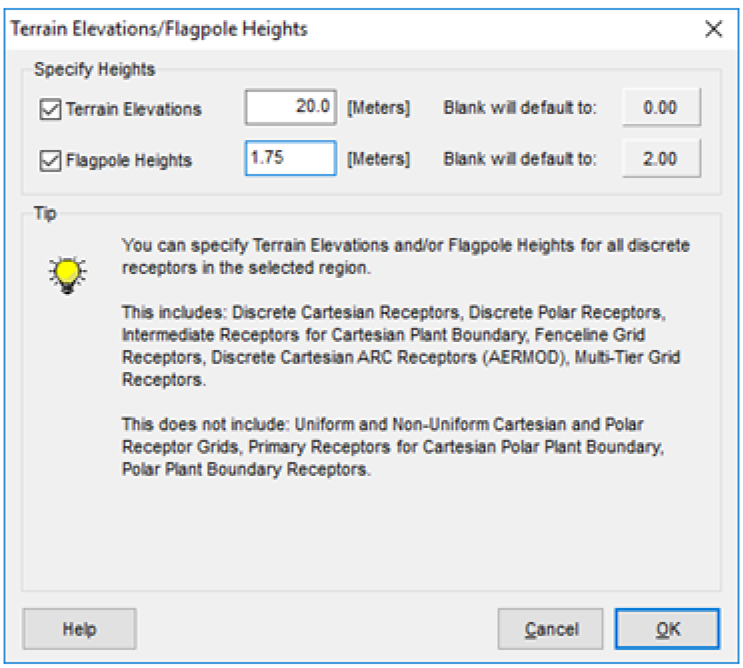
 - Click this button to be able to highlight/select entire rows of cells. Clicking on an already selected row will clear that selection.



- Click to clear all the values in the table.

Terrain Elevations\Flagpole Heights

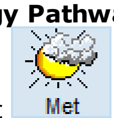
The **Terrain Elevations/Flagpole Heights** dialog is displayed when you use Elevations/Flagpole Heights Tool to graphically define terrain elevations and/or flagpole heights for discrete receptors.



Terrain Elevations/Flagpole Heights dialog

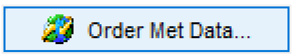
Check the **Terrain Elevations** or **Flagpole Heights** box and then specify a value in the corresponding text box. If left blank, the terrain elevations value will default to 0m and the flagpole heights will default to the value specified in the Receptor Summary window. The specified values will be applied to all discrete receptors that are found within the defined polygon.

Meteorology Pathway

The **Meteorology Pathway** allows you to specify the input meteorological data file and other meteorological variables, including the period to process for the meteorological files. You have access to the **Meteorology Pathway** dialog by selecting **Data | Meteorology Pathway...** from the menu or by clicking on the Met 

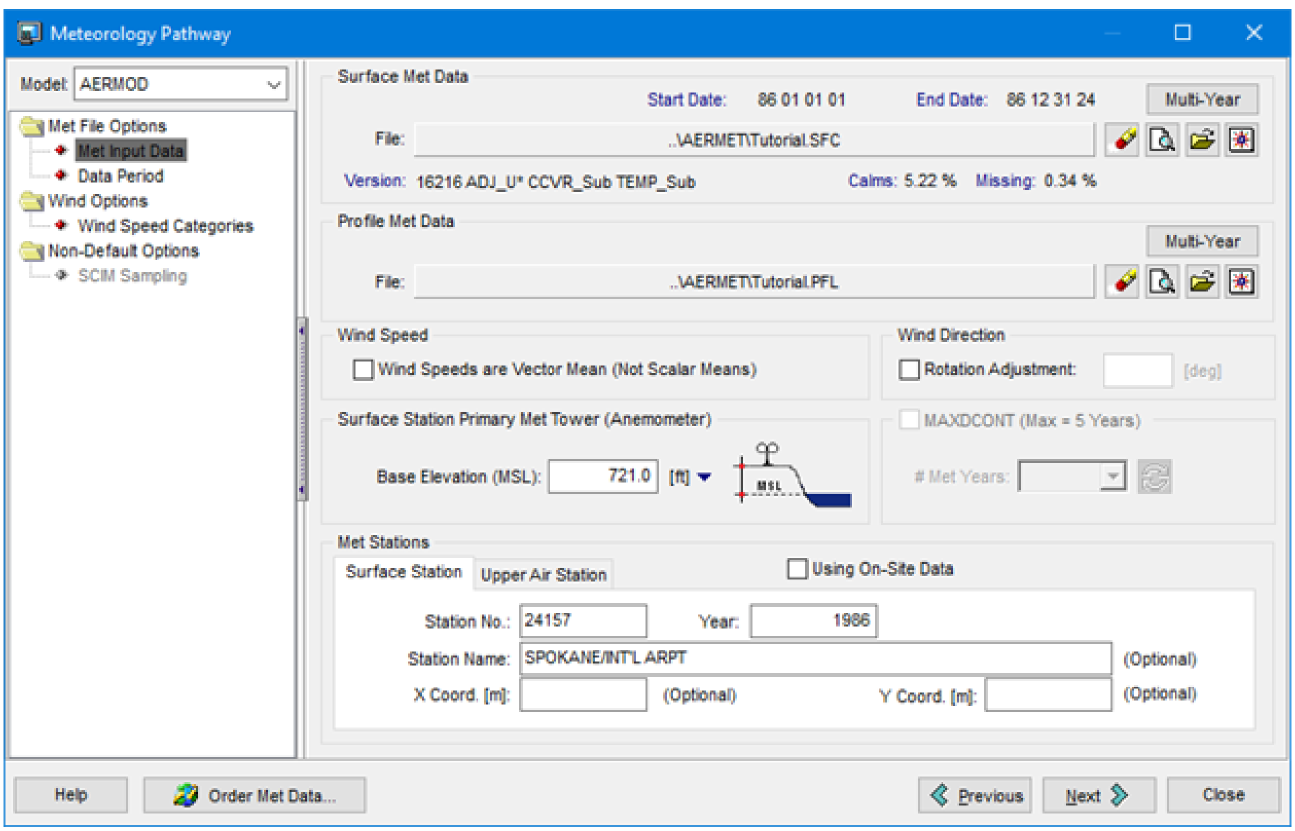
menu toolbar button.

The **Meteorology Pathway** dialog uses a two-pane view. The tree located on the left side of the dialog is used for navigation and item selection. Select an item (marked as ◆) in the tree to display the available options on the right panel.



- Press this button to link to Lakes Environmental's meteorological data services page. Order custom data from the prognostic meteorological models WRF or MM5 in AERMET-ready format. Modelers in the United States can order AERMOD-ready (.SFC and .PFL) format data from local surface and upper air meteorological stations.

> This option requires an internet connection.



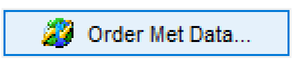
Meteorology Pathway dialog

In the **Meteorology Pathway** you can define the following:

- Met Input Data
- Data Period
- Wind Speed Categories
- Wind Profile Exponents
- Vertical Temperature Gradients
- SCIM Sampling

Met Input Data

The Met Input Data screen allows you to specify the meteorological data file and information on the meteorological stations. Meteorology Pathway - Met Input Data from the tree located on the left side of the Meteorology Pathway dialog, to display the available met input options.



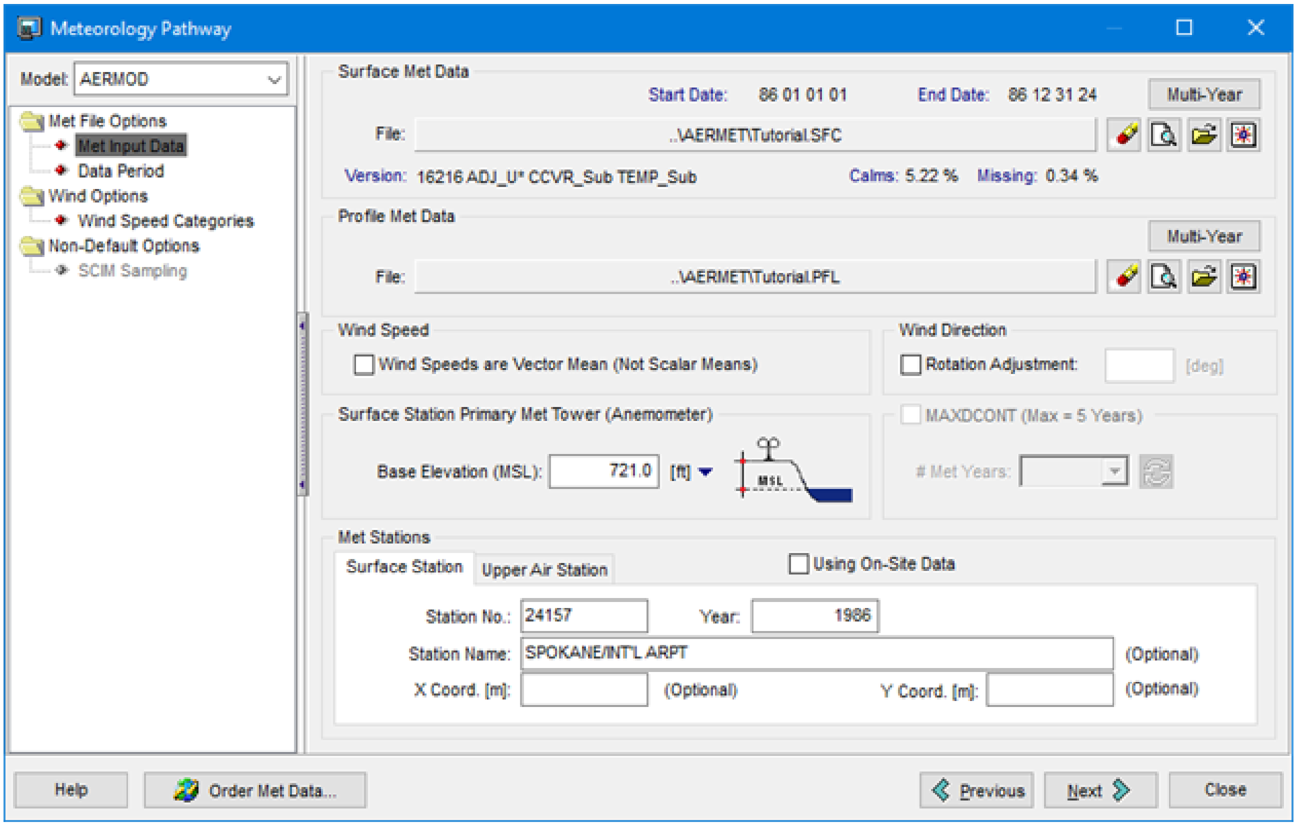
- Press this button to link to Lakes Environmental’s meteorological data services page. Order custom data from the prognostic meteorological models WRF or MM5 in AERMET-ready format. Modelers in the United States can order AERMOD-ready (.SFC and .PFL) format data from local surface and upper air meteorological stations.

> This option requires an internet connection.

The inputs specified in the **Met Input Data** dialog differ slightly depending on the model being used. See below the required inputs for each model:

AERMOD

The AERMOD model requires the following met inputs to be specified:



Meteorology Pathway - Met Input Data - AERMOD

Surface Met Data

AERMOD uses hourly meteorological data from separate surface and profile data files. In this panel you must specify the surface met data file you will be using for your project.

- **Start Date/End Date:** Start and end date for the data time period in format **YY MM DD HH**.
- **File Name:** The surface met data file contains observed and calculated surface variables, one record per hour.

> The full path of the specified met data file will be displayed on the panel. If the specified met data file is located in the same path as the project, then only the file name will be displayed.

- **Multi-Year:** Click this button to display the **Muti-Year Files Utility** from where you can specify multiple year .SFC files to be combined into one single .SFC file. For more information on how to use this utility see the Muti-Year Files Utility help topic.

- **Available options:**

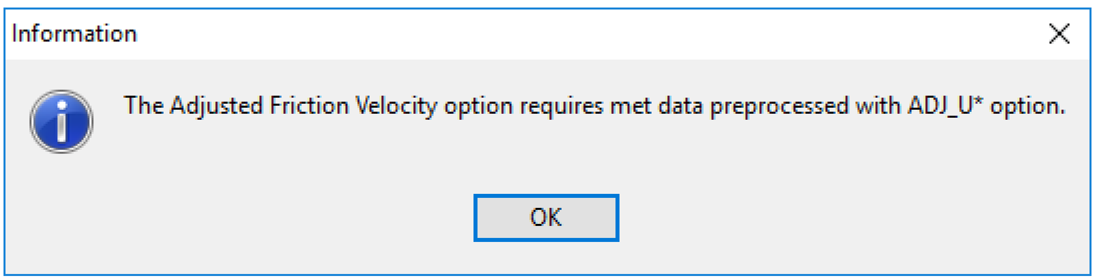
 - Click this button to clear the currently selected surface file.

 - Click this button to view the contents of the file. Select **Text** to view the contents in the default Text Editor, or **Grid** to launch Met Viewer utility.

 - Click this button to select a surface file.

 - Click this button to open selected surface file in WRPLOT View.

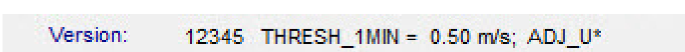
If the met data was calculated using **Adjust Surface Friction Velocity (ADJ_U*)** option, the following message will appear when the .SFC file is selected:



Clicking **Yes** will select **Non-Default Options** in the Dispersion Options of the Control Pathway and check the box for **Adjusted Friction Velocity (u*) in AERMET (ADJ_U*)**.

See Also: Surface Met Data File Format.

You will also see the following data displayed:



- **Version:** Displays the version of the met file, which corresponds to the version of the US EPA AERMET model used to preprocess the met data. The flags to indicate whether the wind speed threshold (THRESH_1MIN including the threshold value) and the adjust surface friction velocity (ADJ_U*) options were used.

AERMOD version **16216r** will no longer run with met data preprocessed with **AERMET** version **11059**. AERMOD 16216r will run with met data

> preprocessed with **AERMET 12345**, **13350** and **14134** but a warning message will be generated. It is recommended that you update your met data files by pre-processing them using the latest **AERMET** version (Version 16216 as of December 2016).

- **Calms:** Displays the percentage of calm hours in the .SFC met file.
- **Missing:** Displays the percentage of missing hours in the .SFC met file.

> If the number of Missing values exceeds 10%, the numbers will be highlighted:
>
> Calms: 0.89 % Missing: 17.03 %
>
> If more than 10% of values are missing, when the generated met files are used in AERMOD it will give a warning that the calculated data may not be acceptable for regulatory applications.

Profile Met Data

AERMOD uses hourly meteorological data from separate surface and profile data files. In this panel you must specify the profile met data file you will be using for your project.

> If the **Profile Met Data File** has the same name as the **Surface File** and is in the same location, it will be automatically loaded when the Surface file is specified.

- **File Name:** The profile met data file contains the observations made at each level of an on-site tower, or the one level observations taken from NWS data.

> The full path of the specified met data file will be displayed on the panel. If the specified met data file is located in the same path as the project, then only the file name will be displayed.

- **Multi-Year:** Click this button to display the **Muti-Year Files Utility** from where you can specify multiple year .PFL files to be combined into one single .PFL file. For more information on how to use this utility see the Muti-Year Files Utility help topic.
- **Available options:**

- Click this button to clear the currently selected profile file.

- Click this button to view the contents of the file. Select **Text** to view the contents in the default Text Editor, or **Grid** to launch Met Viewer utility.

- Click this button to select a profile file.

- Click this button to open selected profile file in WRPLOT View.

See Also: Profile Met Data File Format.

Wind Speed (Vector Mean or Scalar Mean)

Introduced with the US EPA AERMOD Version 16216r, you can now specify if the wind speed in your meteorological data file is vector (resultant) mean or scalar mean. If selecting this option, make sure you can confirm that the wind speed for your met data is vector mean instead of scalar mean as the AERMOD model will apply adjustments if this option is selected. Wind speeds derived from NWS or FAA airport data, for example, are scalar mean.

Surface Station Primary Met Tower (Anemometer)

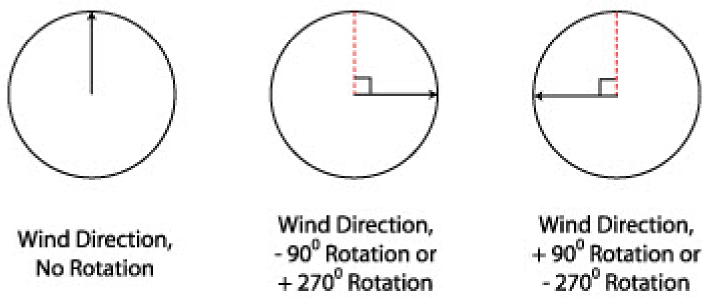
This option allows you to specify the base elevation above mean sea-level (MSL) of the surface station's primary met tower. If you wish to specify the base elevation in feet, click the down arrow beside the **[m]** indicator, and choose **ft**.

This does not denote the height of the anemometer.

Wind Direction (Rotation Adjustment)

This option allows you to correct the meteorological data for wind direction alignment problems. All input wind directions or flow vectors are rotated by a user-specified amount. Since the model results at particular receptor locations are often quite sensitive to the transport wind direction, this option should be used only with extreme caution and with clear justification.

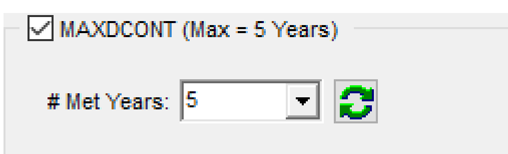
- **Rotation Adjustment [deg]:** Specify here the angle in degrees to rotate the input wind direction measurements. The specified value will be subtracted from the wind direction measurements. This option may be used to correct for known (and documented) calibration errors. Since the Short Term models use the flow vector (direction toward which the wind is blowing) as the basic input, the rotation adjustment value may also be used to convert input data as wind direction, (direction from which the wind is blowing), to flow vector by setting the rotation equal to 180 degrees.



MAXDCONT

> This option is only available if **NAAQS Contribution Files (MAXDCONT)** option is checked in the **Contributions** tab of the US EPA NAAQS Options section of Output Pathway.

- **# Met Years:** this field displays the number of years of met data that is being used for the model run. This option was introduced in AERMOD model version 12060 to minimize memory storage requirements for AERMOD MAXDCONT routines in case you are using less than 5 years of met data.



Surface, Upper Air & On-Site Meteorological Stations

The following information is requested for the surface, upper air and on-site meteorological stations -

- **Station No.:** This is the station number for the meteorological station. For National Weather Service (NWS) stations this is a 5-digit WBAN (Weather Bureau Army Navy) number.

- **Year:** This is the year of meteorological data being processed. If you are using more than one year of met data, then you should specify the first year of the met data.

- **Station Name:** This is an optional parameter specifying the name of the station. The model accepts a name of up to 40 characters including spaces. Spaces in the name will be substituted by underscores in the input file.

> AERMOD View will read the station number and year from the met file header and display it automatically. If the meteorological station is a NWS station, then AERMOD View will also display the station name.

- **X Coord.:** If you wish, you can specify here the X coordinate for the location of the meteorological station. At the present time, the U.S. EPA models do not use the station locations (X & Y Coordinates).

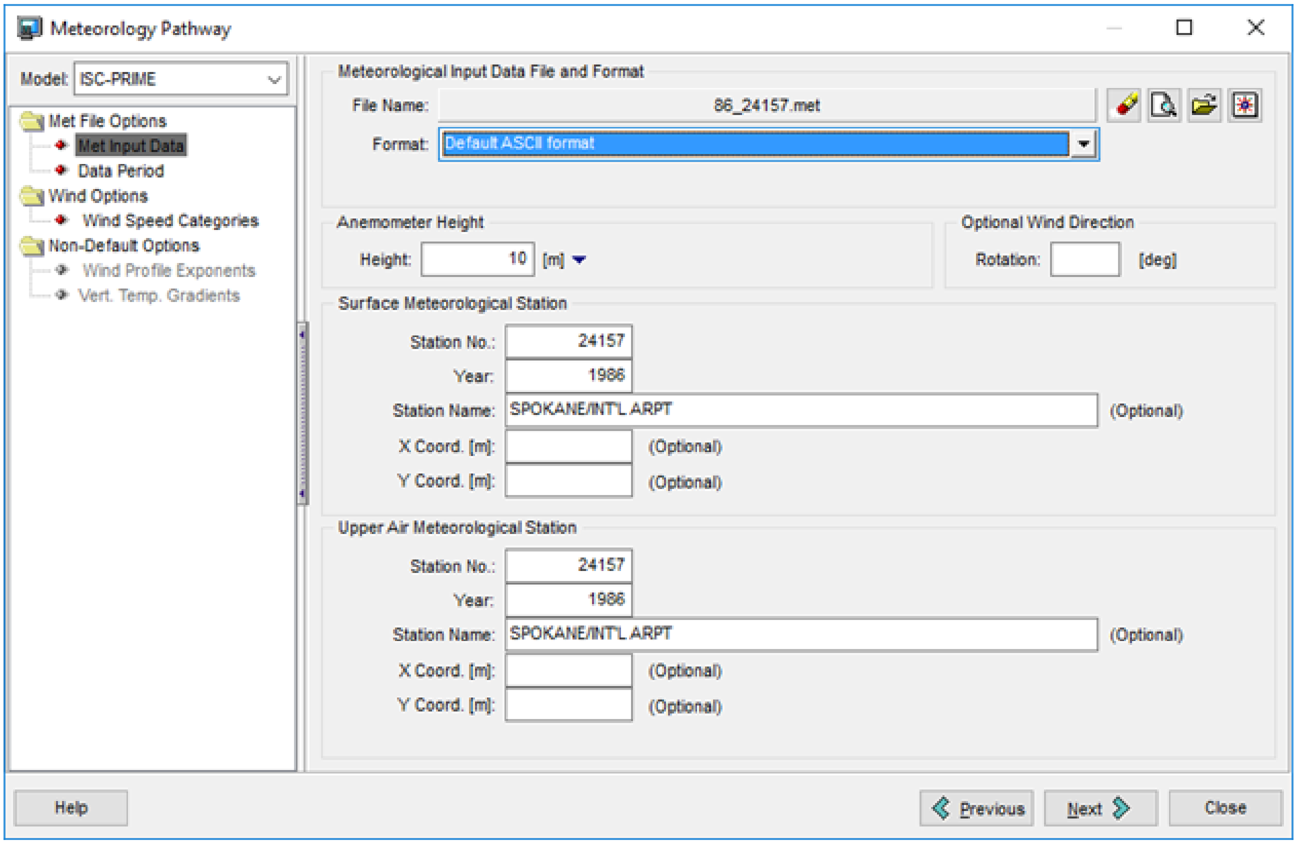
- **Y Coord.:** If you wish, you can specify here the Y coordinate for the location of the meteorological station. At the present time, the U.S. EPA models do not use the station locations (X & Y Coordinates).

- **Using On-Site Data:** This box must be checked if your pre-processed met data includes on-site data. If this box is checked, than you have access to the On-Site Station tab where you can specify the parameters described above.

> It is important that the station number and met data year be provided correctly since the EPA model will compare these parameters with the ones provided in the meteorological data files. If a mismatch is found, non-fatal warning messages will be issued.

ISCST3 & ISC-PRIME

The **ISCST3 & ISC-PRIME** models require the following met inputs to be specified -



Meteorology Pathway - Met Input Data - ISC-PRIME/ISCST3

Meteorological Input Data File and Format

ISCST3 & ISC-PRIME uses hourly meteorological data as one of the basic model inputs. The meteorological data is read into the models from a separate data file. In this panel you must specify the met data file you will be using for your project.

- **File Name:** Press the  button to specify the name and location of the meteorological data file to be used. The full path of the specified met data file will be displayed on the panel. If the specified met data file is located in the same path as the project, then only the file name will be displayed.

- **Format:** Select from the drop-down list the format of the meteorological data file specified. The following formats are available for selection -

- **Default ASCII format:** This is the default ASCII format for a sequential hourly file.

See Also: ASCII File Format.

- **Specify FORTRAN ASCII read format:** If this option is selected than the FORTRAN read format for an ASCII sequential hourly file should be specified in the User Format field. The user specified ASCII format can be up to 60 characters long and may be used to specify the READ format for files that differ from the default format.

- **FREE-formatted reads:** This is the free-formatted reads for an ASCII sequential hourly file.

- **UNFORMatted file (RAMMET or MPRM):** Unformatted file generated by the RAMMET or MPRM preprocessors. The UNFORMatted file format is no longer supported by ISCST3 (dated 00101) but is still currently provided in ISC-PRIME.

> If Dry Deposition and/or Wet Deposition were selected on the Control Pathway, the Unformatted file (RAMMET or MPRM)) cannot be used. This is due to the fact that the deposition algorithms require additional meteorological variables not found in this file format.

> See Also: Unformatted (Binary) File Format.

- **CARD image:** “CARD image” data using a default ASCII format. This option differs from option 1 by the addition of hourly wind profile exponents and hourly vertical potential temperature gradients in the input file.

> If the Gas Dry Deposition option was selected in the Control Pathway, then you must use the CARD Image format. This is due to the fact that PCRAMMET does not support preprocessing for Gas Dry Deposition, thus MPRM must be used.

> See Also: MPRM file format with CARD option.

Anemometer Height

In this panel you must specify the height above ground at which the wind speed data was collected for the met data being used. The models will adjust the input wind speeds from the anemometer height to the release height. This way, the accurate specification of anemometer height is important to obtaining the correct model results.

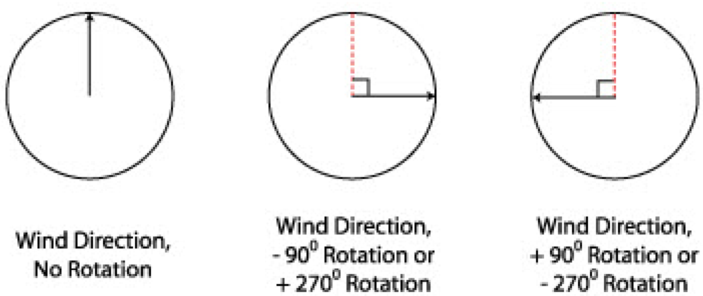
- **Height:** This is the height of the anemometer measurement above ground. You can select the measurement units of meters or feet.

> For National Weather Service (NWS) data, you should check records such as the Local Climatological Data summary report for the particular station to determine the correct anemometer height for the data period used in the modeling, since the anemometer location and height may change over time.

Optional Wind Direction

This option allows you to correct the meteorological data for wind direction alignment problems. All input wind directions or flow vectors are rotated by a user-specified amount. Since the model results at particular receptor locations are often quite sensitive to the transport wind direction, this option should be used only with extreme caution and with clear justification.

- **Rotation:** Specify here the angle in degrees to rotate the input wind direction measurements. The specified value will be subtracted from the wind direction measurements. This option may be used to correct for known (and documented) calibration errors, or to adjust for the alignment of a valley if the meteorological station is located in a valley with a different alignment than the source location. Since the Short Term models use the flow vector (direction toward which the wind is blowing) as the basic input, the rotate value may also be used to convert input data as wind direction, (direction from which the wind is blowing), to flow vector by setting the rotation equal to 180 degrees.



Surface & Upper Air Meteorological Stations

The following information is requested for both the surface and the upper air meteorological stations -

- **Station No.:** This is the station number for the meteorological station. For National Weather Service (NWS) stations this is a 5-digit WBAN (Weather Bureau Army Navy) number.

- **Year:** This is the year of meteorological data being processed. If you are using more than one year of met data, then you should specify the first year of the met data.

- **Station Name:** This is an optional parameter specifying the name of the station. The model accepts a name of up to 40 characters including spaces. Spaces in the name will be substituted by underscores in the input file.

> AERMOD View will read the station number and year from the met file header and display it automatically. If the meteorological station is a NWS station, then AERMOD View will also display the station name.

- **X Coord.:** If you wish, you can specify here the X coordinate for the location of the meteorological station. At the present time, the U.S. EPA models do not use the station locations (X & Y Coordinates).

- **Y Coord.:** If you wish, you can specify here the Y coordinate for the location of the meteorological station. At the present time, the U.S. EPA models do not use the station locations (X & Y Coordinates).

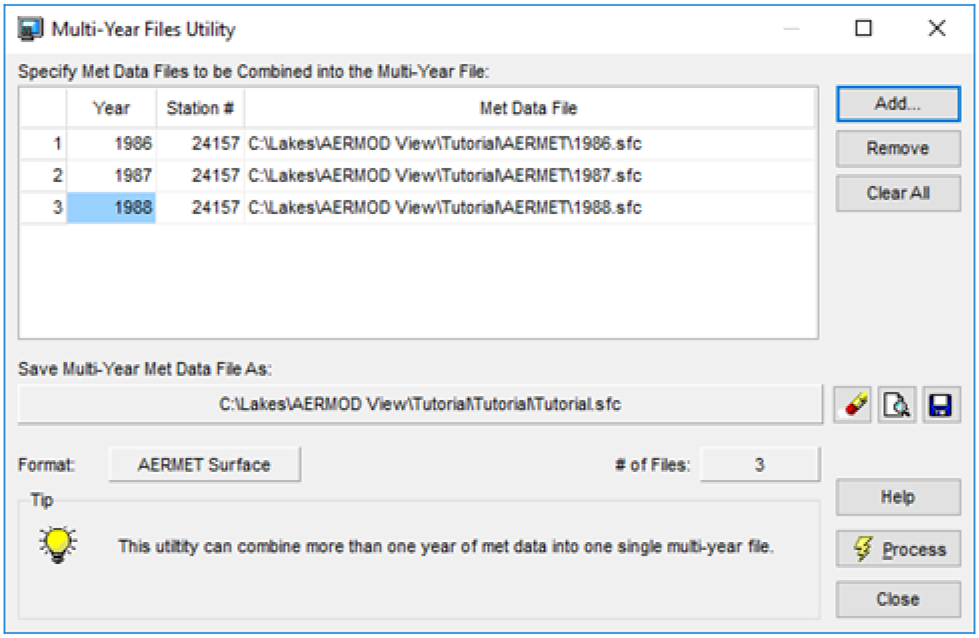
> It is important that the station number and met data year be provided correctly since the EPA model will compare these parameters with the ones provided in the meteorological data files. If a mismatch is found, non-fatal warning messages will be issued.

Multi-Year Met Data File Utility

The **Multi-Year Files Utility** allows you to combine more than one year of meteorological data files into one single file which can then be used for your AERMOD View projects.

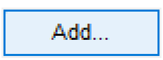
You can access this utility by clicking the **Multi-Year** button in the Meteorology Pathway | Met Input Data.

> The files you are combining must all be for the same format and for the same station ID.

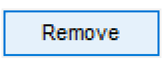


Multi-Year Files Utility

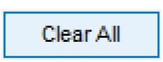
The following options are available -



- Click this button to add a data file.



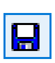
- Click this button to remove selected data file.

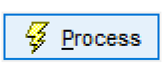


- Click this button to remove all data files.

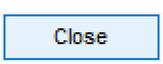

- Click this button to clear the file name and path.


- Click this button to view the merged file in text editor.


- Click this button to specify file name and path.



- Click this button to generate merged met file.



- Click this button to close the utility and return to Met Pathway | Met Input Data.

How to Create a Multi-Year Met Data File:

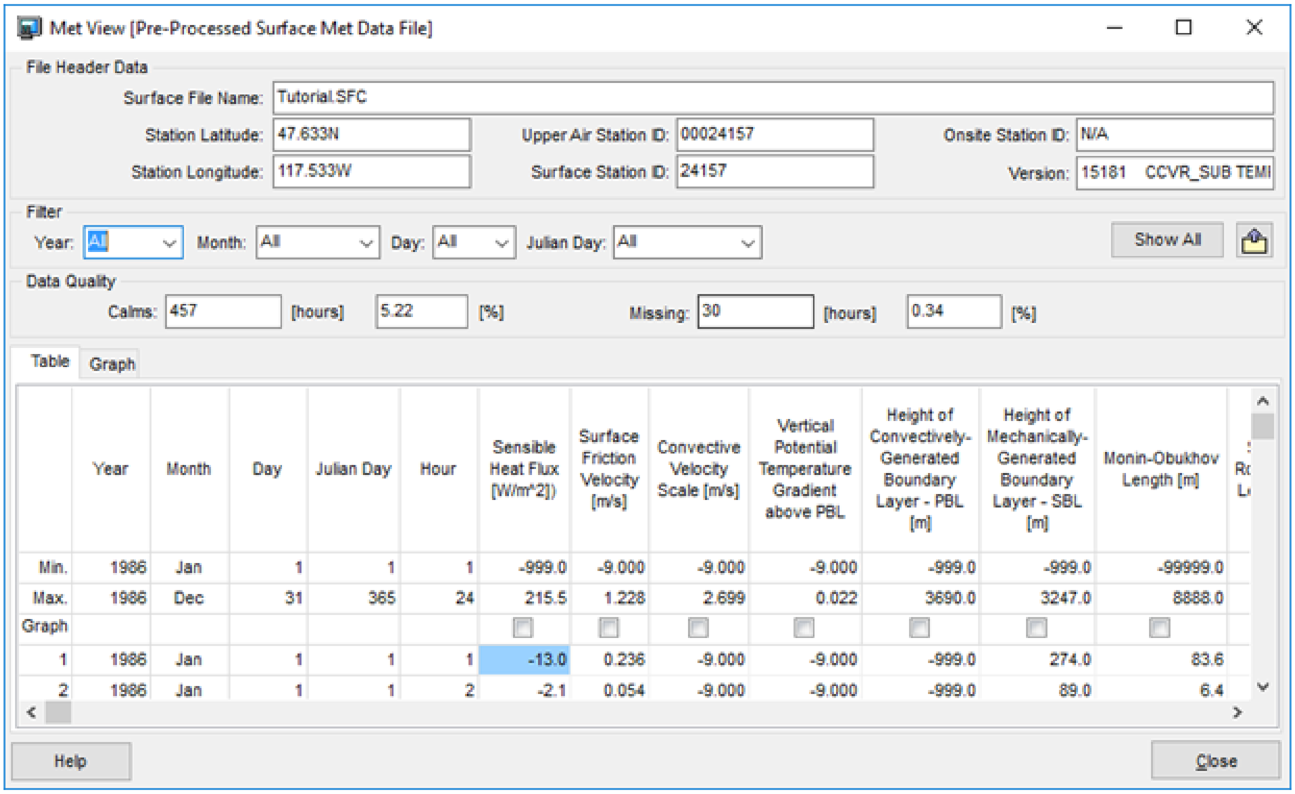
1. Click on the Add... button to open the **Specify Met Data File** dialog where you specify the met data files to be combined. The year, station #, and file path of the added met data files will then be displayed in the table.

2. Click on the button to specify the name and location where the multi-year met data file will be saved.

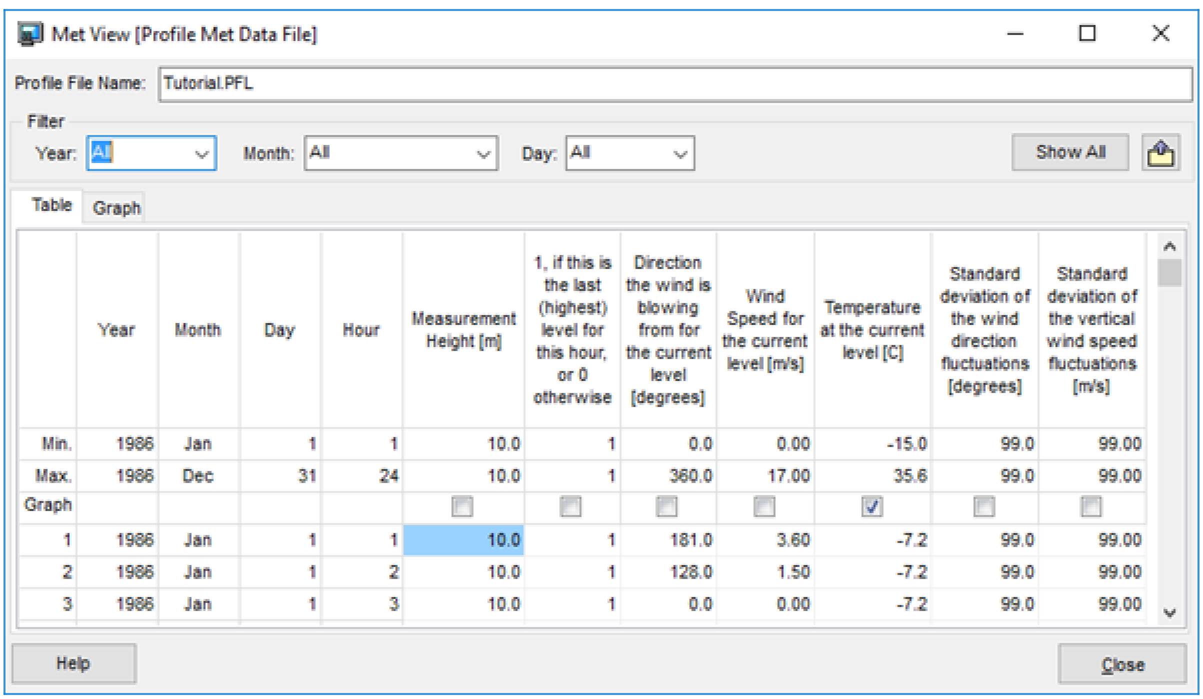
3. Finally, click the Process button. A dialog will be displayed informing you that your Multi-Year Met Data file has been created and provides you with the option of viewing this file in a text editor.

Met View - Table

Met View is a utility that allows you to view the Surface (*.sfc) and Profile (*.pfl) files in a tabular or graphical format. You have access to **Met View** by pressing the **Preview** ( ) to the right of the meteorological file you wish to view. Select **Grid** option to load the **Met View** utility in tabular format.



Met View for Surface Met Data File - Table



Met View for Profile Met Data File - Table

File Header Data (For Surface Met Data File Only)

The **File Header Data** panel displays the name of your output file and the information contained in the header of the file.

Filter

Met View also allows you to refine the information displayed. You can do so using the selection boxes in the **Filter** panel. Available filters include:

- **Year**
- **Month**
- **Day**
- **Julian Day**

To show all the data available, press the **Show All** button.

You can export the data table to either CSV or Excel format by clicking on the **Export** ( ) button.

Data Quality (For Surface Met Data File Only)

This panel shows you the number of calm hours and missing hours along with their respective percentages in the data set.

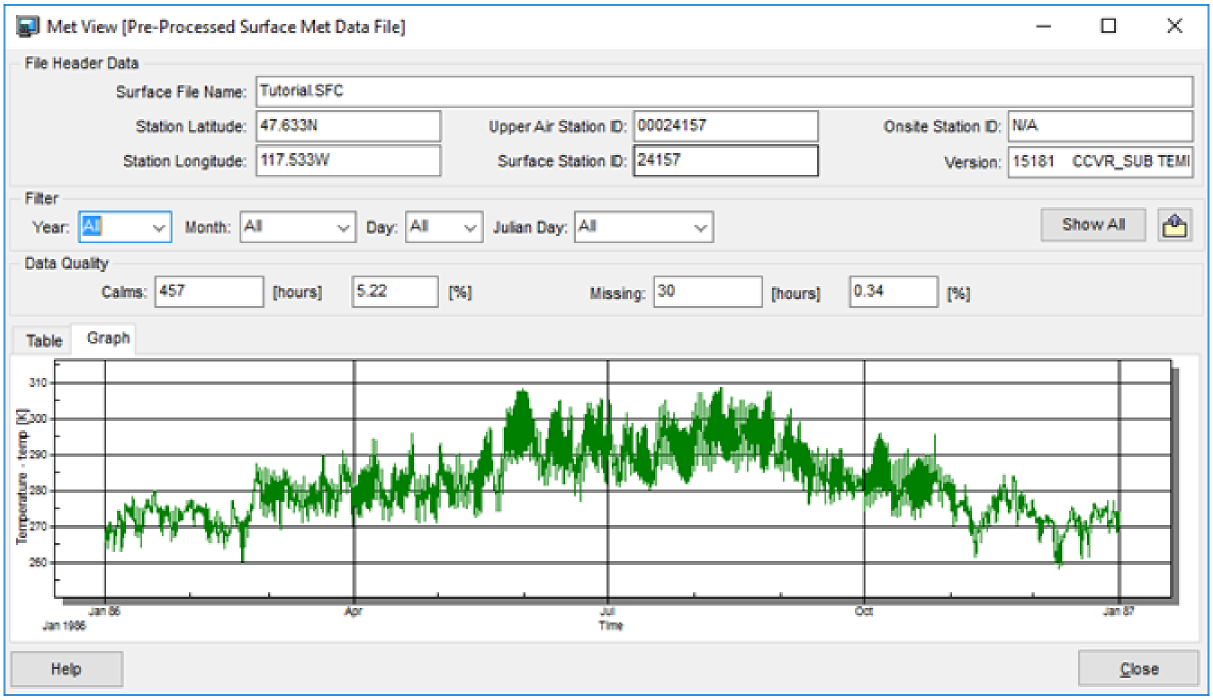
Table

Met View allows you view the data from a meteorological data file in a tabular format.

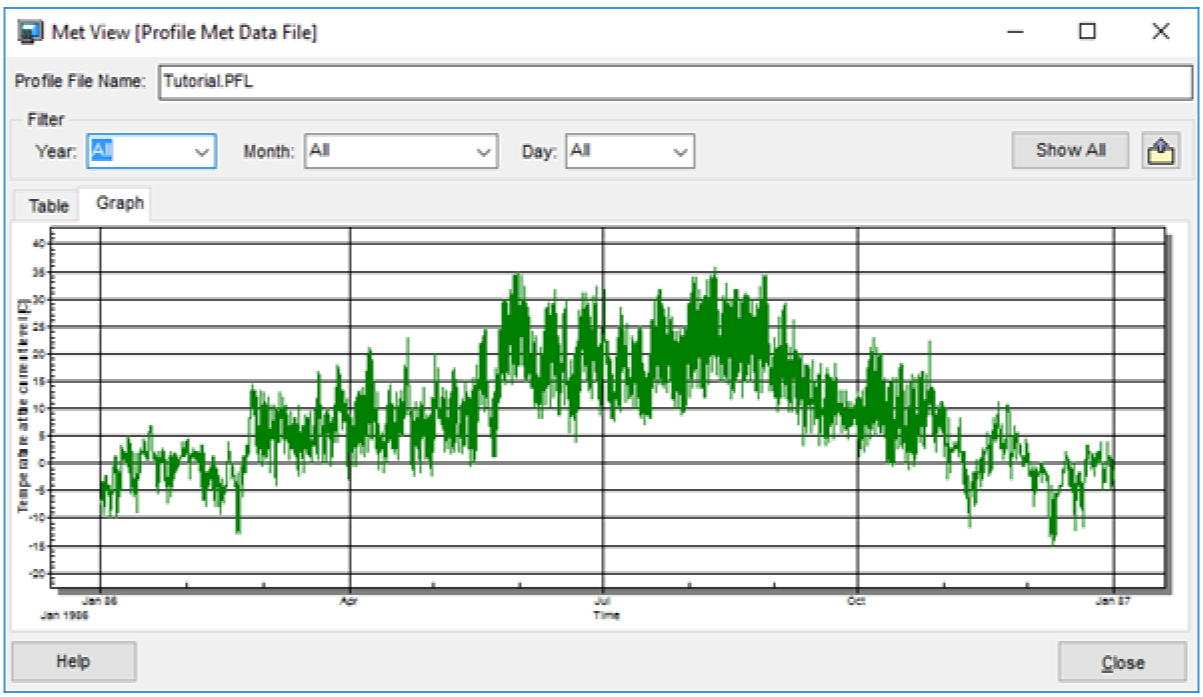
Note the third row down, labeled **Graph**. Checking the box in this row for any of the columns (parameters) will add data for that parameter to the graph. To view this data in graphical format, click the Graph tab.

Met View - Graph

Met View is a utility that allows you to view the Surface (*.sfc) and Profile (*.pfl) files in a graphical or tabular format. You have access to **Met View** by pressing the **Preview** ( ) to the right of the meteorological file you wish to view. Select **Grid** option to load the **Met View** utility in tabular format and then the **Graph** tab.



Met View for Surface Met Data File - Graph



Met View for Profile Met Data File - Graph

File Header Data (For Surface Met Data File Only)

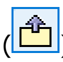
The **File Header Data** panel displays the name of your output file and the information contained in the header of the file.

Filter

Met View also allows you to refine the information displayed. You can do so using the selection boxes in the **Filter** panel. Available filters include:

- **Year**
- **Month**
- **Day**
- **Julian Day**

To show all the data available, press the **Show All** button.

You can export the data table to either CSV or Excel format by clicking on the **Export** () button.

Data Quality (For Surface Met Data File Only)

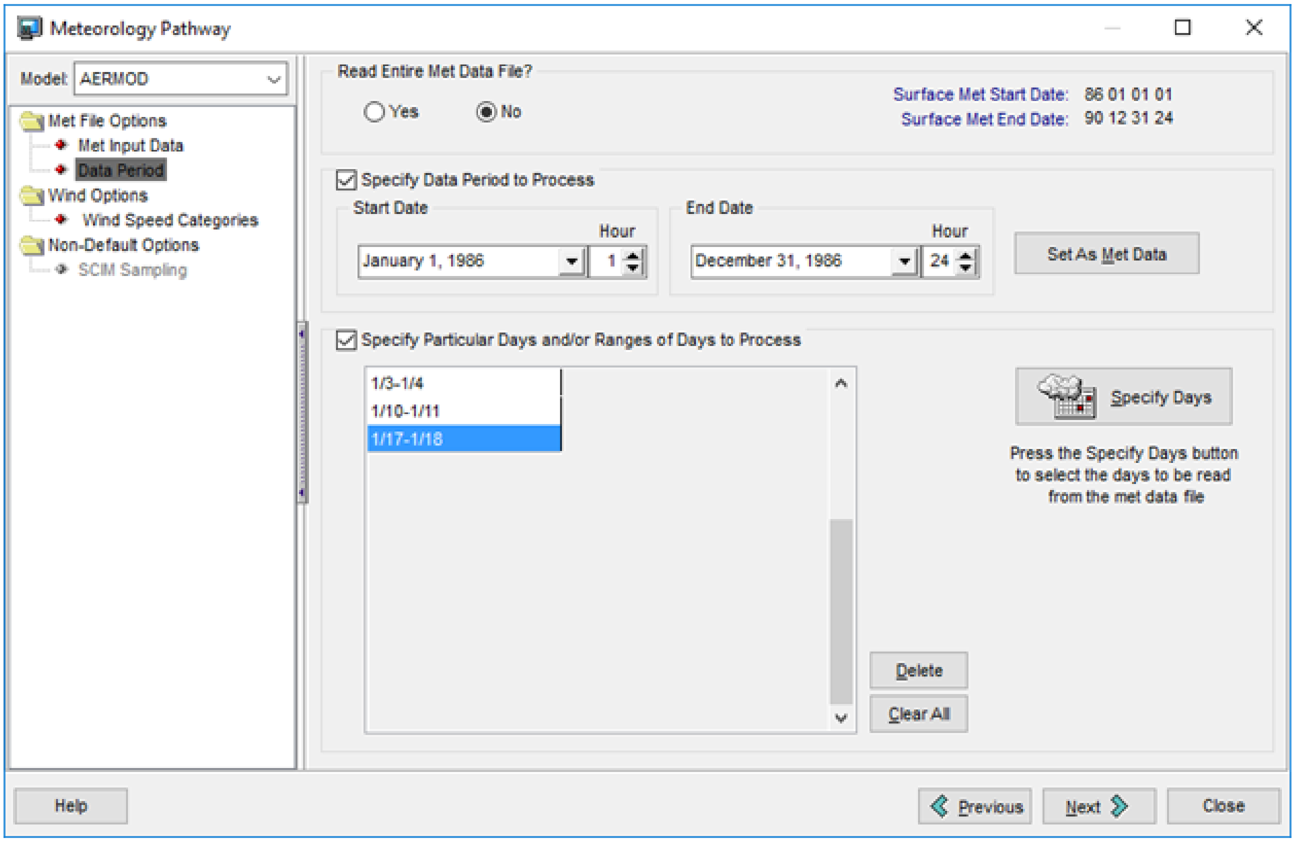
This panel shows you the number of calm hours and missing hours along with their respective percentages in the data set.

Graph

Met View allows you view the data from a meteorological data file in a graphical format. The X-axis represents the timeline of the data file, whereas the Y-axis represents the meteorological parameters which correspond to the selected column(s) from the tabular format.

Data Period

The **Data Period** screen allows you to specify particular days or ranges of days to process from the sequential meteorological file input if you do not want to read the entire met data file. Select **Meteorology Pathway - Data Period** from the tree located on the left side of the Meteorology Pathway dialog, to display the available data period options.



Meteorology Pathway dialog - Data Period

At the top of the dialog you can specify whether or not you want the model to read the entire meteorological data file. If you select **No**, the following options become available for selection -

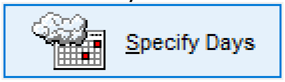
Specify Data Period to Process

Check this box if you wish to specify which period within the meteorological data file the model reads. The following are the inputs necessary if this option is selected:

- **Start Date:** You must specify the year, month, and day of the first record to be read from the meteorological data file.
- **End Date:** You must specify the year, month, and day of the last record to be read from the meteorological data file.
- **Hour:** These are optional parameters that may be used to specify the start and end hours for the data period to be read. If either one is to be specified, then both must be specified. If the start and end hours are not specified, then processing begins with hour 1 of the start date, and ends with hour 24 of the end date.

Any records in the data file that occur before the **Start Date** and after the **End Date** are ignored.

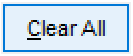
Specify Particular Days and/or Ranges of Days to Process

Check this box if you wish to specify which days or range of days should be read by the model. Click on the 

button to display the Specify Days to Process dialog.

Once you have made your selections the days and/or range of days that you specified are displayed in the list.

Click on this button to delete the selected entry from the list.

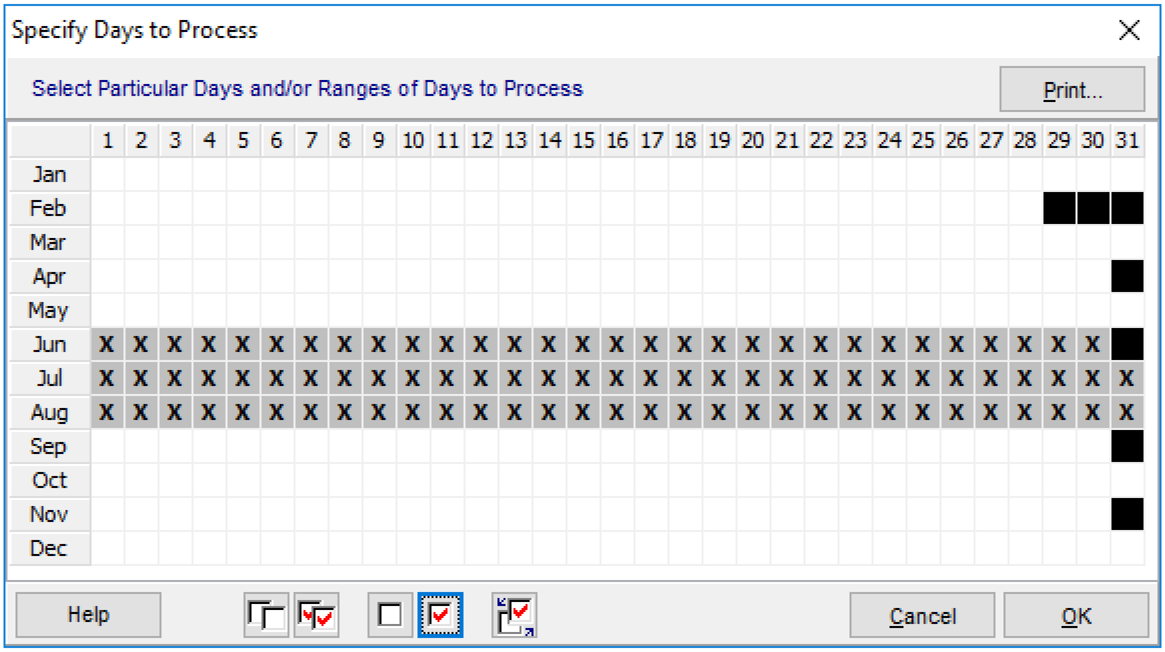


Click on this button to clear the entire list.

Any period averages calculated by the model will apply only to the period of data actually processed. Therefore, if you want to calculate a six-month average you can select period average in the Pollutant/Averaging window, and then specify the six-month period.

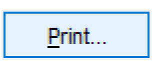
Specify days to process

The **Specify Days to Process** dialog allows you to select the days and/or range of days you would like to process. This can be useful if you wish, for example, to do a seasonal analysis.

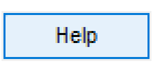


Specify Days to Process dialog

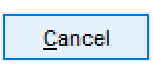
The following buttons are found in the Specify Days to Process dialog:



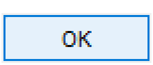
Displays the **Print** dialog, where you can specify printing options such as printer, paper size etc.



Displays the Help contents for the **Specify Days to Process** dialog.



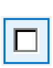
Closes the **Specify Days to Process** dialog, canceling any range of days selected.



Closes the **Specify Days to Process** dialog, applying the range of days selected.

 Unselects all the days.

 Selects all the days.

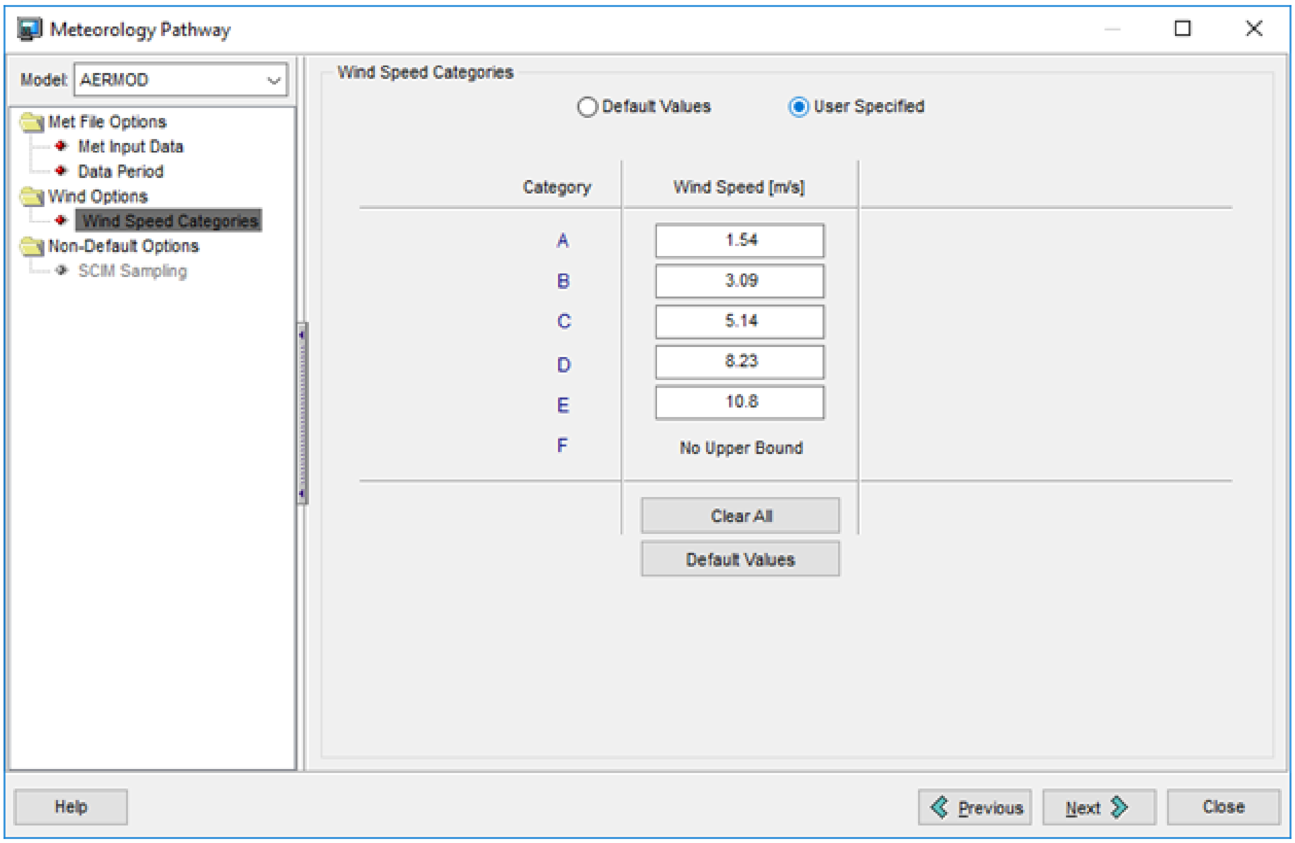
 Unselects the highlighted days.

 Selects the highlighted days.

 Switches the selection of highlighted days.

Wind Speed Categories

The **Wind Speed Categories** screen allows you to specify whether you want the model to use the default wind speed categories or user specified values. Select **Meteorology Pathway - Wind Speed Categories** from the tree located on the left side of the Meteorology Pathway dialog, to display the available options.



Meteorology Pathway dialog - Wind Speed Categories

The US EPA models use six wind speed categories. These are defined by the upper bound wind speed which must be defined for the first five categories. The sixth category is assumed to have no upper bound. If you select the **Default Values** option, AERMOD View automatically inputs the default values for wind speed categories in the panels and does not allow you to change these values. The default values used by the models are as follows:

Category	Wind Speed (m/s)
A	1.54
B	3.09
C	5.14
D	8.23
E	10.8
F	No Upper Bound

If you chose the **User Specified** option, you can specify wind speed values other then the default values in the text boxes for each category. These values are defined by the upper bound wind speed for the first five categories, the sixth category is assumed to have no upper bound.

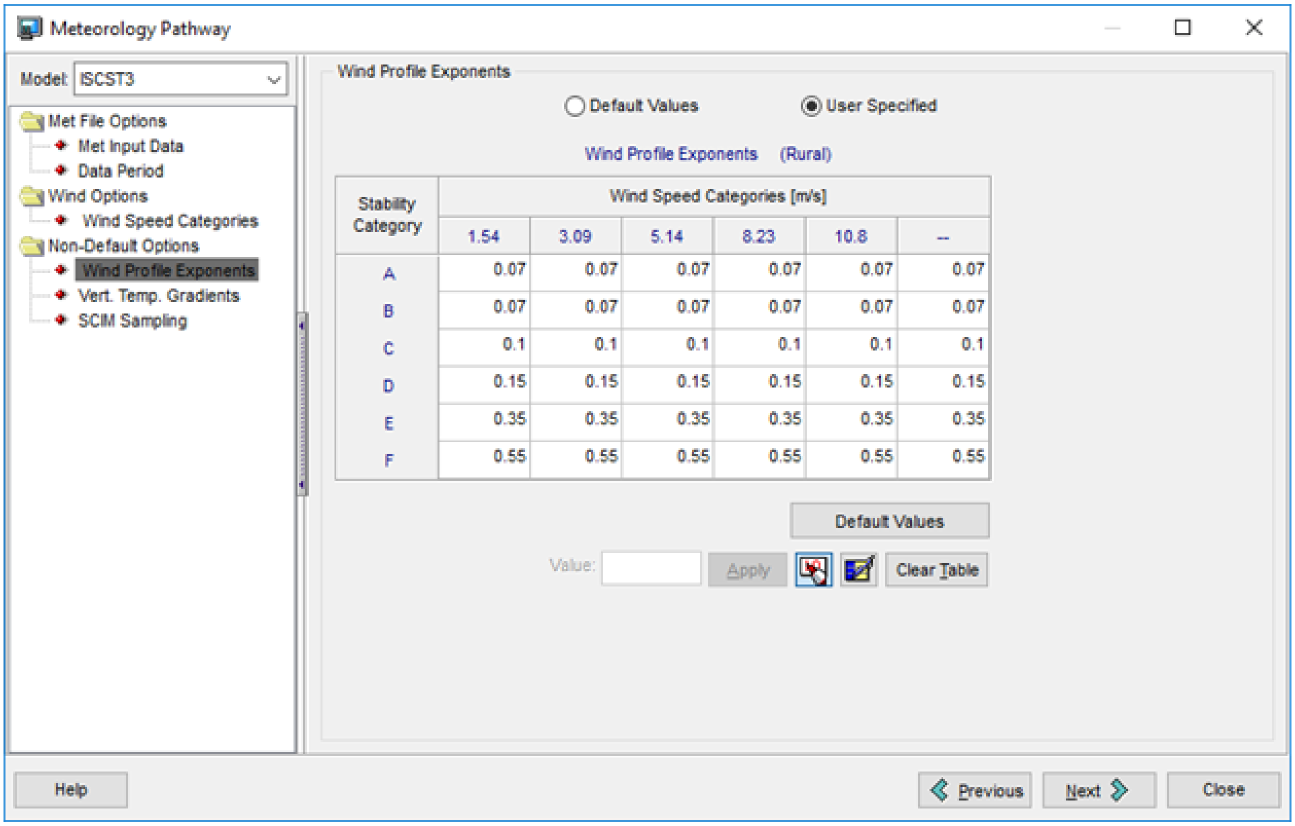
Clear All — Click on this button to clear all the values in the text boxes.

Default Values — Click on this button to place the default wind speed values in the fields.

Wind Profile Exponents

ISCST3 & ISC-PRIME Only

The Wind Profile Exponents screen provides you the option of specifying wind profile exponent values other than the default. Select Meteorology Pathway - Wind Profile Exponents from the tree located on the left side of the Meteorology Pathway dialog, to display the available options.



Meteorology Pathway dialog - Wind Profile Exponents

Wind Profile Exponents are the values of the exponent used to specify the profile of wind speed with height according to the power law. The ISCST3 and ISC-PRIME models use default wind profile exponents if the Regulatory Default option is selected in the Control Pathway. If the Non-Default Options is selected, you then have the choice of specifying wind profile exponent values other than the default.

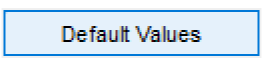
If you select the **Default Values** option, AERMOD View automatically inputs the default values for the wind profile exponents and does not allow you to change these values. The default values used by the models for rural and urban dispersion coefficients are as follows:

Pasquil Stability Category	Wind Profile Exponents	
	Rural	Urban
A	0.07	0.15
B	0.07	0.15

Pasquil Stability Category	Wind Profile Exponents	
	Rural	Urban
C	0.10	0.20
D	0.15	0.25
E	0.35	0.30
F	0.55	0.30

If you chose the **User Specified** option, you can specify wind profile exponents other then the default values in the table.

> You can only specify wind profile exponents if the Non-Default Options was selected in the Control Pathway.



Click on this button to place the default wind profile exponents in the table.

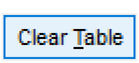
You can assign the values in the table using the following controls:

Value: Type in the value you wish to assign to selected cells.

- Click to apply the contents of the **Value** field to the selected cells.


- Click this button to be able to manually edit values in individual cells.


- Click this button to be able to highlight/select entire rows of cells. Clicking on an already selected row will clear that selection.

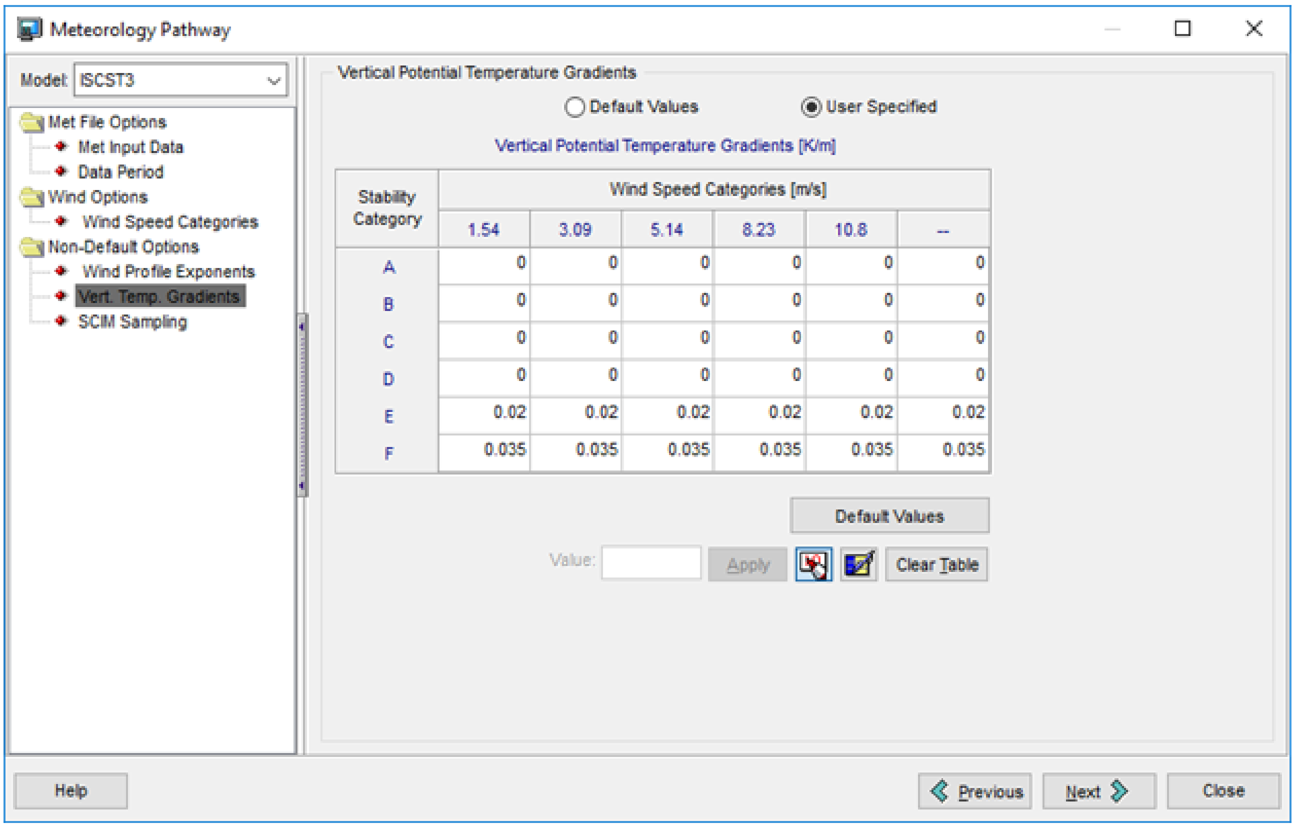


- Click to clear all the values in the table.

Vertical Temperature Gradients

ISCST3 & ISC-PRIME Only

In the **Vertical Temperature Gradients** screen you have the option of specifying vertical temperature gradients values other than the default. Select **Meteorology Pathway - Vertical Temperature Gradients** from the tree located on the left side of the Meteorology Pathway dialog, to display the available options.



Meteorology Pathway - Vertical Temperature Gradients

Vertical potential temperature gradient is the change of potential temperature with height, used in modeling the plume rise through a stable layer, and indicates the strength of the stable temperature inversion. A positive value means that potential temperature increases with height above ground and indicates a stable atmosphere.

The ISCST3 and ISC-PRIME models use default vertical temperature gradients if the Regulatory Default option is selected in the Control Pathway. If the Non-Default Options is selected, you then have the choice of specifying vertical temperature gradients values other than the default.

If you select the **Default Values** option, AERMOD View automatically inputs the default values for the vertical potential temperature gradients and does not allow you to change these values. The default values used by the models for rural and urban dispersion coefficients are as follows:

Pasquill Stability Category	Wind Profile Exponents	
	Rural (K/m)	Urban (K/m)
A	0.0	0.0
B	0.0	0.0
C	0.0	0.0
D	0.0	0.0
E	0.02	0.02
F	0.035	0.035

If you chose the **User Specified** option, you can specify vertical potential temperature gradients other then the default values in the table.

> You can only specify vertical temperature gradients if the Non-Default Options was selected in the Control Pathway.

Default Values Click on this button to place the default vertical potential temperature gradients in the table.

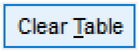
You can assign the values in the table using the following controls:

Value: Type in the value you wish to assign to selected cells.

Apply - Click to apply the contents of the **Value** field to the selected cells.

 - Click this button to be able to manually edit values in individual cells.

 - Click this button to be able to highlight/select entire rows of cells. Clicking on an already selected row will clear that selection.



- Click to clear all the values in the table.

SCIM Sampling

ISCST3 & AERMOD Only

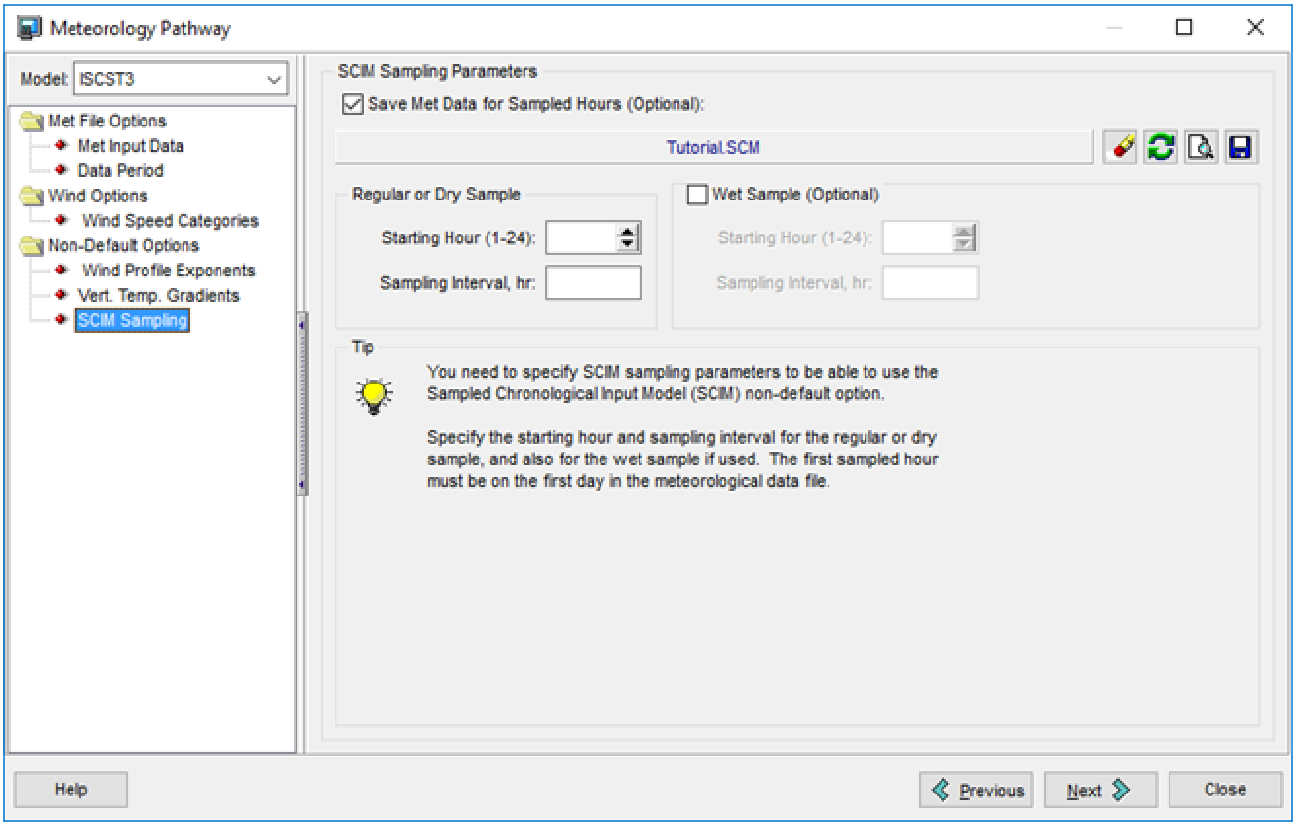
The **SCIM Sampling** screen provides you with the option to reduce model runtime and is primarily applicable to multi-year model simulations. The SCIM (Sampled Chronological Input Model) option samples the meteorological data at a user-specified regular interval to approximate the long-term (Annual) average impacts.

Select **Meteorology Pathway - SCIM Sampling** from the tree located on the left side of the Meteorology Pathway dialog, to display the available options.

> The SCIM Sampling option is available only if you have selected the Non-Default, TOXICS, SCIM option in the Control Pathway and can only be used with the Annual averaging period.

The SCIM Sampling parameters to be specified differ depending on the model, ISCST3 or AERMOD. See below the required parameters -

ISCST3



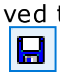
Meteorology Pathway - SCIM Sampling (ISCST3)

For the ISCST3 model, the SCIM option has the following restrictions:

1. Can only be used with the Annual averaging option.

2. The Total Deposition (Dry and Wet) option is ignored. The user is advised to calculate dry and wet deposition rates separately using the Dry Deposition and Wet Deposition options and to add the two to obtain the total deposition rate when the SCIM option is used.

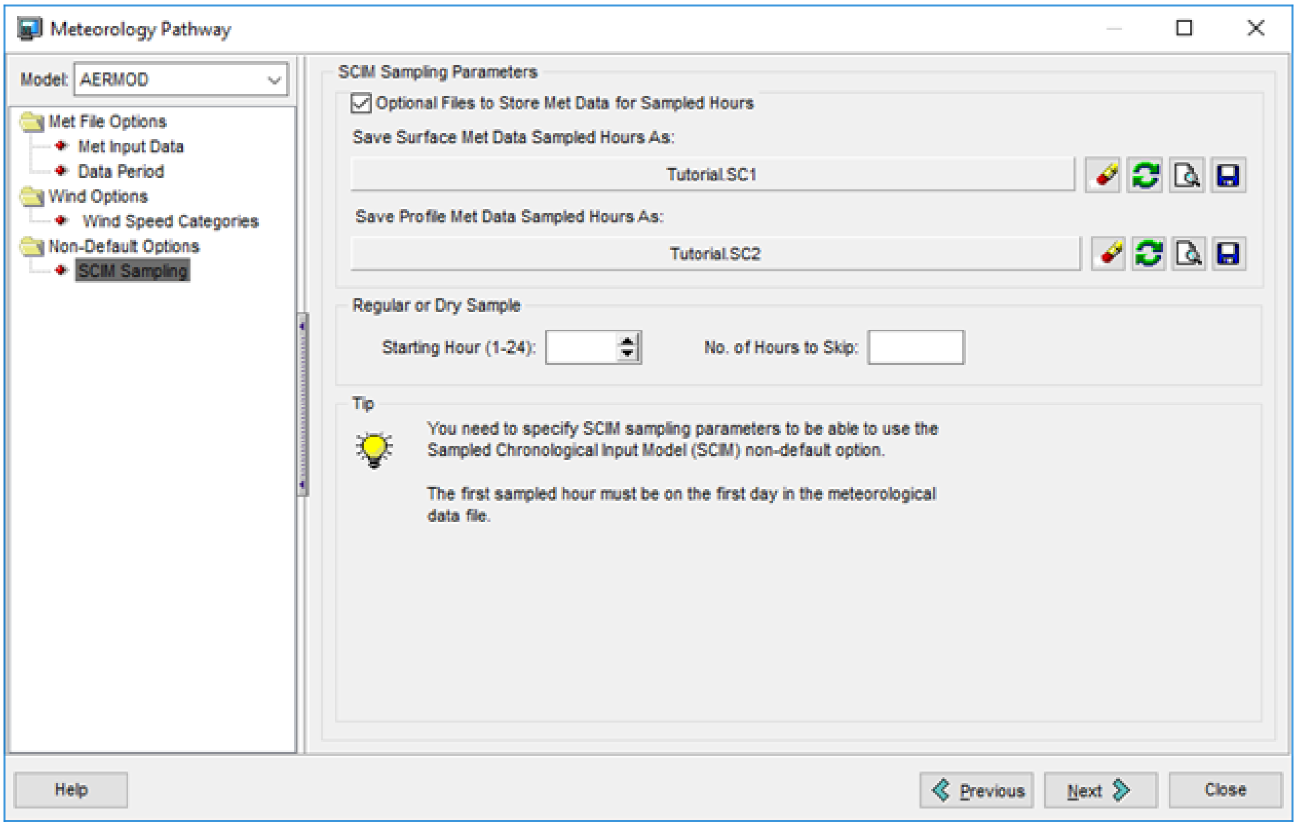
The following parameters should be defined -

- **Save Met Data for Sampled Hours:** By default this file will be have the same name as your project (projectname.scm) and will be saved to your project folder. If you wish to specify the name and location of this file click on the  button. This output file will contain the meteorological data for the sampled hours.

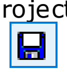
- **Regular or Dry Sample:** You must specify the starting hour and the number of hours to skip (sampling interval) for the regular or dry sample. The starting hour can range from 1 to 24 and must be on the first day in the meteorological data file. The number of hours to skip must be greater than 1.

- **Wet Sample:** Check this option if you want to specify the starting hour and the number of hours to skip (sampling interval) for the wet sample. The only restriction for the wet sample option is that the starting hour cannot be greater than the number of hours to skip. Since wet deposition does not occur at regular intervals, you can also specify the wet sampling interval to reduce uncertainty introduced by sampling for wet deposition.

AERMOD

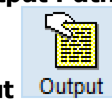


Meteorology Pathway - SCIM Sampling (AERMOD)

- **Optional Files to Store the Met Data for the Sampled Hours:** By default these files will be have the same name as your project (projectname.sc1 and projectname.sc2) and will be saved to your project folder. If you wish to specify a different name and location for these files click on the  button. These output files will contain the surface and profile meteorological data for the sampled hours.

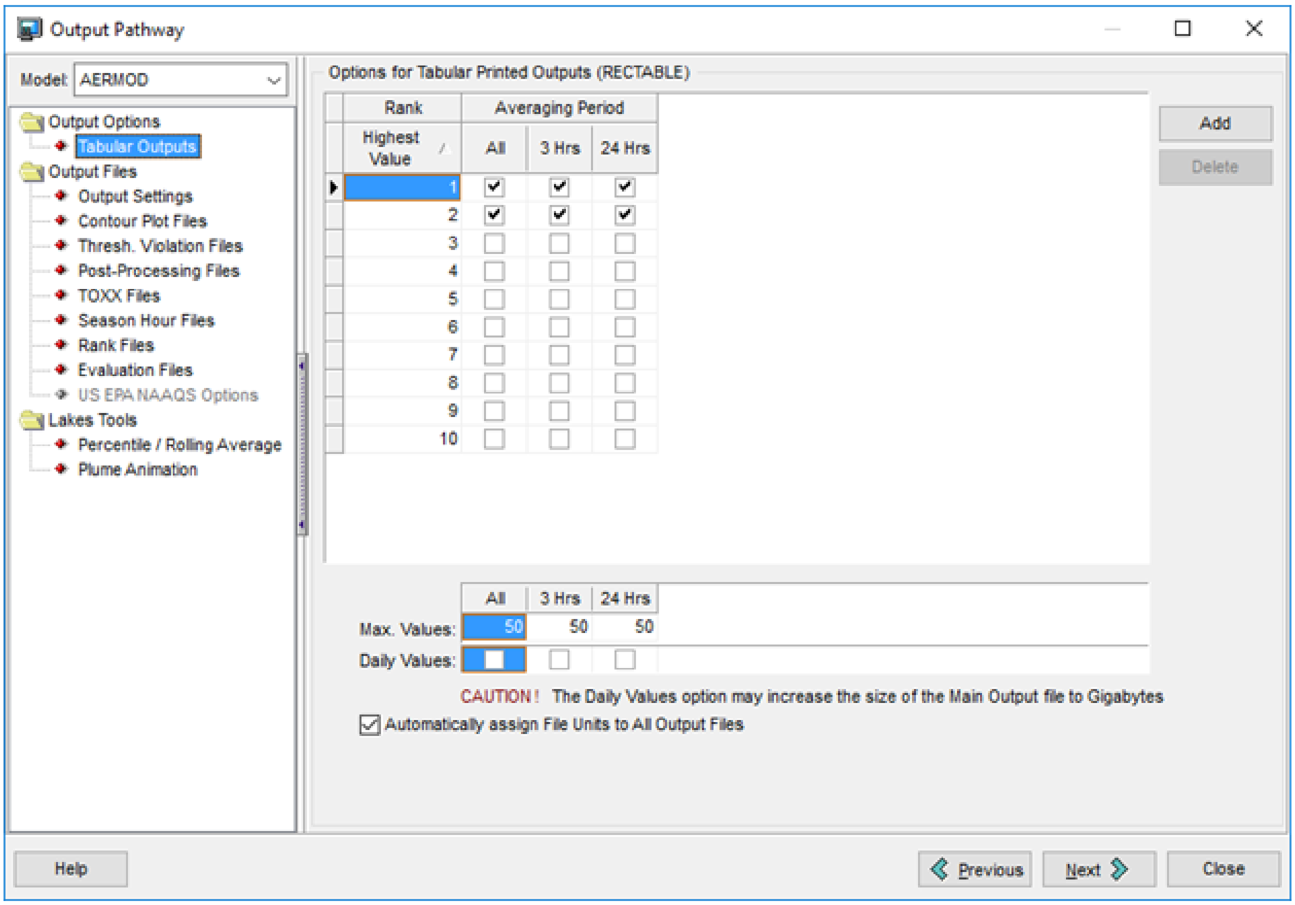
- **Regular or Dry Sample:** You must specify the starting hour and the number of hours to skip (sampling interval) for the regular or dry sample. The starting hour can range from 1 to 24 and must be on the first day in the meteorological data file. The number of hours to skip must be greater than 1.

Output Pathway

The **Output Pathway** allows you to specify the output options for a particular run such as contour plot files and threshold violation files. You have access to the **Output Pathway** dialog by selecting **Data | Output Pathway...** from the menu or by clicking on the **Output** 

menu toolbar button.

The **Output Pathway** dialog uses a two-pane view. The tree located on the left side of the dialog is used for navigation and item selection. Select an item (marked as ◆) in the tree to display the available options on the right panel.



Output Pathway dialog

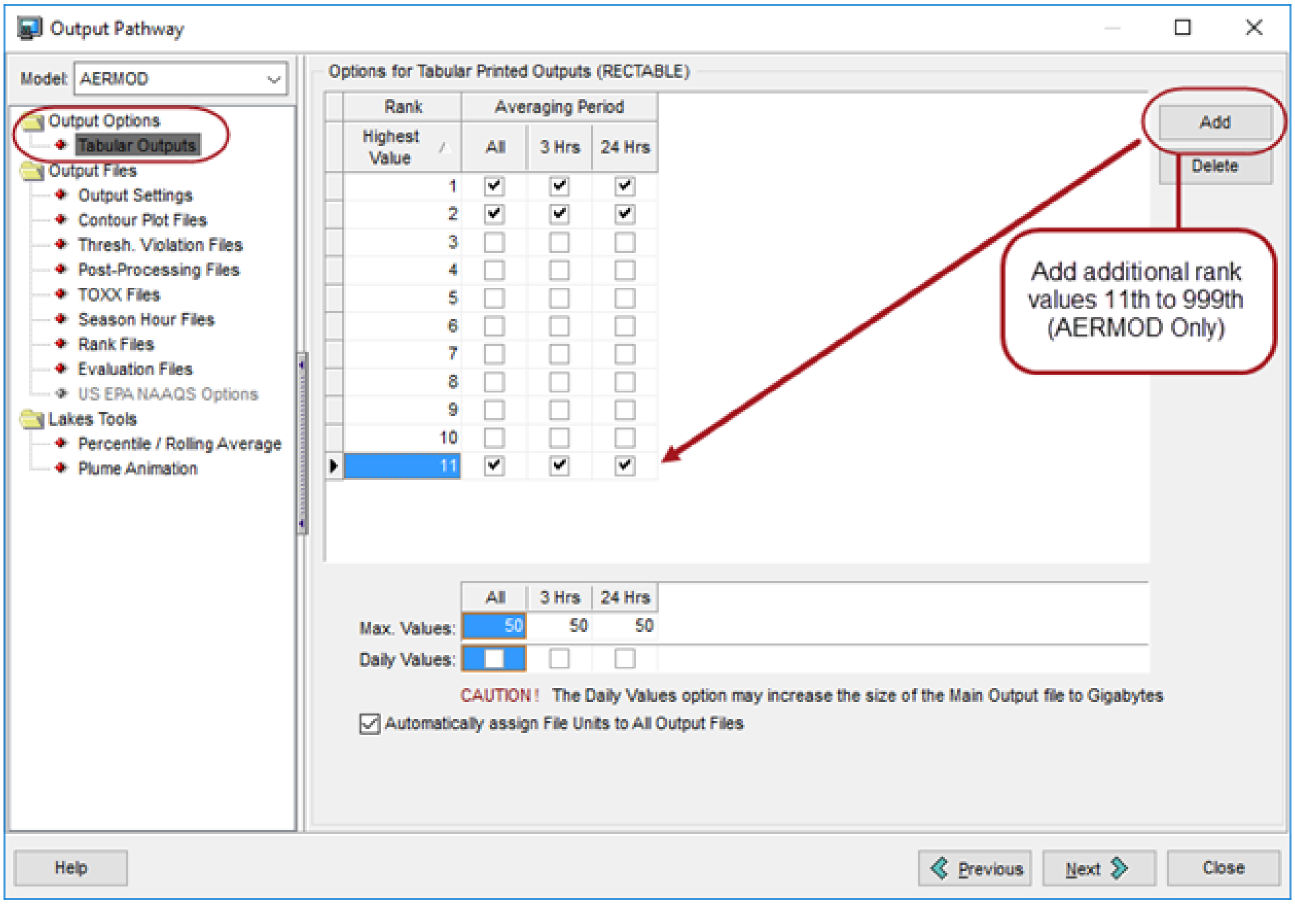
In the **Output Pathway** you can define the following:

- Tabular Outputs (RECTABLE - MAXTABLE - DAYTABLE)
- Output Settings (SUMMFILE)

- Contour Plot Files (PLOTFILE)
- Threshold Violation Files (MAXIFILE)
- Post-Processing Files (POSTFILE)
- TOXX Files (TOXXFILE)
- Season Hour Files (SEASONHR)
- Rank Files (RANKFILE)
- Evaluation Files (EVALFILE)
- Percentile / Rolling Average
- Plume Animation

Tabular Outputs

In the **Tabular Outputs** screen you can define tabular output options for each short-term averaging period selected in the Pollutant/Averaging screen in the Control Pathway. The tabular output options do not apply to the Period or Annual averaging time. Select **Output Pathway - Tabular Outputs** from the tree located on the left side of the Output Pathway dialog, to display the available tabular output options.



Output Pathway dialog - Tabular Outputs

> If no short-term averaging period was selected in the Pollutant/Averaging screen in the Control Pathway, then the tabular output options are not available for use. AERMOD View displays a message advising you that no short-term averaging periods were selected for the current run.

There are three tabular output options available for selection:

Highest Values (RECTABLE)

This option produces tables of high values summarized by receptor for each short-term averaging period. For each short-term averaging period, you should check the high values that are applicable to your modeling.

The following are the limits for each model:

- **AERMOD:** Up to the 999th highest value
- **ISCST3:** Up to the 10th highest value
- **ISC-PRIME:** Up to the 6th highest value

Maximum Values (MAXTABLE)

This option defines the number of overall highest values that will be summarized in the output file for each short-term averaging period being modeled. A separate maximum overall value table is produced for each source group.

If, for example, you specify 50 for the maximum values option then the model will produce a table for each short-term averaging period and each source group containing the 50 highest values.

In the output file, the maximum values table is displayed with the following format:

```
*** THE MAXIMUM   50   1-HR AVERAGE CONCENTRATION **
    INCLUDING SOURCE(S):    STCK1    , STCK2    ,

        ** CONC OF SO2       IN MICROGRAMS/M**3

RANK        CONC    (YYMMDDHH) AT      RECEPTOR (XR,YR) OF TYPE
- - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
   1.     101.44812 (88052123) AT ( 439900.00, 5299200.00)  GC
   2.      98.40558 (88072103) AT ( 439800.00, 5299200.00)  GC
   3.      97.77658 (88091003) AT ( 439800.00, 5299100.00)  GC
   4.      97.21135 (88091003) AT ( 439900.00, 5299200.00)  GC
   5.      93.91324 (88052124) AT ( 439700.00, 5299100.00)  GC
```

The above example shows that the first ranked concentration value out of the 50 has a concentration of 101.44812 micrograms/m**3 and occurs on May 21, 1988 at hour 23 for the specified receptor location (439900.00, 5299200.00).

For cases where you want the same maximum value to be applied to all the short-term averaging periods being modeled, the input may be simplified by entering the maximum value for the ALL averaging period option.

> The number of overall maximum values that the model can store is unlimited for ISCST3 and AERMOD but is set at 50 for ISC-PRIME.

Daily Values (DAYTABLE)

This option controls the output options for tables of concurrent values summarized by receptor for each day processed. For each averaging period for which the daily values option is selected, the model will print in the main output file the concurrent averages for all receptors for each day of data processed. Results are output for each source group.

For example, if 2 and 4-hour averages are calculated and the daily values option is selected for both averages, then for the first day of data processed (Day 1), there will be:

- 12 sets of tables for the 2-hr average (one for each 2-hour period in the day)
- 6 sets of tables for the 4-hr average (one for each 4-hour period in the day)

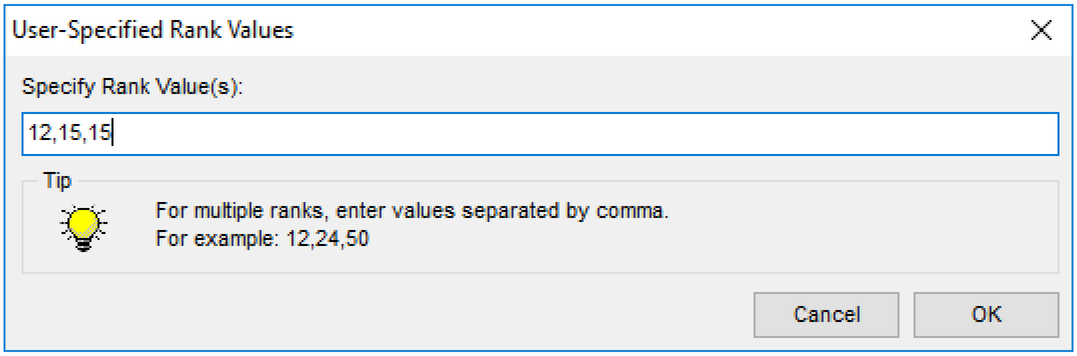
For cases where you want to define daily tables for all the short-term averaging periods being modeled, the input may be simplified by only checking the **Daily Values** check box for the **ALL** averaging period option.

> **Warning:** The **Daily Values** option can produce very large output files, especially when used with a full year of data and very short period averages, such as 1 hour and 3 hour. You should use this option with caution.

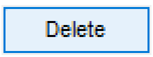
How to Add Additional Ranks (AERMOD Only):

Starting with the release of the US EPA AERMOD Version 11059, user-specified ranks up the 999th highest value can be specified to support significant contribution analyses.

1. Press the **Add** button to display the **User-Specified Rank Values** dialog.
2. You can specify multiple ranks by entering the values separated by comma (e.g., 12,20,50).
3. Press the **OK**. The additional ranks will displayed in the **Tabular Output** table and available for selection



Press this button to add additional ranks to the Tabular Output table (from 11th to 999th)



Press this button to delete any ranks you have added (from 11th and 999th)

In the model main output file, the highest values table is displayed with the following format:

```
                                   *** THE SUMMARY OF HIGHEST  3-HR RESULTS ***

                          ** CONC OF SO2      IN MICROGRAMS/M**3                        **

                                                     DATE                                                      NETWORK
GROUP ID                          AVERAGE CONC    (YYMMDDHH)          RECEPTOR  (XR, YR, ZELEV, ZFLAG)    OF TYPE  GRID-ID
- - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -

ALL      HIGH  1ST HIGH VALUE IS    1078.51770  ON 88110424: AT (  439100.00,  5298300.00,        0.00,      0.00)  GC   UCART1
```

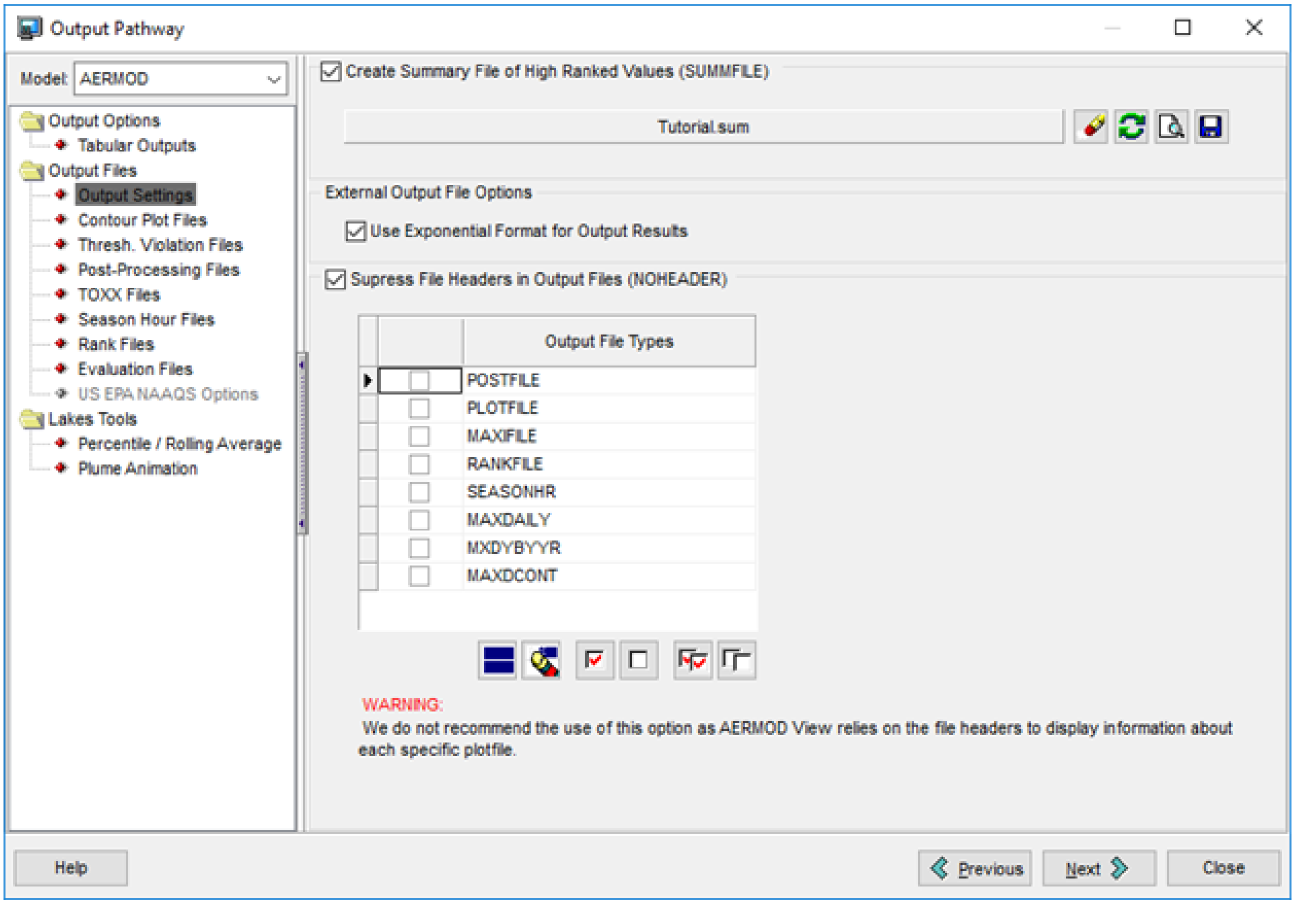
The above example shows that the first highest concentration value at the specified receptor location (439100.00, 5298300.00) is 1078.51770 micrograms/m**3 and occurs on November 4, 1988 at hour 24.

For cases where you want the same high values to be applied to all the short-term averaging periods being modeled, the input may be simplified by checking which high values are to be summarized by receptor for the ALL averaging period option.

Output Settings

AERMOD Only

The **Output Settings** screen allows you to select a few additional output options. Select **Output Files - Output Settings** on the left side of the Output Pathway dialog to access these settings.



Output Pathway - Output Settings screen

The following options are available in the Output Settings screen:

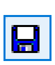
Create Summary File of High Ranked Values (SUMFILE)

Check this box if you wish to output the summary of high ranked values to a separate file. By default the summary files will be saved to the project location (ProjectName.sum).

 Click this button to remove the currently selected summary file name

 Click this button to return to the default file name and location

 Click this button to open a preview of the summary file in WordPad

 Click this button to save the summary file at user specified location and/or under a new file name.

External Output File Options

- **Use Exponential Format for Output Results:** Check this box if you want to specify the use of exponential notation, rather than fixed format, for results that are output to separate result files (keyword: FILEFORM EXP).

> The Exponential Format option may also be useful for applications with relatively large impacts that may overflow the Fortran format specifier of F13.5 used by the AERMOD model for fixed-formatted outputs.

Suppress File Headers in Output Files (NOHEADER)

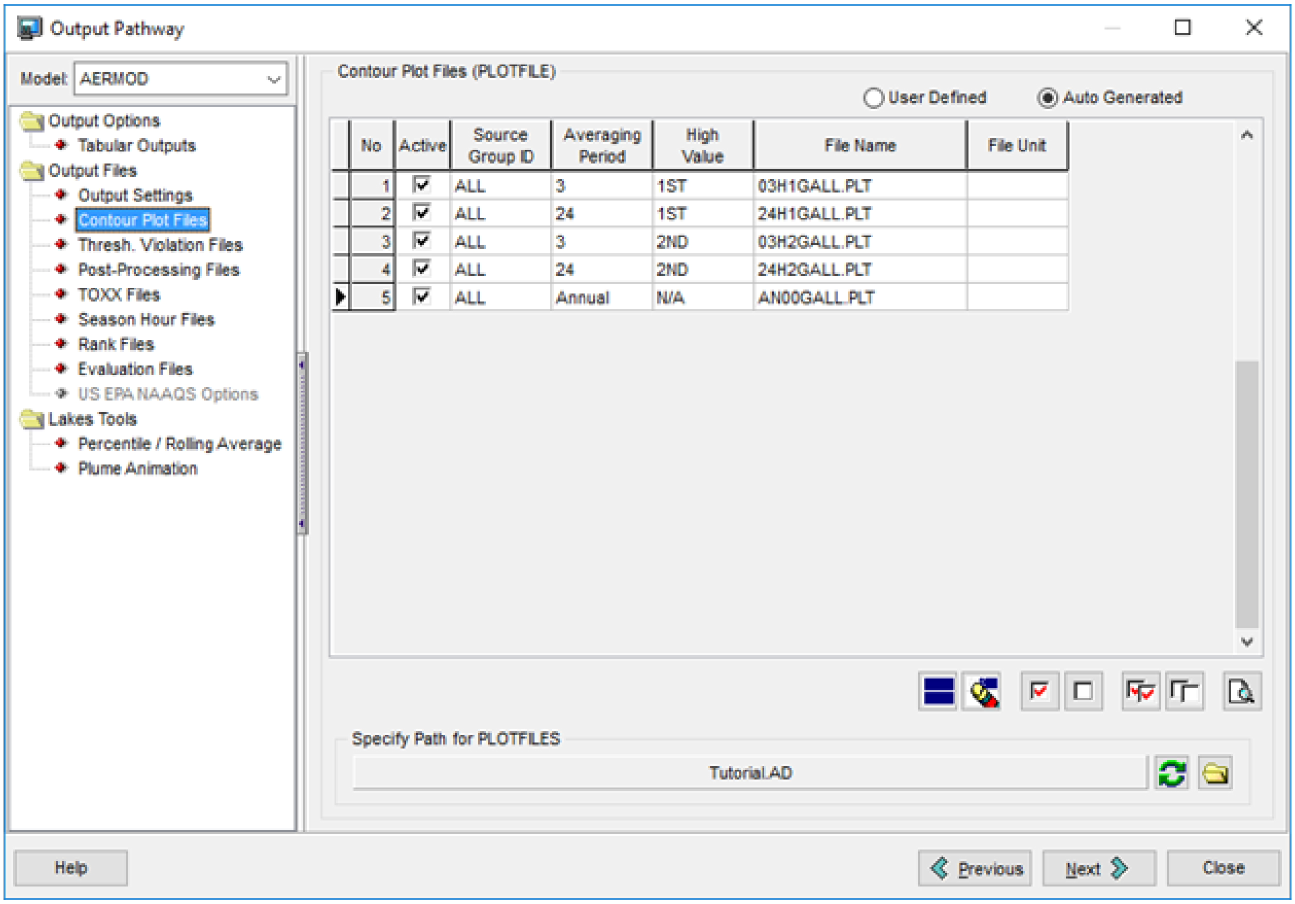
Check this box if you do not want the headers to appear in one or more formatted output files. You can request to suppress file headers from the following output types:

- POSTFILE
- PLOTFILE
- MAXIFILE
- RANKFILE
- SEASONHR
- MAXDAILY
- MXDYBYYR
- MAXDCONT

> **Warning:** It is not recommend that headers be suppressed from PLOTFILEs, MAXIFILEs, and MAXDCONT files as AERMOD View relies on file headers to display further information for contours.

Contour Plot Files

The **Contour Plot Files** screen allows you to request the model to produce files that are suitable for generating contour plots of concentration and deposition results. Select **Output Files - Contour Plot Files** in the left side of the Output Pathway dialog to display these options.



Output Pathway - Contour Plot Files screen

There are two alternatives to setup contour plot files, **Auto Generated** and **User Defined**:

Auto Generated

Contour plot files can be automatically defined by AERMOD View with the **Auto Generated** option. By selecting this option, AERMOD View will automatically setup all possible combinations for generating plotfiles for each run. The following fields are displayed for each plotfile -

- **Active:** This field controls if the plotfile should be created or not. By default, all boxes will be checked for plotfile generation, if you do not want to include one or more of these plotfiles click on the Active check box to uncheck.

- **Source Group ID**: This field displays the Source Group ID for which the plotfile is being created.
- **Averaging Period:** This field displays the Averaging Period for which the plotfile is being created.
- **High Value:** This field displays the High Value (1st, 2nd, 3rd, etc.) for which the plotfile is being created.
- **File Name:** This field displays the file name for the plotfile to be created
- **File Unit:** This is the optional file unit and it is blank by default. If this parameter is omitted, then the model will dynamically allocate a unique file unit for this file.

> The convention for the default name for the plot files is AVGHVSG.plt, where:
>
> AVG = averaging period
>
> HV = high value
>
> SG = source group
>
> Example: 03H1GALL.PLT
>
> 03 = 3-hour averaging period
>
> H1 = 1st High Value
>
> GALL = Source Group ALL

- **Specify Path for PLOTFILES:** By default, all plot files will be saved to default locations. If you wish to specify an alternate location for the plotfiles, click on the  button and select the location to save the files. If you wish to return to the default location, click on the  button.

> The default locations where the plotfiles are saved are determined according to the model that is run:
>
> **ISCST3 Runs**: project directory\ProjectName.IS\
>
> **ISC-PRIME Runs**: project directory\ProjectName.PR\
>
> **AERMOD Runs**: project directory\ProjectName.AD\

There are a number of buttons available in the **Auto-Generated Contour Plot Files** panel. Their functions are described below:

 Click on this button to select all plotfiles in the list.

 Click on this button to unselect any selected plotfiles in the list.

 Click on this button to mark the selected plotfiles as Active. Only active plotfiles will be generated.

 Click on this button to mark the selected plotfiles as inactive.

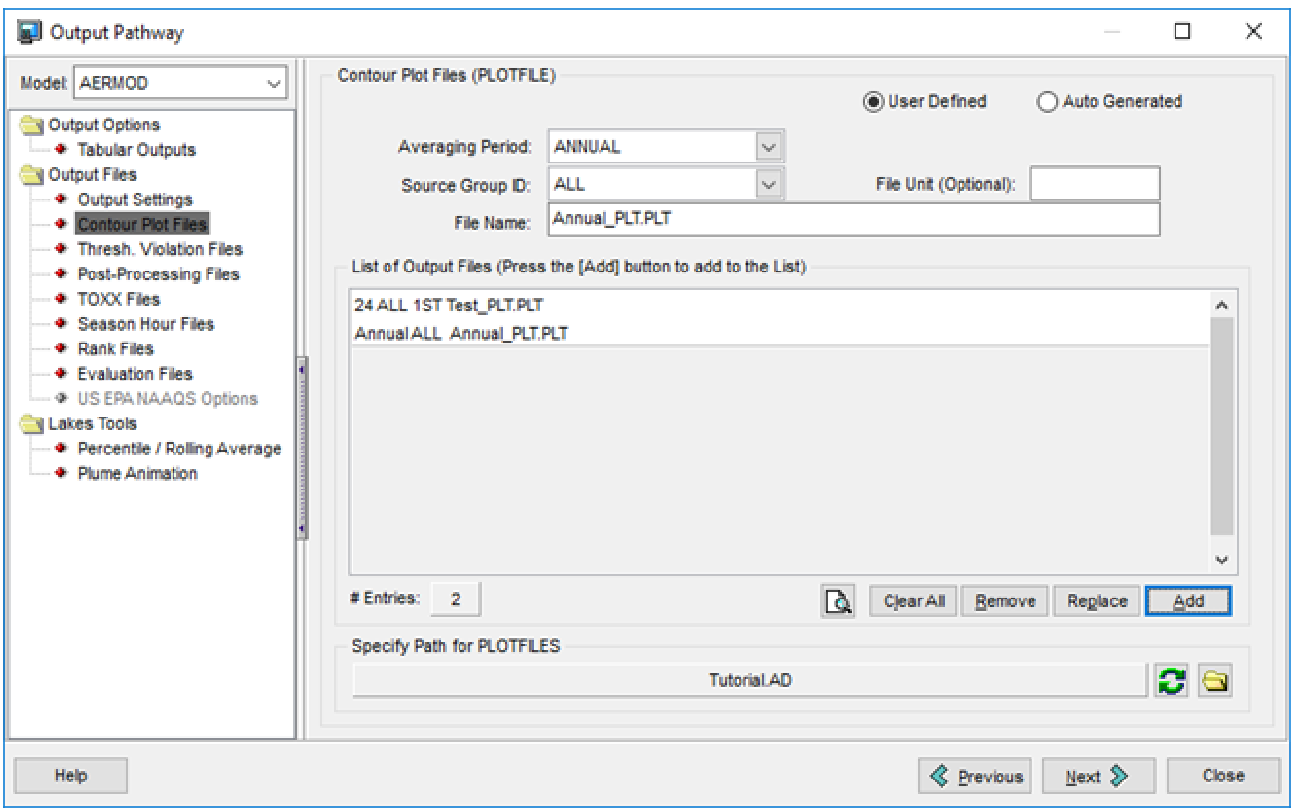
 Click on this button to mark all plotfiles in the list as Active. Only active plotfiles will be generated.

 Click on this button to mark the selected plotfiles as inactive.

 Select this option to preview the contents of the selected plotfile. Please note that you can preview the contour plot file only after running the model.

User Defined

The **User Defined** option should be used if you prefer to define your own plotfiles.



Output Pathway dialog - Contour Plot Files - User Defined

How to Create User Defined Contour Plot Files:

1. Select from the **Averaging Period** drop-down list the averaging period for the plot file. This drop-down list will contain all the averaging periods selected for your project in the Pollutant/ Averaging screen in the Control Pathway.

2. Select from the **Source Group ID** drop-down list the source group ID for the plot file. This drop-down list will contain a list of all Source Group IDs created in the Source Groups window.

3. Enter a file name for the contour plot file in the **File Name** panel.

At the present time, the original U.S. EPA ISCST3 model can only accept file names up to eight characters long, with a three character extension to successfully create the file. The AERMOD and ISC-PRIME models have no such limitations.

4. Select from the **High Value** drop-down list the short-term high value for the plot file. This drop-down list will contain all the high values specified for each averaging period from the Tabular Outputs window. Note that the **High Value** parameter is not applicable for **PERIOD** or **ANNUAL** averages, since there is only one period or annual average for each receptor.

5. Click on the 

button to add the specified plot file to the **List of Output Files** window. You can repeat Steps 1-5 as many times as necessary to created all the desired plot files. The list is used to store all contour plot files specified. Only those combinations contained in the List of Output Files will be considered when running the model.

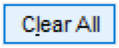
If more than one Output Type is selected in the Control Pathway, then the contour plot file will include all of the output types selected in the following order: Concentration, Total Deposition, Dry Deposition, and/or Wet Deposition. The results for each output type will be printed in separate columns, one record per receptor, in the order given above.

6. By default, all plotfiles will be saved to preset locations. If you wish to specify an alternate location for the plotfiles, click on the  button in the Specify Path for PLOTFILES panel and select the location to save the files. If you wish to return to the default location, click on the  button.

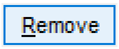
> The default locations where the plotfiles are saved are determined according to the model that is run:
>
> **ISCST3 Runs**: project directory\ProjectName.IS\
>
> **ISC-PRIME Runs**: project directory\ProjectName.PR\
>
> **AERMOD Runs**: project directory\ProjectName.AD\

There is a number of buttons available in the **User Defined Contour Plot Files** panel. Their functions are described below:

 Press this button to view the contents of the selected file. Please note that you can preview the contour plot file only after running the model.



Click on this button to clear all the files from the List of Output Files window.



Click on this button to remove the selected file from the List of Output Files window.

Click on this button to replace the selected file in the List of Output Files with the alternate parameters specified.

 Click on this button to add the specified file parameters to the List of Output Files window.

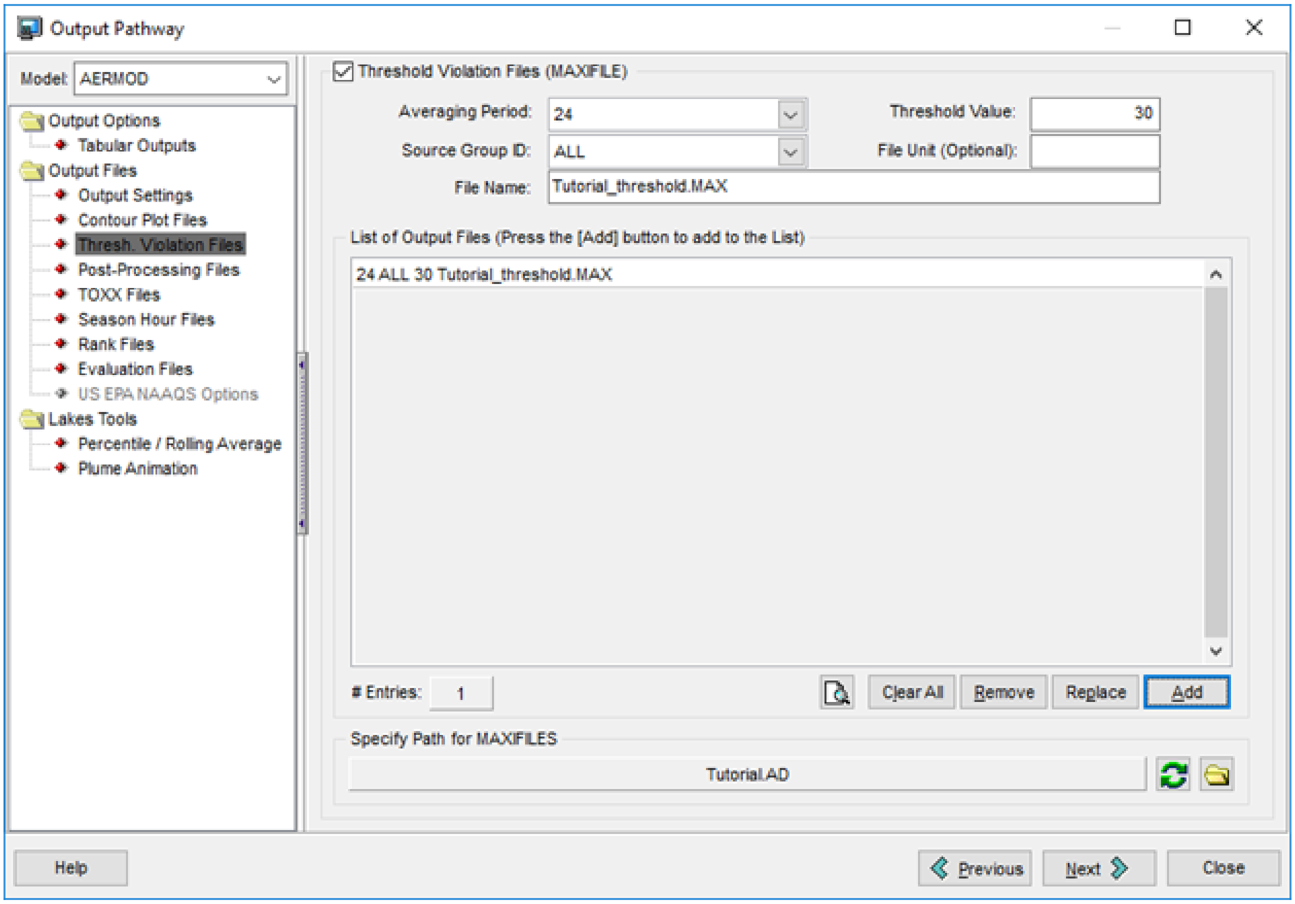
> If you are using the PSD Source Group option, you will notice that the following hard-coded source groups are available in this screen:

- NAAQS - for compliance to the National Ambient Air Quality Standards (NAAQS).
- PSDINC - for compliance with Prevention of Significant Deterioration (PSD) increment consumption

> See also Contour Plot File Format (PLOTFILE)

Threshold Violation Files

The **Threshold Violation Files** screen allows you to request the model to generate files of all occurrences of violations equal to or above a user-specified threshold value. Select **Output Files - Thersh. Violation Files** on the left of the Output Pathway dialog to access these setting.



Output Pathway - Threshold Violation Files

If you wish to specify threshold violation files, check the **Threshold Violation Files (MAXIFILE)** box. If this box is not checked, no threshold violation files will be generated for the current run, although you still can store any information already added to the **List of Output Files** to be used in later runs.

How to Define Threshold Violation Files:

1. Select from the **Averaging Period** drop-down list the averaging period for the threshold violation file. This drop-down list will contain all the averaging periods selected for your project in the Pollutant/Averaging screen in the Control Pathway.

2. Select from the **Source Group ID** drop-down list the source group ID for the threshold violation file. This drop-down list will contain a list of all Source Group IDs created in the Source Groups window.

> Only one threshold violation file may be used for each **Averaging Period/Source Group** combination.

3. Enter a file name for the threshold violation file in the **File Name** panel. The default extension is *.MAX.

> At the present time, the original U.S. EPA ISCST3 model can only accept file names up to eight characters long, with a three character extension to successfully create the file. The AERMOD and ISC-PRIME models have no such limitations.

4. Enter in the **Threshold Value** text box the user-specified threshold value to be used to identify the receptor locations where the concentration or deposition values are equal to or exceed this threshold value.

5. The **File Unit** optional parameter allows you to specify the Fortran logical file unit for the output file. The user-specified file unit must be in the range of 20-100, inclusive. If the **File Unit** is omitted, then the model will dynamically allocate a unique file unit for this file. You can combine results for different **Averaging Periods** and/or **Source Groups** into a single file, by specifying the same **File Name** and **File Unit**.

6. Click on the Add button to add the specified threshold violation file to the **List of Output Files** window. You can repeat Steps 1-5 as many times as necessary to created all the desired files. The list is used to store all threshold violation files specified. Only those combinations contained in the **List of Output Files** will be considered when running the model.

> If more than one Output Type is selected in the Control Pathway, then the threshold violation file will only apply to the first output type selected among: **Concentration**, **Total Deposition**, **Wet Deposition**, and/or **Dry Deposition** options and only the corresponding value will be output in the threshold violation file output file.

> The **Threshold Violation Files** option may produce very large files for runs involving a large number of receptors if a significant percentage of the results exceed the threshold value. These files can get extremely large in certain circumstances, even up to several hundred megabytes. Thus, be sure you have adequate space on your hard drive.

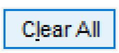
7. By default, all threshold violation files will be saved to preset locations. If you wish to specify an alternate location for the threshold violation files, click on the button in the **Specify Path for MAXIFILES** panel and select the location to save the files. If you wish to return to the default location, click on the button.

> The default locations where the plotfiles are saved are determined according to the model that is run:
> **ISCST3 Runs**: project directory\ProjectName.IS\
> **ISC-PRIME Runs**: project directory\ProjectName.PR\
> **AERMOD Runs**: project directory\ProjectName.AD\

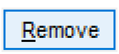
> A plotfile is automatically created and saved to the same directory as the generated MAXIFILE (e.g., CONC10.MAX and CONC10_exceedance.PLT). Plotfiles (*.PLT) created from MAXIFILES (*.MAX) contain information on the number of times the specified threshold value was exceeded at each receptor location. If you open a project created in an older version of AERMOD View, this plotfile will be created as long as the MAXIFILE options are specified in the **Output Pathway**.

There are a number of buttons available in the **Threshold Violation Files** panel. Their functions are described below:

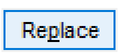

Click this button to view the contents of the selected file. Please note that you can preview the threshold violation file only after running the model.



Click on this button to clear all the files from the **List of Output Files** window.



Click on this button to remove the selected file from the **List of Output Files** window.



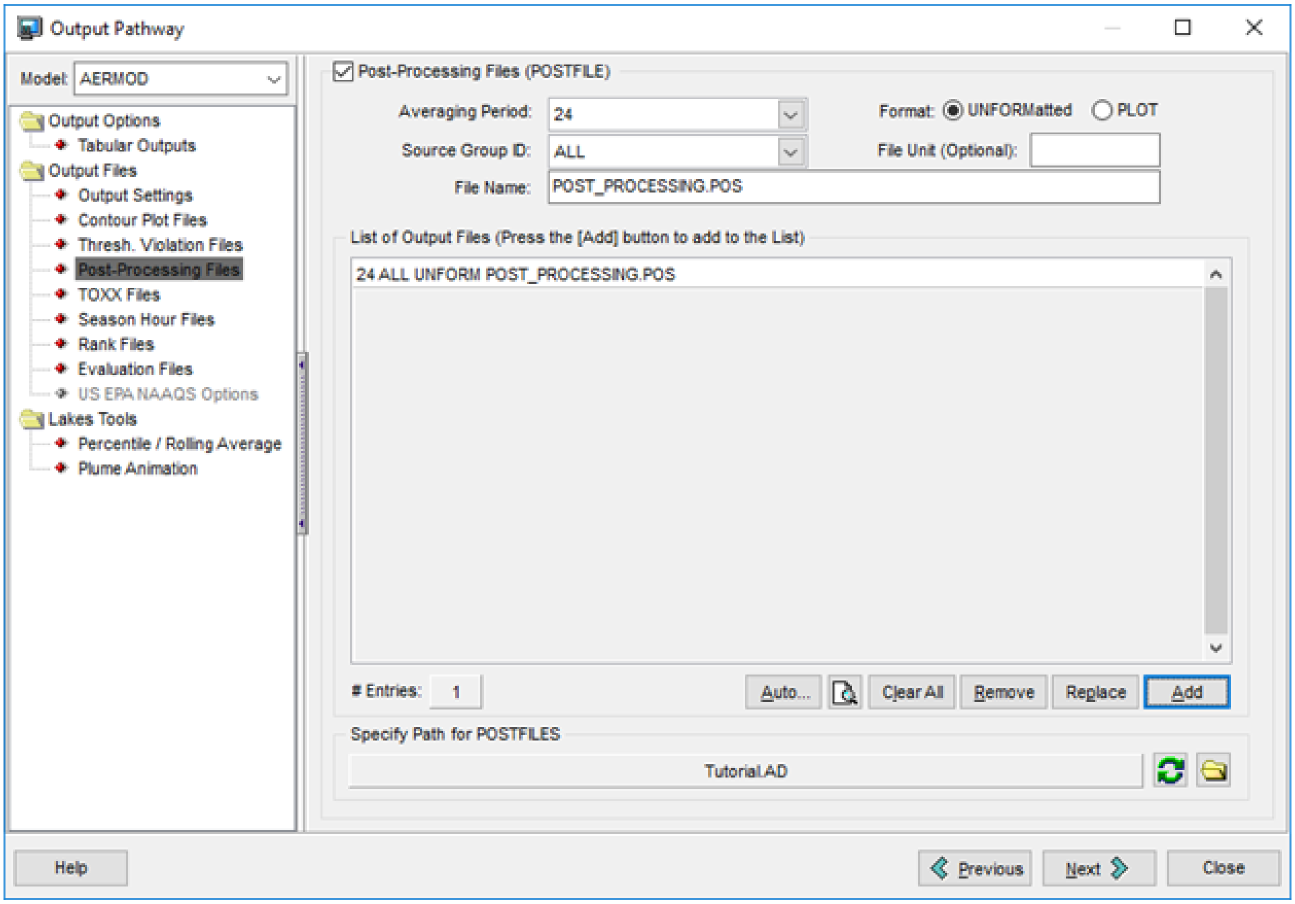
Click on this button to replace the selected file in the **List of Output Files** with the alternate parameters specified.

Click on this button to add the specified file parameters to the **List of Output Files** window.

> See also Threshold Violation File Format (MAXIFILE)

Post-Processing Files

The **Post-Processing Files** screen allows you to produce files of concurrent (raw) results at each receptor suitable for post-processing. Select **Output Pathway - Post-Processing Files** from the tree located on the left side of the Output Pathway dialog, to display the available post-processing file options.



Output Pathway - Post-Processing Files screen

If you wish to specify post-processing files, check the **Post-Processing Files (POSTFILE)** box. If this box is not checked, no post-processing files will be generated for the current run, although you still can store any information already added to the List of Output Files list to be used in later runs.

How to Define Post-Processing Files:

1. Select from the **Averaging Period** drop-down list the averaging period for the post-processing file. This drop-down list will contain all the averaging periods selected for your project in the Pollutant/Averaging screen in the Control Pathway.

2. Select from the **Source Group ID** drop-down list the source group ID for the post-processing file. This drop-down list will contain a list of all Source Group IDs created in the Source Groups window.

3. Enter a file name for the post-processing file in the **File Name** panel. The default extension is *.POS.

> At the present time, the original U.S. EPA ISCST3 model can only accept file names up to eight characters long, with a three character extension to successfully create the file. The AERMOD and ISC-PRIME models have no such limitations.

4. Specify whether you wish the POSTFILE output file to be either **Unformatted** or **Formatted**. The Unformatted option produces a POSTFILE in binary format while the Formatted option produces a POSTFILE in ASCII format.

5. The **File Unit** optional parameter allows you to specify the Fortran logical file unit for the output file. The user-specified file unit must be in the range of 20-100, inclusive. If the **File Unit** is omitted, then the model will dynamically allocate a unique file unit for this file. You can combine results for different **Averaging Periods** and/or **Source Groups** into a single file, by specifying the same **File Name** and **File Unit**.

6. Click on the Add button to add the specified post-processing file to the **List of Output Files** window. You can repeat Steps 1-5 as many times as necessary to created all the desired files. The list is used to store all post-processing files specified. Only those combinations contained in the **List of Output Files** will be considered when running the model.

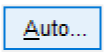
> If more than one Output Type is selected in the Control Pathway, then the specified Post-Processing File (POSTFILE) will include all of the output types selected in the following order: Concentration, Total Deposition, Dry Deposition, and/or Wet Deposition.

> The Post-Processing Files (POSTFILE) option can produce very large files and should be used with some caution. To estimate the size of the file (in bytes), use the following equation:
>
> $$\text{File Size (bytes)} = \frac{(\text{\# of Hrs/Yr})}{(\text{\# of Hrs/Ave})}\,(\text{\# of Rec} + 4)\ 4$$
>
> where:
> # of Hrs/Yr = Number of hourly values for a full year (8760 records)
> # of Hrs/Ave = Number of hours per average period
> # of Rec = Number of receptors
> Divide the result by 1000 to estimate the number of kilobytes (KB) and divide by 1.0E6 to estimate the number of megabytes (MB).

7. By default, all post-processing files will be saved to preset locations. If you wish to specify an alternate location for the post-processing files, click on the button in the **Specify Path for POSTFILES** panel and select the location to save the files. If you wish to return to the default location, click on the button.

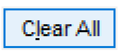
> The default locations where the post-processing files are saved are determined according to the model that is run:
> **ISCST3 Runs**: project directory\projectname.IS\
> **ISC-PRIME Runs**: project directory\projectname.PR\
> **AERMOD Runs**: project directory\projectname.AD\

There are a number of buttons available in the Post-Processing Files panel. Their functions are described below:

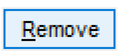


Opens the Auto-Generated POSTFILE Option dialog which allows you to set up all possible combinations for selected averaging periods and source groups.

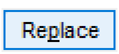

Press this button to view the contents of the selected file. Please note that you can preview the post-processing file only after running the model.



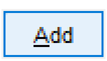
Click on this button to clear all the files from the List of Output Files window.



Click on this button to remove the selected file from the List of Output Files window.

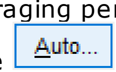


Click on this button to replace the selected file in the List of Output Files with the alternate parameters specified.



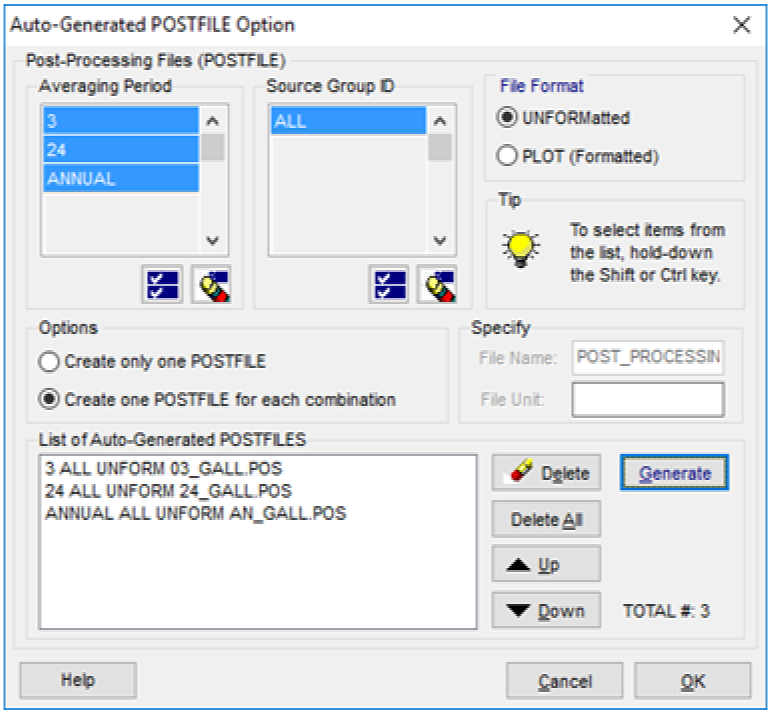
Click on this button to add the specified file parameters to the List of Output Files window.

Auto-Generated POSTFILE Option

The **Auto-Generated POSTFILE Option** allows you to quickly and easily set-up all possible combinations for selected averaging periods and source groups for your post files. You have access to this dialog by clicking on the 

button in the Post-Processing Files window.

> Any existing entries in the List of Output Files located in the Post-Processing Files dialog will be lost if you use the Auto-Generated POSTFILE Option. However you can add other combinations after you have used the Auto-Generated POSTFILE Option.



Auto-Generated POSTFILE Option

How to Use the Auto-Generated POSTFILE Option:

1. Select, from the **Averaging Period** list, the averaging period for the post-processing file. This list will contain all the averaging periods selected for your project in the Pollutant/Averaging window of the Control Pathway.

2. From the **Source Group ID** list window select the source group ID for the post-processing file. This list window will contain a list of all Source Group IDs created in the Source Groups window.

3. From the **File Format** panel specify whether you wish the post file to be either **Unformatted** or **Formatted**. The Unformatted option produces a post file in binary format while the Formatted option produces a post file in ASCII format.

4. From the Options frame, select one of the following options -

   - **Create only one POSTFILE:** This option creates only one post file that will contain results for all the selected combinations of averaging periods and source groups. If this option is selected, then you will be required to specify a **File Name** and the **File Unit**.

   - **Create one POSTFILE for each combination:** This option creates a post file for each combination of averaging period and source group. In this case, AERMOD View assigns file names for each post file.

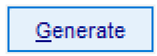
> The convention for the default name for the post files is avsgrpfm.plt, where:
> **av** = averaging period, the first 2 characters are reserved for defining the averaging period
> **sg** = source group, 4 characters reserved for defining the source group

> **fm** = format, the last 2 characters are reserved for defining the file format
> **Example:** 01GALLUN.POS
> 01 = 1 hour averaging period
> GALL = Source Group ALL
> UN = Unformatted

5. If you selected the **Create only one POSTFILE** option, then you need to define the following under the **Specify** frame:

   - **File Name:** Enter a file name, up to eight character long, for the post-processing file in the **File Name** panel. The default extension is *.POS.

   - **File Unit:** This optional parameter allows you to specify the Fortran logical file unit for the output file. The user-specified file unit must be in the range of 20-100, inclusive. If the File Unit is omitted, then the model will dynamically allocate a unique file unit for this file. By specifying the same File Name and File Unit, you can combine results for different combinations of averaging periods and source groups into a single post file.

6. Once you've selected all the options, press the 

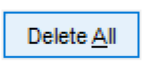
button to create all the specified combinations. The generated post files are placed in the List of Auto-Generated POSTFILES. If you press the OK button, all the generated post files will be automatically placed in the List of Output Files in the Post-Processing Files window.

There are a number of buttons found in the **Auto-Generated POSTFILE Option** dialog, their functions are described below:


Press this button to select all the items from the list.

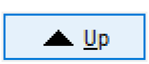

Press this button to clear the current selection.

Press this button to delete selected items from the list.



Press this button to delete all items from the list.

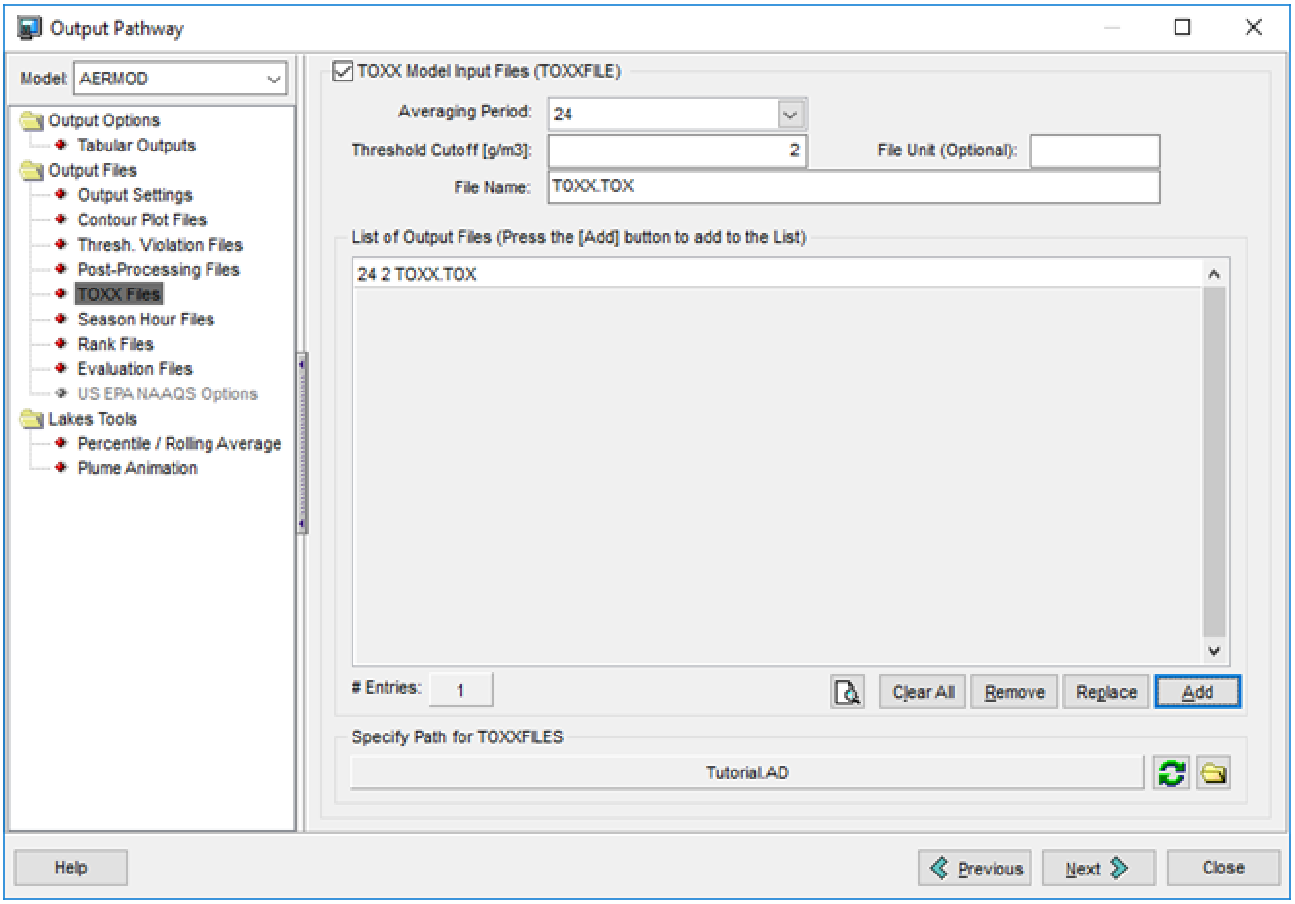
Press this button to move the selected item one row up.



Press this button to move the selected item one row down.

TOXX Files

In the **TOXX Files** screen you can produce unformatted (binary) files of raw results above a threshold value with a special structure for use with the TOXX model component of TOXST. Select **Output Pathway - TOXX Files** from the tree located on the left side of the Output Pathway dialog, to display the available options.



Output Pathway dialog - TOXX Files

If you wish to specify TOXX files, check the **TOXX Model Input Files (TOXXFILE)** box. If this box is not checked, no TOXX files will be generated for the current run, although you still can store any information already added to the List of Output Files list to be used in later runs.

How to Define TOXX Files:

1. Select from the **Averaging Period** drop-down list the averaging period for the TOXX file. This drop-down list will contain all the short-term averaging periods selected for your project in the Pollutant/Averaging screen in the Control Pathway.

> Only one TOXX file may be used for each averaging period. While the TOXXFILE option may be specified for any of the short-term averaging periods that were specified in the Control Pathway, the model will generate a non-fatal warning message if a period other than the 1-hour average is specified. This is because the TOXST model currently supports only 1-hour averages.

2. Enter in the **Threshold Cutoff** text box the user-specified threshold cutoff value. Note that the units of the Threshold Cutoff parameter are in g/m3, regardless of the input and output units selected in the Emission Output Units window.

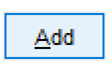
3. Enter a file name for the TOXX file in the **File Name** panel. The default extension is *.TOX.

> At the present time, the original U.S. EPA ISCST3 model can only accept file names up to eight characters long, with a three character extension to successfully create the file. The AERMOD and ISC-PRIME models have no such limitations.

> The TOXXFILE option can be specified for each averaging period, but a different File Name should be used for each file. Different file names should be specified for each file since the structure of the output file generated by the TOXXFILE option does not allow for a clear way to distinguish between results for different averaging periods.

4. The **File Unit** optional parameter allows you to specify the Fortran logical file unit for the output file. The user-specified file unit must be in the range of 20-100, inclusive. If the File Unit is omitted, then the model will dynamically allocate a unique file unit for this file. You can combine results for different Averaging Periods and/or Source Groups into a single file, by specifying the same File Name and File Unit.

> When using the TOXXFILE option, the user will normally place a single source in each source group.

5. Click on the 

button to add the specified TOXX file to the List of Output Files window. You can repeat Steps 1-5 as many times as necessary to created all the desired files. The list is used to store all TOXX files specified. Only those combinations contained in the List of Output Files will be considered when running the model.

> If more than one Output Type is selected in the Control Pathway, the TOXXFILE threshold will only apply to the first output type selected among the **Concentration**, **Total Deposition**, **Wet Deposition**, and/or **Dry Deposition**

options, and only the corresponding value will be output in the TOXXFILE output file.

The **TOXXFILE** option may produce very large files for runs involving a large number of receptors if a significant percentage of the results exceed the threshold value. These files can get extremely large in certain circumstances, even up to several hundred megabytes. Therefore, please be sure you have adequate space on your hard drive.

6. By default, all TOXX files will be saved to preset locations. If you wish to specify an alternate location for the TOXX files, click on the button in the **Specify Path** for TOXXFILES panel and select the location to save the files. If you wish to return to the default location, click on the button.

The default locations where the TOXX files are saved are determined according to the model that is run:
ISCST3 Runs: project directory\projectname.IS\
ISC-PRIME Runs: project directory\projectname.PR\
AERMOD Runs: project directory\projectname.AD\

There are a number of buttons available in the TOXX Model Input Files panel. Their functions are described below:

Press this button to view the contents of the selected file. Please note that you can preview the TOXX file only after running the model.

Clear All Click on this button to clear all the files from the List of Output Files window.

Remove Click on this button to remove the selected file from the List of Output Files window.

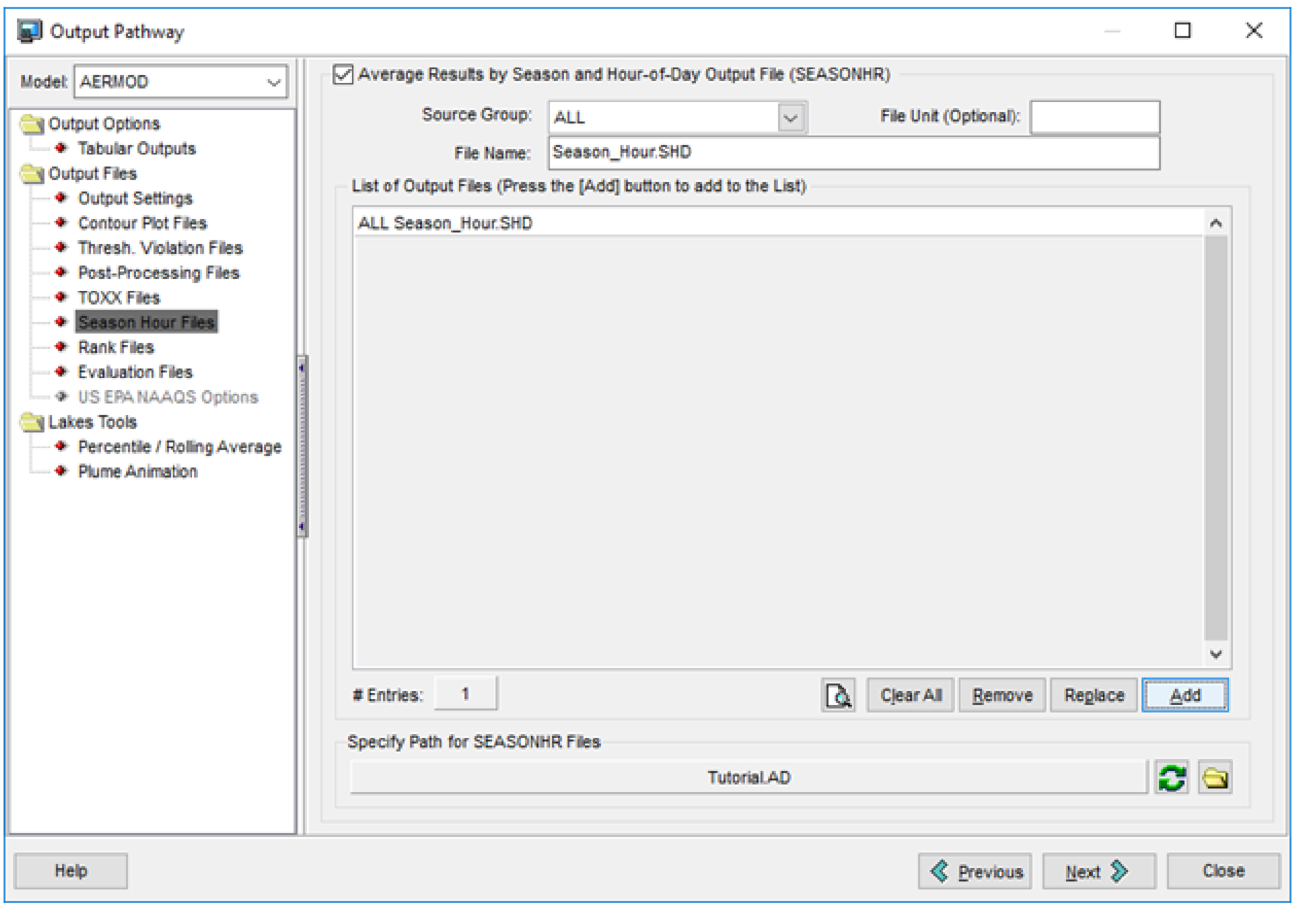
Replace Click on this button to replace the selected file in the List of Output Files with the alternate parameters specified.

Add Click on this button to add the specified file parameters to the List of Output Files window.

The TOXXFILE option is not compatible with the Re-Start File option. If you have selected the Re-Start File option on the Optional Files window, then AERMOD View will disable the options in the TOXX Model Input Files screen and will display a message indicating this option is not available.

Season Hour Files

The **Season Hour Files** screen allows you to produce output files containing average results for concentration, deposition, dry deposition and/or wet deposition by season and hour-of-day. Select **Output Pathway - Season Hour Files** from the tree located on the left side of the Output Pathway dialog, to display the available options.



Output Pathway - Season Hour Files screen

> If you are using the ISCST3 model, the Season Hour Files option is available only if you have selected the Non-Default, TOXICS, Season by Hour-of-Day Output Option in the Control Pathway.

If you wish to specify season hour files, check the **Average Results by Season and Hour-of-Day Output Files (SEASONHR)** box. If this box is not checked, no season hour files will be generated for the current run, although you still can store any information already added to the **List of Output Files** dialog to be used in later runs.

How to Define Season Hour Files:

1. Select from the **Source Group ID** drop-down list the source group ID for the season hour file. This drop-down list will contain a list of all Source Group IDs created in the Source Groups window.

2. Enter a file name for the season hour file in the **File Name** panel. The default extension is *.SHD.

> At the present time, the original U.S. EPA ISCST3 model can only accept file names up to eight characters long, with a three character extension to successfully create the file. The AERMOD and ISC-PRIME models have no such limitations.

3. The **File Unit** optional parameter allows you to specify the Fortran logical file unit for the output file. The user-specified file unit must be greater than 20. If the File Unit is left blank, then the model will dynamically assign a file unit.

4. Click on the 

button to add the specified season hour file to the **List of Output Files** window. You can repeat Steps 1-5 as many times as necessary to created all the desired files. The list is used to store all season hour files specified. Only those combinations contained in the List of Output Files will be considered when running the model.

5. All season hour files will be saved to the default location project directory\projectname.IS\. If you wish to specify an alternate location for the season hour files, click on the  button in the **Specify Path for SEASONHR Files** panel and select the location to save the files. If you wish to return to the default location, click on the  button.

There are a number of buttons available in this window. Their functions are described below:

 Press this button to view the contents of the selected file. Please note that you can preview the season hour file only after running the model.

Clear All Click on this button to clear all the files from the List of Output Files window.

Remove Click on this button to remove the selected file from the List of Output Files window.

Replace Click on this button to replace the selected file in the List of Output Files with the alternate parameters specified.

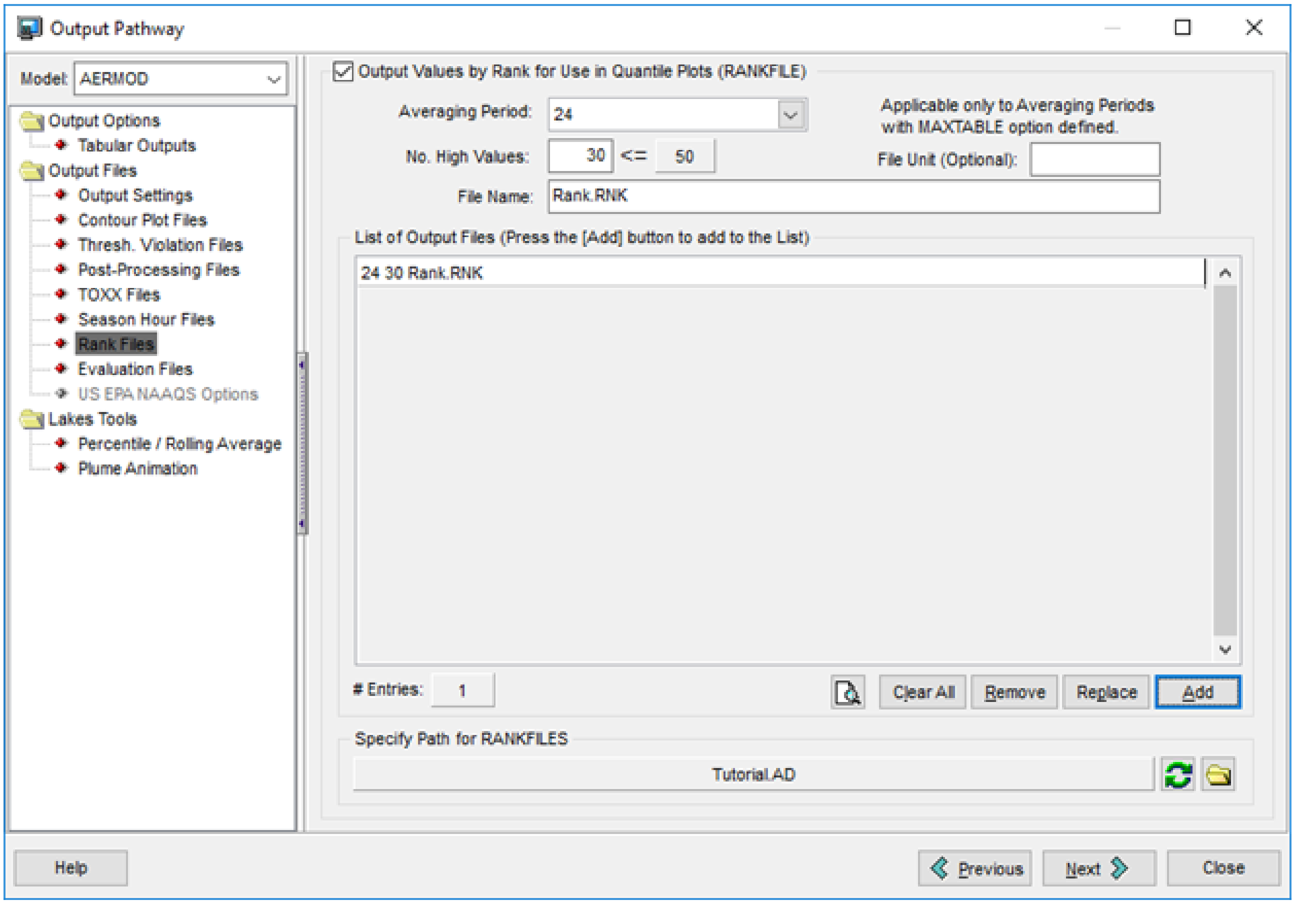
Add Click on this button to add the specified file parameters to the List of Output Files window.

See also Season by Hour File Format (SEAONHR)

Rank Files

AERMOD Only

The **Rank Files** screen allows you to output values by rank for use in Q-Q (quantile) plots. Select **Output Pathway - Rank Files** from the tree located on the left side of the Output Pathway dialog, to display the available options.



Output Pathway dialog - Rank Files

If you wish to specify rank files, check the **Output Values by Rank for Use in Quantile Plots (RANKFILE)** box. If this box is not checked, no rank files will be generated for the current run, although you still can store any information already added to the List of Output Files list to be used in later runs.

How to Define Rank Files:

1. Select from the **Averaging Period** drop-down list the averaging period for the rank file.

> In order to use the **Rank Files** option for a particular averaging period, the **Maximum Values** option should first be defined for the averaging period in the Tabular Outputs window.

2. Specify a value for the **No. High Values**. This must be less than or equal to the maximum value for the specified averaging period in the Tabular Outputs window.

3. Enter a file name for the rank file in the **File Name** panel. The default extension is *.RNK.

At the present time, the original U.S. EPA ISCST3 model can only accept file names up to eight characters long, with a three character extension to successfully create the file. The AERMOD and ISC-PRIME models have no such limitations.

4. The **File Unit** optional parameter allows you to specify the Fortran logical file unit for the output file. The user-specified file unit must be in the range of 25-100 inclusive. If the **File Unit** is left blank, then the model will dynamically assign a file unit.

5. Click on the Add button to add the specified rank file to the **List of Output Files** window. You can repeat Steps 1-5 as many times as necessary to created all the desired files. The list is used to store all rank files specified. Only those combinations contained in the List of Output Files will be considered when running the model.

6. All rank files will be saved to the default location project directory\projectname.AD\. If you wish to specify an alternate location for the rank files, click on the button in the **Specify Path for RANKFILES** panel and select the location to save the files. If you wish to return to the default location, click on the button.

There are a number of buttons available in this window. Their functions are described below:

Press this button to view the contents of the selected file. Please note that you can preview the rank file only after running the model.

Clear All Click on this button to clear all the files from the List of Output Files window.

Remove Click on this button to remove the selected file from the List of Output Files window.

Replace Click on this button to replace the selected file in the List of Output Files with the alternate parameters specified.

Add Click on this button to add the specified file parameters to the List of Output Files window.

See also Rank File Format (RANKFILE)

Rank File Format (RANKFILE)

The formatted data files generated by the RANKFILE includes several header records, each identified by an asterisk (*) in column one, followed by data records. See below the contents of the file:

Header

- Model Name + Model Version + First Title
- List of modeling option keywords applicable to the results
- Averaging period included in the file
- The number of ranked values included
- The Fortran format used for writing the data records
- Column headers for the variables included in the file

Data Record

The variables provided on each data record include:

- **RANK:** Rank
- **CONC:** Concentration value
- **DATE:** Date (YYMMDDHH)
- **X:** X coordinate for the receptor location
- **Y:** Y coordinate for the receptor location
- **ZELEV:** Receptor terrain elevation
- **ZHILL:** Receptor hill heights
- **FLAG:** Flagpole receptor height
- **GRP:** Source Group ID

Since only one occurrence per data period is included, the RANKFILE may not include the number of ranked values requested.

```
* AERMOD (04300): C:\ISCView4\Tutorial\tutorial.isc
* MODELING OPTIONS USED:
*  CONC                          TOXICS ELEV
*         RANK-FILE OF UP TO    20 TOP  3-HR VALUES FOR     1 SOURCE GROUPS
*         INCLUDES OVERALL MAXIMUM VALUES WITH DUPLICATE DATA PERIODS REMOVED
*         FORMAT: (1X,I3,1X,F13.5,1X,I8.8,2(1X,F13.5),3(1X,F7.2),2X,A8)
*RANK AVERAGE CONC   DATE          X              Y          ZELEV    ZHILL    ZFLAG     GRP
*____ _____________ ________ _____________ _____________ _______ _______ _______ ________
   1       0.00058 88012515  439096.71875 5298504.00000    0.00     0.00     0.00  ALL
   2       0.00042 88012909  439096.71875 5298504.00000    0.00     0.00     0.00  ALL
   3       0.00039 88012406  439096.71875 5298504.00000    0.00     0.00     0.00  ALL
   4       0.00036 88012324  439096.71875 5298504.00000    0.00     0.00     0.00  ALL
   5       0.00030 88012912  439096.71875 5298504.00000    0.00     0.00     0.00  ALL
   6       0.00030 88011709  439515.46875 5298836.50000    0.00     0.00     0.00  ALL
   7       0.00030 88011024  439375.87500 5298836.50000    0.00     0.00     0.00  ALL
   8       0.00029 88012209  439096.71875 5298770.00000    0.00     0.00     0.00  ALL
   9       0.00029 88013015  439375.87500 5298770.00000    0.00     0.00     0.00  ALL
  10       0.00029 88012115  439166.53125 5298304.50000    0.00     0.00     0.00  ALL
```

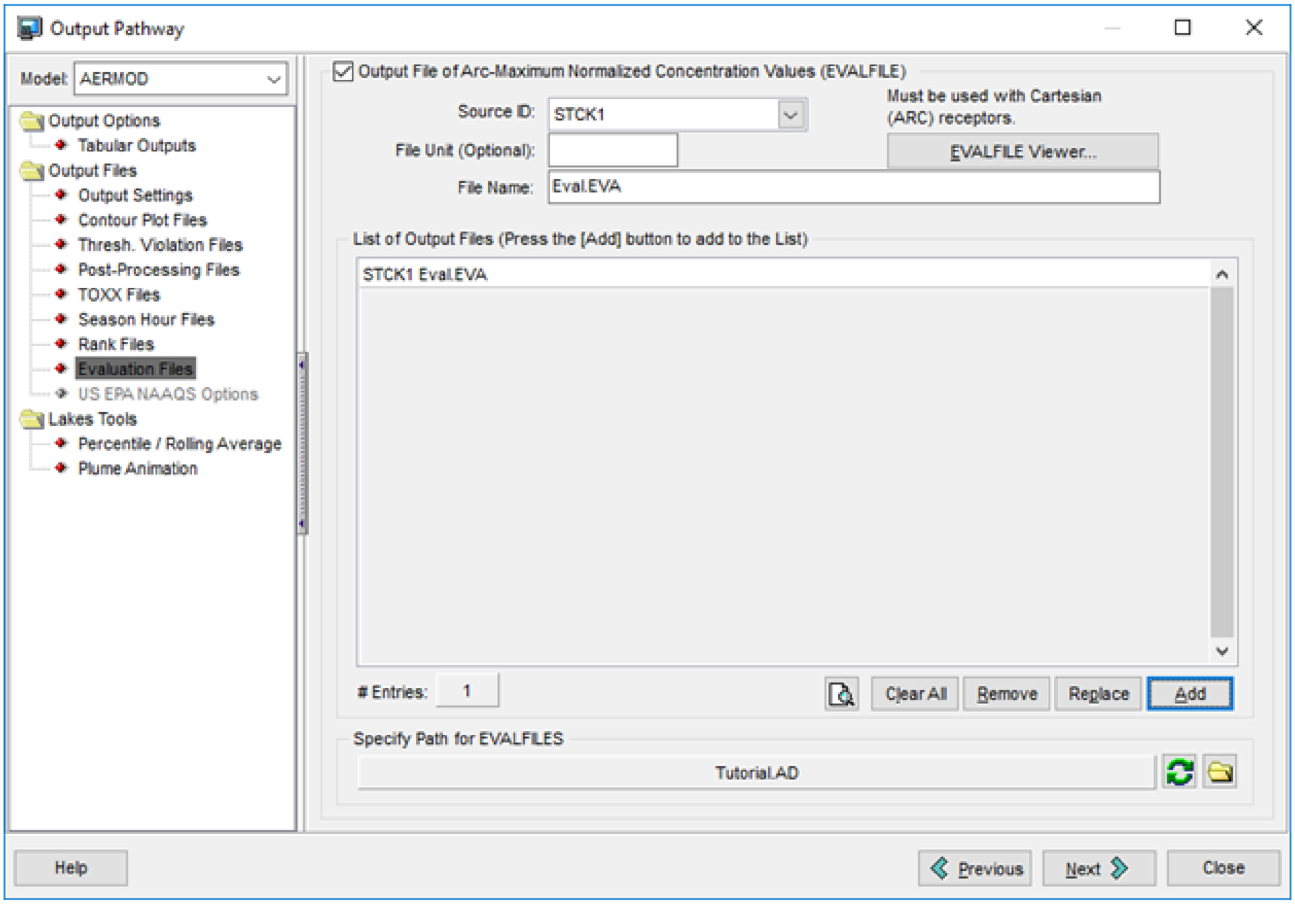
Header

Data Record

Evaluation Files

AERMOD Only

In the **Evaluation Files** screen you have the option to produces evaluation files including arc-maximum normalized concentration values for each hour of meteorology and for each source specified. The arc groupings of the receptors must be specified using the Discrete Cartesian (ARC) receptors. Select **Output Pathway - Evaluation Files** from the tree located on the left side of the Output Pathway dialog, to display the available options.



Output Pathway dialog - Evaluation Files

If you wish to specify evaluation files, check the **Output File of Arc-Maximum Normalized Concentration Values (EVALFILE)** box. If this box is not checked, no evaluation files will be generated for the current run, although you still can store any information already added to the List of Output Files dialog to be used in later runs.

How to Define Evaluation Files:

1. Select from the **Source ID** drop-down list source ID for which results are requested in the evaluation file. The drop-down list contains a list of all Source IDs created in the Source Inputs window.

2. Enter a file name for the evaluation file in the **File Name** panel. The default extension is *.EVA.

> At the present time, the original U.S. EPA ISCST3 model can only accept file names up to eight characters long, with a three character extension to

> successfully create the file. The AERMOD and ISC-PRIME models have no such limitations.

3. The **File Unit** optional parameter allows you to specify the Fortran logical file unit for the output file. The user-specified file unit must be in the range of 25-100 inclusive. If the File Unit is left blank, then the model will dynamically assign a file unit.

4. Click on the Add button to add the specified evaluation file to the List of Output Files window. You can repeat Steps 1-5 as many times as necessary to created all the desired files. The list is used to store all evaluation files specified. Only those combinations contained in the List of Output Files will be considered when running the model.

5. All evaluation files will be saved to the default location project directory\projectname.AD\. If you wish to specify an alternate location for the evaluation files, click on the button in the Specify Path for EVALFILES panel and select the location to save the files. If you wish to return to the default location, click on the button.

There are a number of buttons available in this window. Their functions are described below:

EVALFILE Viewer... Click on this button to display the EVALFILE Viewer dialog. Please note that you must run the model before you can view the Evaluation Files.

Press this button to view the contents of the selected file. Please note that you can preview the evaluation file only after running the model.

Clear All Click on this button to clear all the files from the List of Output Files window.

Remove Click on this button to remove the selected file from the List of Output Files window.

Replace Click on this button to replace the selected file in the List of Output Files with the alternate parameters specified.

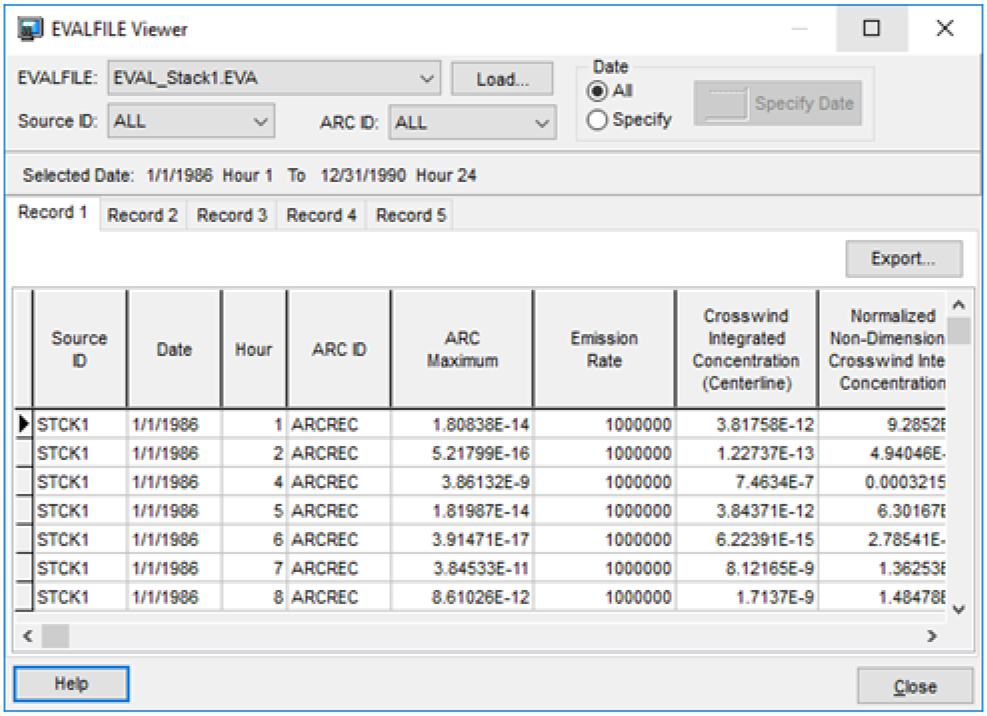
Add Click on this button to add the specified file parameters to the List of Output Files window.

> See also Evaluation File Format (EVALFILE)

EVALFILE Viewer

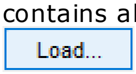
The **EVALFILE Viewer** dialog displays, in a grid, the contents of the EVALFILE file produced by the U.S. EPA AERMOD model. You have access to this dialog by clicking on the

button from the Evaluation Files window.



EVALFILE Viewer dialog

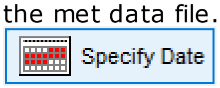
From the **EVALFILE** drop-down list you may select the evaluation file you wish to view. This list contains all the evaluation files created for the current project run. You can also click on the



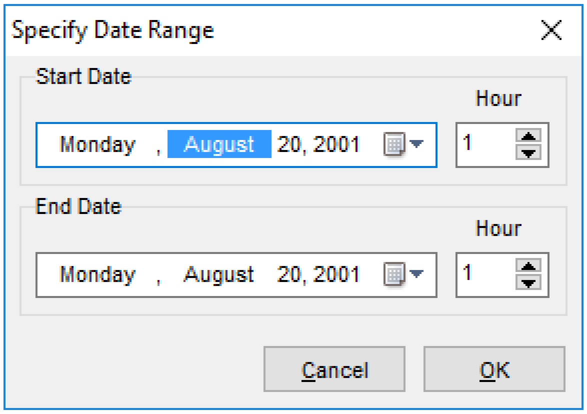
button to load and view a previously created evaluation file.

You may view data for a desired source by selecting it from the **Source ID** drop-down list. Similarly, you may view data by **ARC ID** by selecting the desired ARC ID from the drop-down list.

Each **Record** tab contains different measured parameters for the sources specified, for each hour of the met data file. If you wish to view the results for a specific date and/or time, click on the

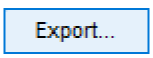


button to display the **Specify Date Range** dialog.



Specify Date Range dialog

In this dialog you can specify the start and end year, month, day and hour for which you wish to view data. Once selected, click on the **OK** button and return to the **EVALFILE** Viewer dialog where the data will be displayed for the selected date range.



Using this button allows you to export the EVALFILE data into a comma delimited file (*.csv) where you can open the data in Microsoft Excel to further analyze your data.

When you click on the **Export...** button, you will be required to select either current record or all records. By selecting the **Current Record...** you will only be exporting what is currently displayed in the grid for the current tab record. Alternatively, if you select **All Records...** you will be exporting the data currently displayed on all of the five record tabs.

List of Parameters

The EVALFILE grid is composed of five tabs. Each tab contains information for the selected sources and the specified hour(s) of meteorological data. The information provided in each record is displayed below.

All Records	
	Source ID (eight characters)

	Date variable (DD/MM/YY)
	Hour variable (1-24)
	Arc ID (eight characters)
Record 1	
	Arc maximum χ/Q
	Emission rate for arc maximum (including unit conversions)
	Crosswind integrated concentration based on true centerline concentration
	Normalized non-dimensional crosswind integrated concentration
Record 2	
	Downwind distance corresponding to arc maximum (m)
	Effective wind speed corresponding to arc maximum (m/s)
	Effective σv corresponding to arc maximum (m/s)
	Effective σw corresponding to arc maximum (m/s)
	σy corresponding to arc maximum (m)
	Effective plume height corresponding to arc maximum (m)
Record 3	
	Monin-Obukhov length for current hour (m)
	Mixing height for current hour (m)
	Surface friction velocity for current hour (m/s)
	Convective velocity scale for current hour if unstable (m/s), or σz for current hour if stable (m)
	Buoyancy flux for current hour (m4/s3)

	Momentum flux for current hour (m4/s2)
Record 4	
	Bowen ratio for current hour
	Plume penetration factor for current hour
	Centerline χ/Q for direct plume
	Centerline χ/Q for indirect plume
	Centerline χ/Q for penetrated plume
	Nondimensional downwind distance
Record 5	
	Plume height/mixing height ratio
	Non-dimensional buoyancy flux
	Source release height (m)
	Arc centerline χ/Q
	Flow vector for current hour (degrees)
	Effective height for stable plume reflections (m)

US EPA NAAQS Options

AERMOD Only

The **US EPA NAAQS Options** screen is only available if the following condition are set in Pollutant/Averaging section of Control Pathway:

- **Pollutant: SO2**, **NO2**, or **PM-2.5 NAAQS**
  - **If NO2** is selected:
    - **Averaging Time Options: 1-Hour** (uncheck all others)
    - **1-Hour Average Options:** Either option
  - **If SO2** is selected:
    - **Averaging Time Options: 1-Hour** (uncheck all others)
    - **1-Hour Average Options:** Either option

- If **PM-2.5 NAAQS** is selected:
  - **Averaging Time Options: 24-Hour**
  - **24-Hour Average Options:** Either option

Introduced with the US EPA AERMOD Version 11059, three new output file options have been incorporated on the OU Pathway to support processing for the 1-hour NO2 and 1-hour SO2 NAAQS, especially the analyses that may be required to determine a source's (or group of sources) contributions to modeled violations of the NAAQS for comparison to the Significant Impact Level (SIL).

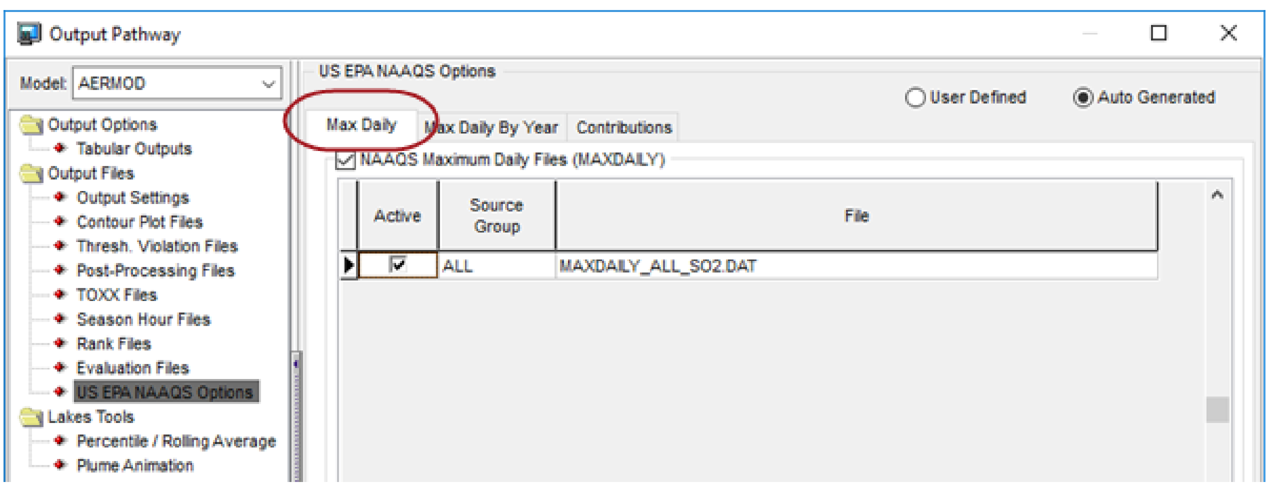
The 1-hour NO2 and SO2 NAAQS is based on the annual distribution of daily maximum 1-hour values, averaged across the number of years processed.

The three output options are described below:

NAAQS Maximum Daily File (MAXDAILY)

Output file of daily maximum 1-hour concentrations for a specified source group, for each day in the data period processed.

The MAXDAILY option is only applicable for 1-hour NO2 and 1-hour SO2 NAAQS



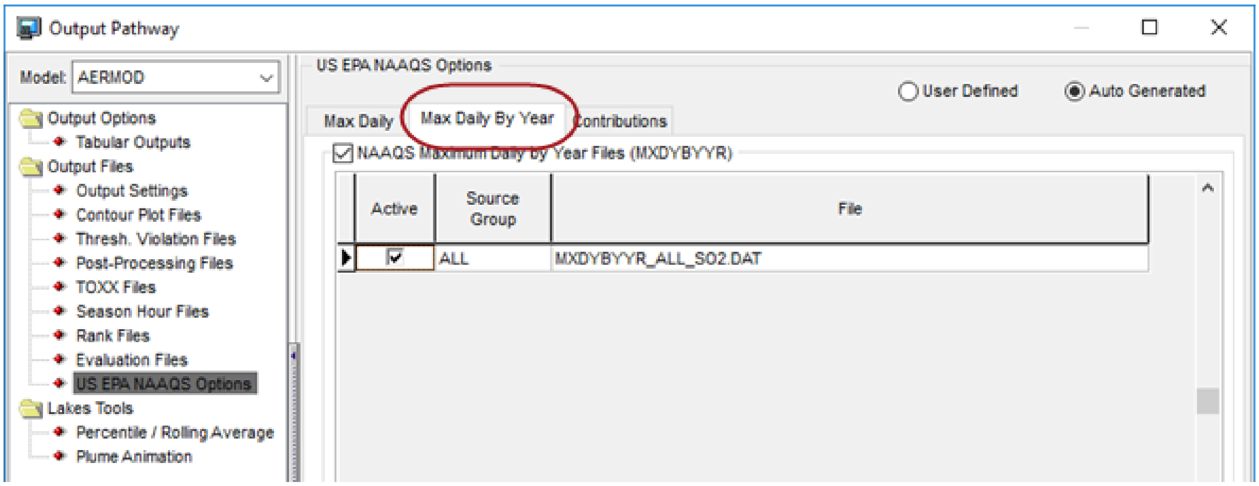
- **User Defined:** If this option is selected, the user has full control of which options to specify (file name and Source Group) for the MAXDAILY option.

- **Auto Generated:** This option automatically defines a MAXDAILY file for each for each Source Group specified in your project. These entries can be made active or inactive.

NAAQS Maximum Daily by Year Files (MXDYBYYR)

Output file with a summary of daily maximum 1-hour concentrations by year for each rank specified on the RECTABLE keyword.

> The MYDYBYYR option is only applicable for 1-hour NO2 and 1-hour SO2 NAAQS

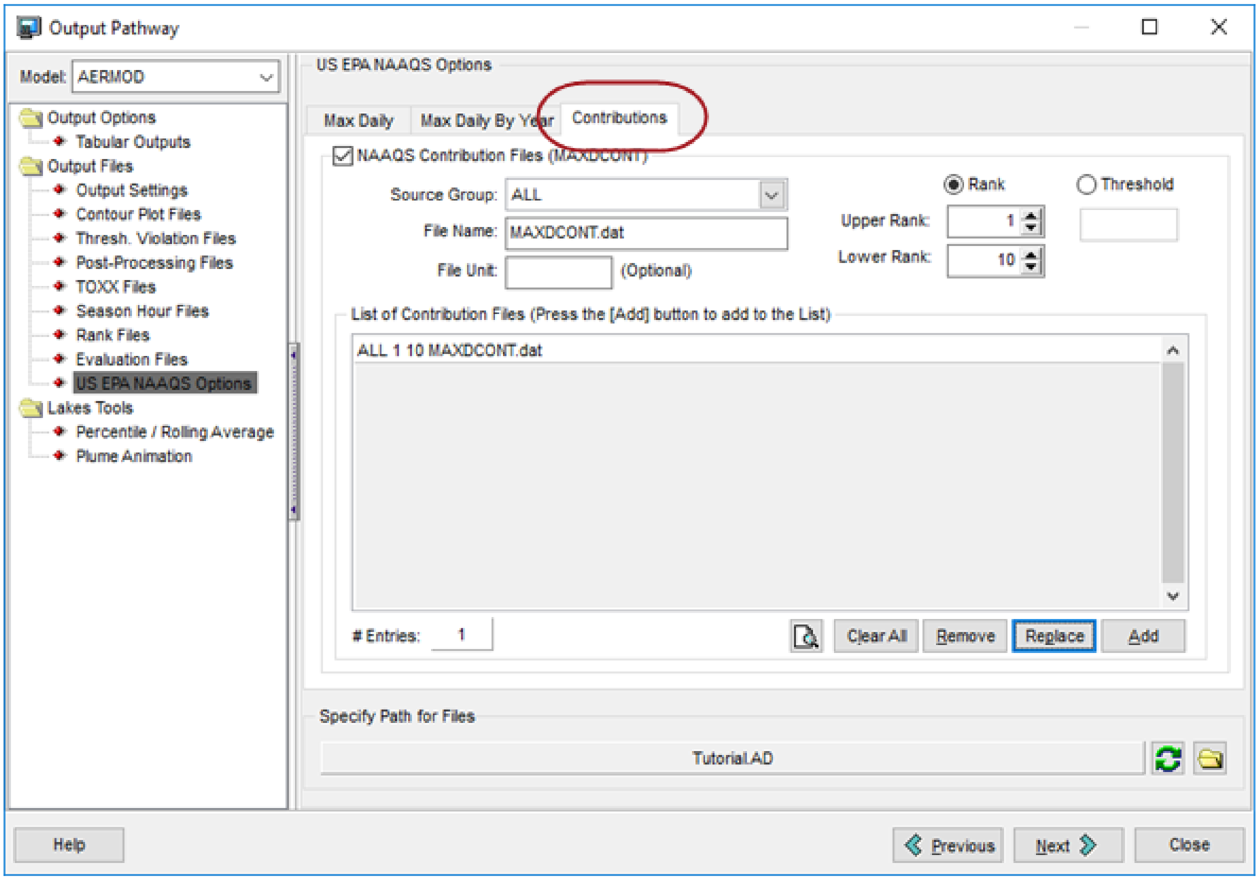


- **User Defined:** If this option is selected, the user has full control of which options to specify (file name and Source Group) for the MYDYBYYR option.
- **Auto Generated:** This option automatically defines a MYDYBYYR file for each for each Source Group specified in your project. These entries can be made active or inactive.

NAAQS Contribution Files (MAXDCONT)

Option to output contributions of each source group to ranked values averaged across years for a reference source group, paired in time and space.

> The **MAXDCONT** option is only applicable for 24-hour PM2.5, 1-hour NO2, and 1-hour SO2 NAAQS



For the MAXDCONT option, the user is responsible for defining the options to be included in the AERMOD run by specifying the following:

- **Source Group:** this is the source group toward which contributions should be determined
- **File Name:** this is the file name and it can be up to 200 characters in length
- **File Unit:** this is the optional file unit
- **Rank option:**
  - **Upper Rank:** this is the upper bound ranks (implies higher concentrations)
  - **Lower Rank:** this is the lower bound ranks (implies lower concentrations)
- **Threshold option:**
  - **Threshold Value:** this is the user-specified concentration threshold for Source Group impacts which serves as a lower bound on the range of ranks analyzed

> **Warning:** Once the above MAXDCONT parameters are defined, you must press the Add button to include these option to the **List of Contribution Files**.

 Press this button to view the contents of the selected file. Please note that you can preview the MAXDCONT file only after running the model.

Clear All Click on this button to clear all the files from the **List of Contribution Files** window.

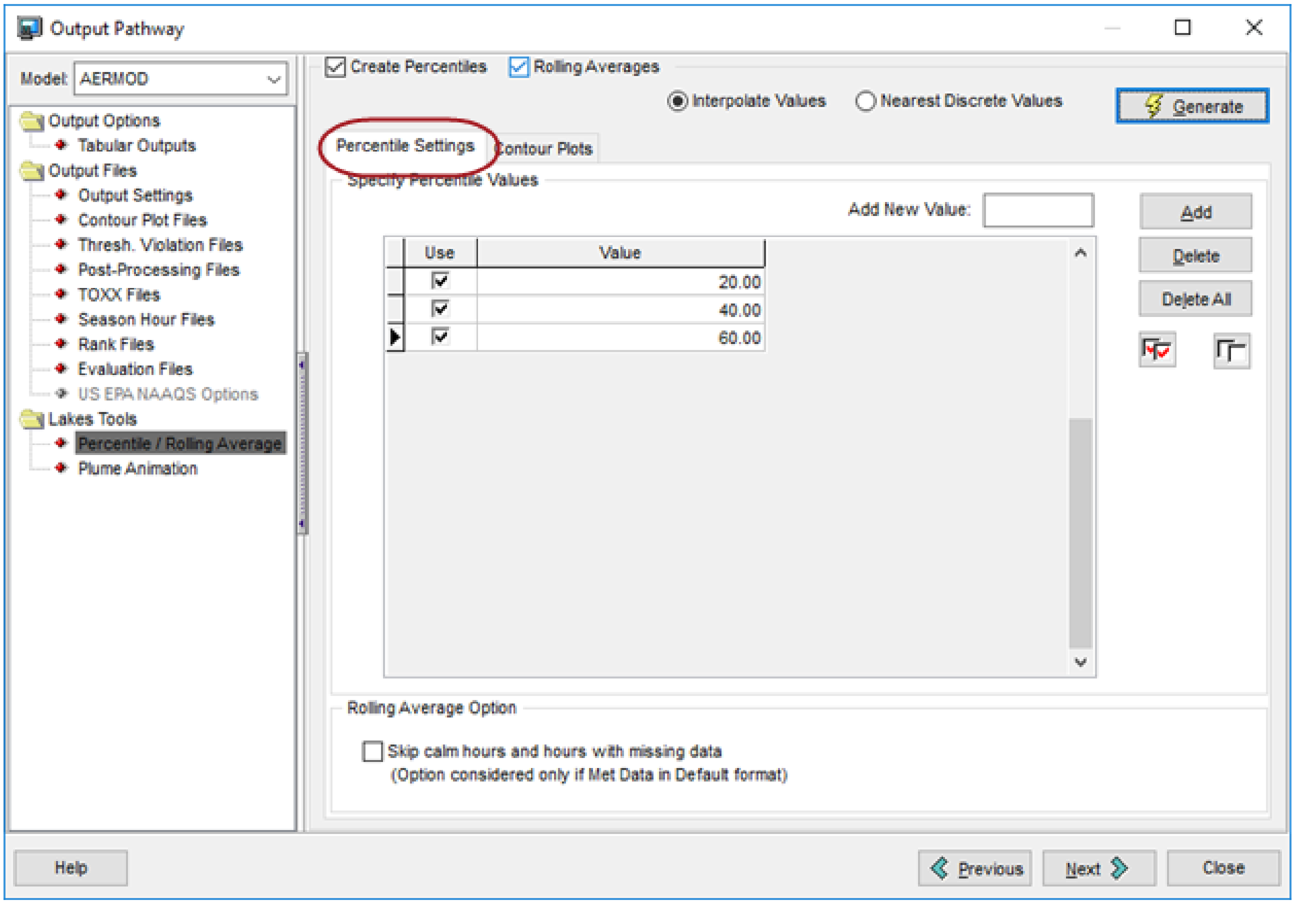
Remove Click on this button to remove the selected file from the **List of Contribution Files** window.

Replace Click on this button to replace the selected file in the **List of Contribution Files** with the alternate parameters specified.

Add Click on this button to add the specified file parameters to the **List of Contribution Files** window.

Percentiles/Rolling Average Settings

In the **Percentiles/Rolling Average Settings** screen you specify the settings in order to generate percentile plot files. Percentile values are routinely used to express air quality standards in an international setting. Select **Output Pathway - Percentiles/Rolling Average** from the tree located on the left side of the Output Pathway dialog, to display the **Percentile Settings** tab options.



Output Pathway dialog - Percentiles/Rolling Average - Settings

How to Specify the Percentile and/or Rolling Averages Settings:

1. At the top of the screen you may check either one or both of the following options:

   - **Create Percentiles:** Check this box if you wish to generate percentile plots.
   - **Rolling Averages:** Check this box to generate rolling or running average plots. This type of average differs from the U.S. EPA averaging calculations, also known as block averages.
   - **Interpolate Values:** Select this option if you want the values to be interpolated in the percentile plotfile.
   - **Nearest Discrete Values:** Select this option if you want the nearest discrete value to be output in the percentile plotfile.

See Also: Calculating Rolling Averages.

2. If the **Create Percentiles** option was checked, then you need to specify for which percentiles plotfiles should be created. Type in the percentile value in the **Add New Value** field and then press the Add button to add this value to the table. You may repeat this for as many percentiles as you wish to add. Only percentile values added to the table and checked for use will be considered when percentile plotfiles are generated.

3. When creating the percentile plot files a temporary post file (*.POS) is generated. Please note that this post file can be very large.

> The percentile and rolling averages plot files are features created by Lakes Environmental that use the U.S. EPA model-generated POSTFILES. If POSTFILES are present in your project folder, you can specify different averaging periods and percentile values without having to run the model again.

4. If the **Rolling Averages** option was checked, you can specify that calm hours and hours with missing data should not be considered when calculating the rolling averages. As long as your met data is in the default format, then AERMOD View will calculate rolling averages by skipping calm hours and hours with missing data. Check the **Skip calm hours and hours with missing data** check box to use this option.

5. Once you have specified your settings continue to the Percentiles/Rolling Average - Contour Plot Files screen to preview the list of the percentiles/rolling averages plot files to be generated when you run the model.

The following buttons are available for use in this window:

Add Adds percentile values to the table

Delete Deletes the selected percentile value from the table.

Delete All Deletes all the percentile values from the table.

 Press this button to check the Use column box for all the percentile entries in the table. Only the percentile values that are checked will be considered when generating the plotfiles.

 Press this button to uncheck the Use column box for all the percentile entries in the table. Any percentile value that is not checked will not be considered when generating the plotfiles.

Calculating Rolling Averages

Rolling averages are a requirements in the state of New Jersey, (United States) and some European countries. The rolling average calculations differ from the U.S. EPA averaging calculations, also known as block averages in the following way:

See below an example for the 3-hour average:

Hour	Concentration Value	U.S. EPA 3-Hour Averaging Calculations	Percent View Rolling Averages
1	10	---	---
2	10	---	---
3	40	20	20
4	40	---	30
5	10	---	30
6	10	20	20

Based on the above example, the U.S. EPA 3-hour averaging is calculated in the following way:

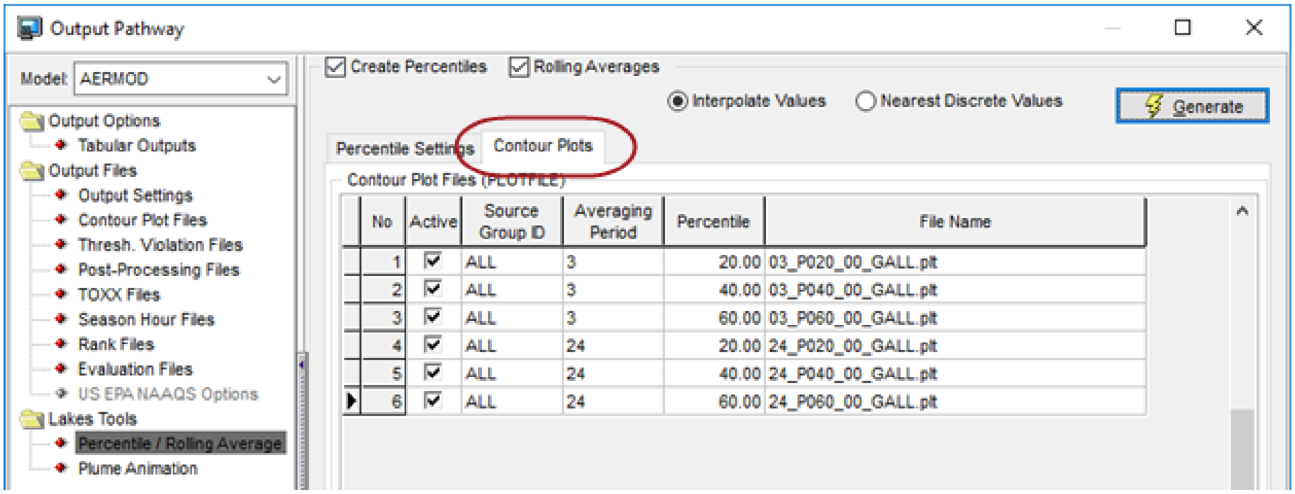
- 1st hour + 2nd hour + 3rd hour = 60 divided by 3 hours = 20
- 4th hour + 5th hour + 6th hour = 60 divided by 3 hours = 20

The rolling average implemented in Percent View is calculated in the following way:

- 1st hour + 2nd hour + 3rd hour = 60 divided by 3 hours = 20
- 2nd hour + 3rd hour + 4th = 90 divided by 3 hours = 30
- 3rd hour + 4th hour + 5th hour = 90 divided by 3 hours = 30
- 4th hour + 5th hour + 6th hour = 60 divided by 3 hours = 20

Percentiles/Rolling Average Contour Plot Files

The **Percentiles/Rolling Average Contour Plots** screen displays all the possible combinations of **Percentile/Rolling Average** plot files that were setup for your project in the Percentiles/Rolling Average Settings page. Select **Output Pathway - Percentiles/Rolling Average Contour Plots** from the tree located on the left side of the Output Pathway dialog, to display the available options.



Output Pathway dialog - Percentiles/Rolling Average - Contour Plots tab

Each plot file to be created is displayed in this dialog in tabular format. You can check or uncheck the **Create Percentiles** and **Rolling Averages** boxes to view the corresponding plot files. Described below are each of the fields in this table -

- **Active:** This field controls whether the plot files should be created or not. AERMOD View will only generate plot files that are checked or active.

- **Source Group ID:** This field displays the Source Group ID for which the plot files are being created.

- **Averaging Period:** This field displays the Averaging Period for which the plot files are being created. A plot file is created for each short-term averaging period selected for your project in the Pollutant/Averaging screen in the Control Pathway.

- **Percentile:** This field displays the percentile value for which the plot file is being created. This field is not displayed if only the **Rolling Averages** option is checked.

- **High Value:** This field displays the High Value (1st, 2nd, 3rd, etc.) for which the plot file is being created. This field will be available if only the **Rolling Averages** option was selected.

- **File Name:** This field displays the file name for the plotfile to be created.

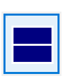
The convention for the default name for the plot files for Percentiles is av_percen_srgp.plt, where:
av = averaging period, the first 2 characters are reserved for defining the averaging period
percen = percentile, 6 characters are reserved for defining the percentile
srgp = source group, 4 characters reserved for defining the source group (e.g., GALL, G001, etc.)
Example: 03_P99_50_GALL.PLT
03 = 3 hour averaging period
P99_50 = 99.50 Percentile
GALL = Source Group ALL

The convention for the default name for the plot files for Rolling Averages is av_srg_hv.plt, where:
av = averaging period, the first 2 characters are reserved for defining the averaging period
srg = source group, 3 characters reserved for defining the source group (e.g. ALL, 001, 002)
hv = high value (e.g. h1, h2, h3..., h10)
Example: 03_ALL_H1.PLT
03 = 3 hour averaging period
ALL = Source Group ALL
H1 = 1st high value

By default, all plot files will be saved to the "Percentile" folder within your project folder - please see the tip below. If you wish to specify an alternate location for the plot files, click on the  button in the Specify Path for Percentiles/Rolling Average PLOTFILES panel and select the location to save the files. If you wish to return to the default location, click on the  button.

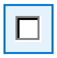
The default locations where the plot files are saved are determined according to the model that is run:
ISCST3 Runs: project directory\projectname.IS\Percentile
ISC-PRIME Runs: project directory\projectname.PR\Percentile
AERMOD Runs: project directory\projectname.AD\Percentile

There are a number of buttons available in this window. Their functions are described below:

 Click on this button to select all plot files in the list.

 Click on this button to unselect any selected plot files in the list.

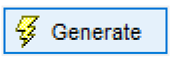
 Click on this button to mark the selected plot files as Active. Only active plot files will be generated.

 Click on this button to mark the selected plot files as inactive.

 Click on this button to mark all plot files in the list as Active. Only active plot files will be generated.

 Click on this button to mark the selected plot files as inactive.

 Select this option to preview the contents of the selected plot file. Please note that you can preview the plot file only after running the model.



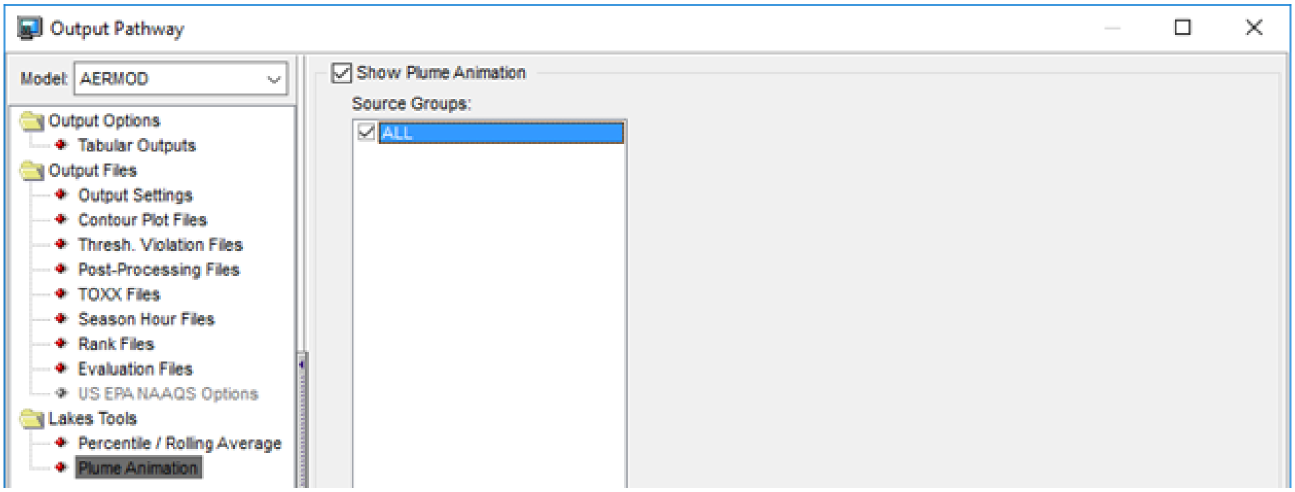
Click on this button to regenerate percentile contour plot files without re-running your project. See note below.

> Do not check the Delete Temporary 1-Hour Binary POSTFILE *.POS in the Percentiles/Rolling Average Settings screen if you would like to use the Generate option. AERMOD View generates the percentile/rolling average contour plot files by extracting information from the model generated POSTFILES.

Plume Animation

The **Plume Animation** section allows you to display plume animation such that you can view the individual plumes based on the selected time range, source groups and output type. This feature generates plume animation by extracting information from the AERMOD model-generated **POSTFILES**.

AERMOD View's **Plume Animation** feature utilizes the 1-hour average POSTFILE output to produce hourly contours. Note that this file can grow to very large sizes when processing a huge number of time periods with numerous receptors.



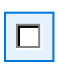
If you wish to generate the plume animation regarding your project, please follow the steps outlined below.

1. Enable the **Show Plume Animation** option.

    - Select all groups

    - Deselect all groups

    - Select currently highlighted group

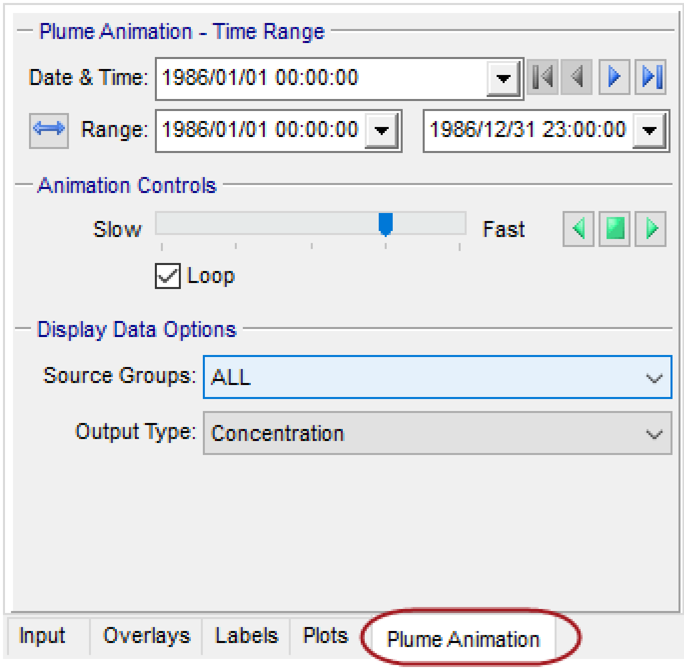
    - Deselect currently highlighted group

2. Select the corresponding **Source Groups** from which you wish to display pollutant concentration contours.

3. When finished, click **Close**.

4. The **Plume Animation tab** now appears in **Tree View** (as shown below). However, you need to first run the AERMOD model successfully in order to generate results for plume animations.

5. After the AERMOD model is run successfully, click on the **Plume Animation** tab and view individual plume animations.



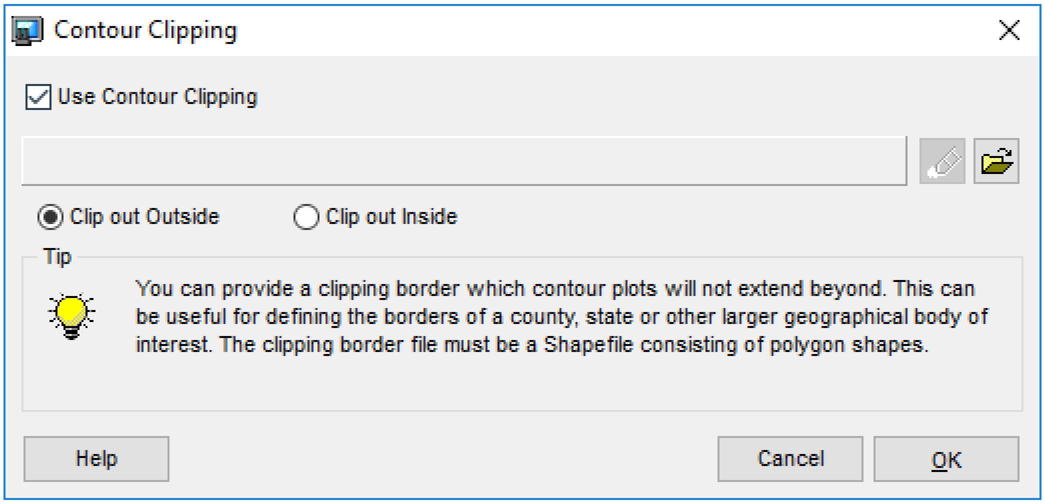
See Also: Tree View - Plume Animation Tab.

Contour Clipping

The contour clipping option gives you the ability to clip the contours along a specified boundary. The boundary is defined by specifying a shapefile which contains the shape of the desired clipping region. For example, you could clip to the borders of a county or town.

How To Use Contour Clipping:

1. You can access the **Contour Clipping** dialog by selecting **Output | Contour Clipping...** from the menu. The **Contour Clipping** dialog appears.



Contour Clipping dialog

2. Click the **Use Contour Clipping** check box. This clickable option is useful later on if you ever want to turn clipping on or off at anytime.

3. Click the **Open File** button and specify a shapefile containing the clipping region you wish to define.

4. Click the **OK** button to apply the clipping and close the **Contour Clipping** dialog.

Building Inputs

The **Building Inputs** dialog allows you to define information for the buildings in your project. You can either define your buildings in text mode or in graphical mode.

To define buildings in the text mode, simply enter all the necessary information directly into the **Building Inputs** dialog. You have access to this dialog by selecting **Data | Building Inputs...** from the menu or by clicking on the **Building** 

menu toolbar button.

To define buildings in the graphical mode, you can use any of the four building tools located on the Application Toolbar. The **Building Inputs** dialog opens automatically with some parameters already defined. Complete the necessary inputs to finish defining your building.

The parameters to be specified change depending on the type of building you are defining. Refer to the following sections for the building specific parameters:

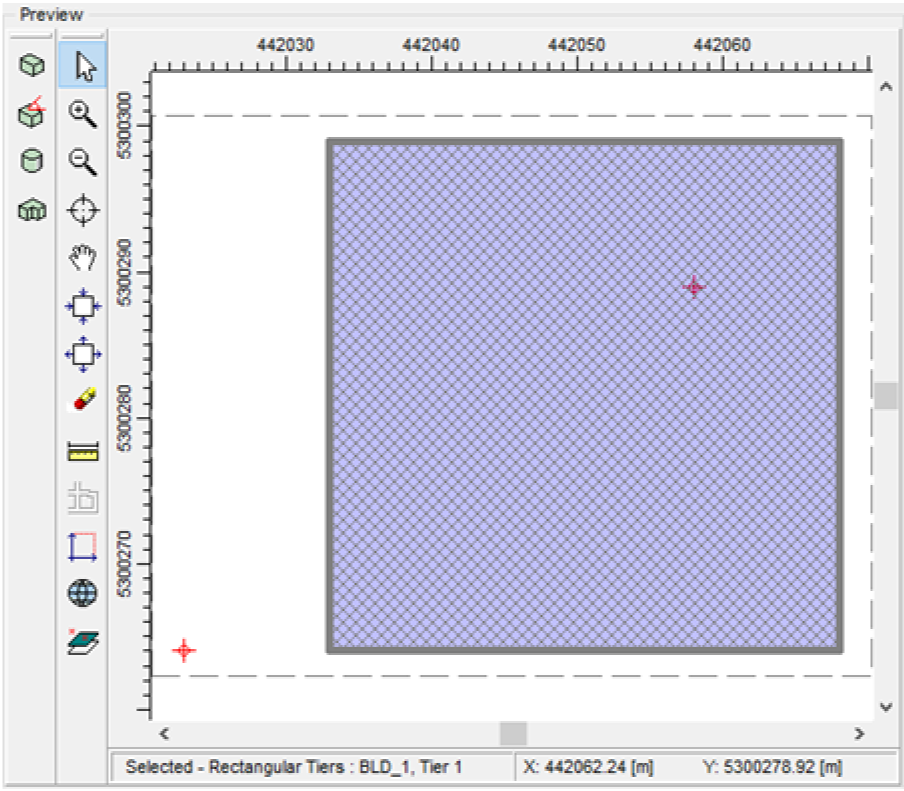
- Rectangular Building Inputs
- Angled Rectangular Building Inputs
- Circular Building Inputs
- Polygonal Building Inputs

The following options in the **Building Inputs** dialog are always visible and common, independent of the type of building being defined. See a description of these options below:

Preview Panel

This area displays the location of the buildings already defined. You can also graphically define your buildings in this area using the building tools, instead of the main drawing area of the interface.

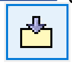
> The **Preview** panel will display any maps available in the project, however if you zoom in to a certain level, the raster maps will no longer be displayed.



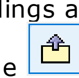
Preview Panel - Building Inputs dialog

- **Building Tools:** You can use any the building tools to graphically define your buildings. See the following sections on how to use these tools -
  - How to Graphically define a Rectangular Building
  - How to Graphically define an Angled Rectangular Building
  - How to Graphically define a Circular Building
  - How to Graphically define a Polygonal Building
- **Annotation Tools:** The annotation tools allow you import base maps, manipulate what is visible in the preview area, zoom in and pan on locations of interest. These tools are also found on the Annotation Toolbar in the main interface.

Import Buildings

The Building Inputs dialog allows you the option of importing buildings from either an Excel spreadsheet with Lakes Format (*.xls, *xlsx) or a BPIP Input File (*.bpi, *.pip, *.inp, *.prm) into your project. Simply click on the  button and specify the name and location of the file to import.

Export Buildings

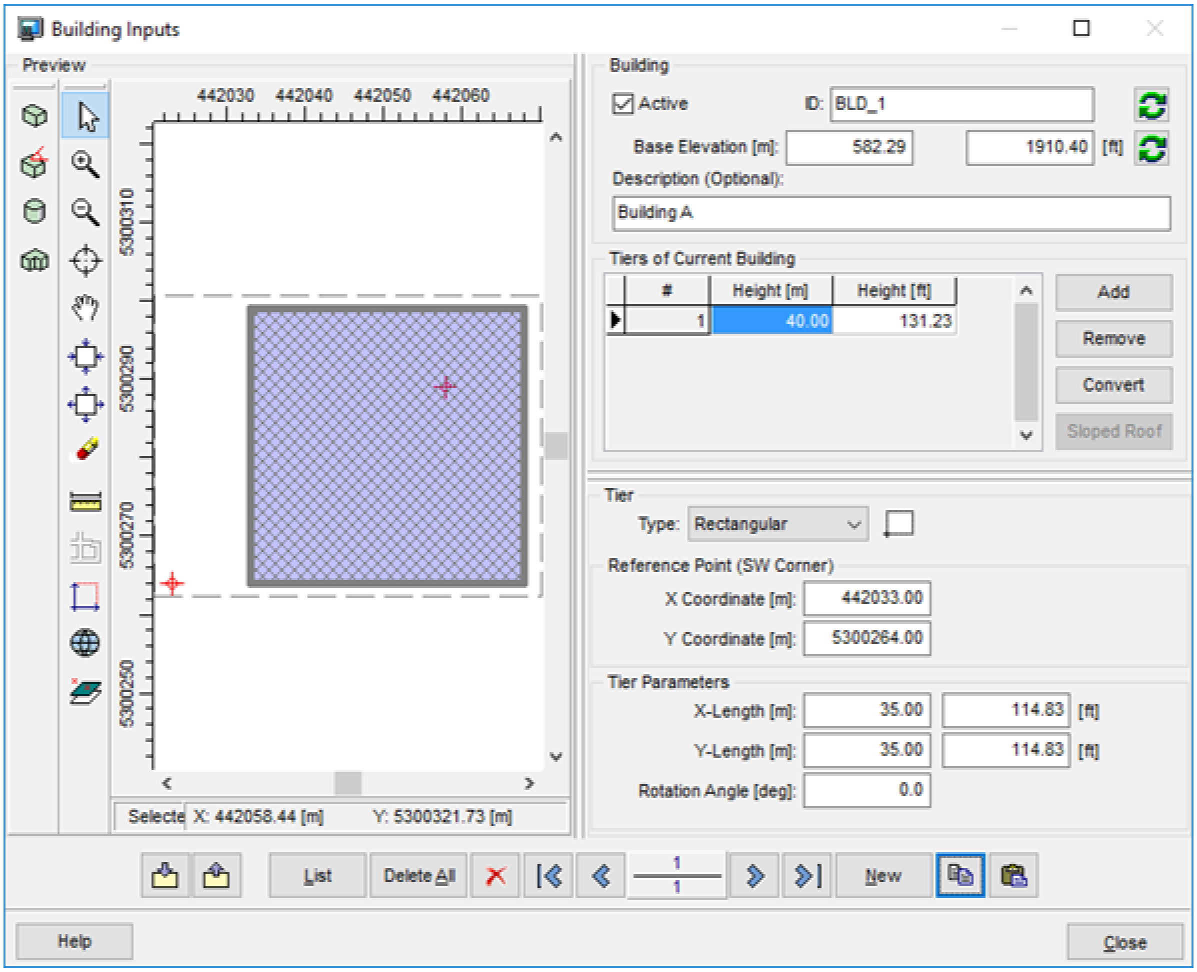
The Building Inputs dialog allows you the option of exporting buildings as an Excel spreadsheet with Lakes Format (*.xls) or a BPIP Input File (*.bpi). Simply click on the  button and specify the name and location of the file to export.

Record Navigator Buttons

At the bottom of the Building Inputs dialog are the Record Navigator buttons, which will help you manage information for more than one set of buildings defined.

Rectangular Building Inputs

The following parameters are requested in the **Building Inputs** dialog to define a rectangular building:



Building Inputs dialog - Rectangular Building Inputs

Building

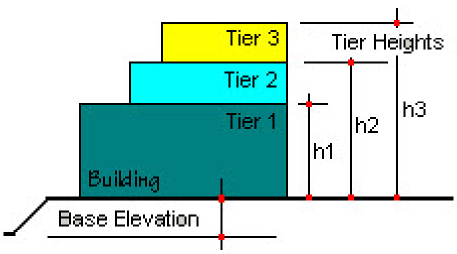
- **Active:** You have the option of changing your buildings' active status. By default this box will be checked, making your buildings active. If you do not want the building to be active, then you should uncheck this box. Inactive buildings are outlined on the drawing area in light gray and are not considered in the modeling analysis.

- **ID:** AERMOD View automatically places a default ID for the building, BLD_1, BLD_2, ..., or, BLD_n. You can change this default ID to any name you prefer. The name can have a maximum of 8 characters including spaces. If you prefer to use the default ID click on the button.

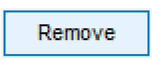
- **Base Elevation:** In this field you must input the building base elevation in meters [m] or in feet [ft]. You can click on the  button to display the default base elevation.
- **Description:** In this optional field you can provide a description of the building.

Tiers of Current Building

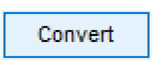
A tier is defined as the section of a building with a constant height. You must define at least one tier and specify the height of the tier in relation to the building base elevation in meters [m] or in feet [ft]. The tier height is defined as the distance from the base of the building to the top of the building tier. The following buttons are found in this panel:



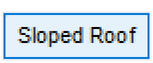
This button allows you to add a tier to your building.



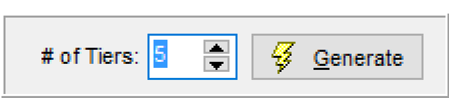
This button allows you to delete the currently selected tier. You will be prompted to confirm deletion.



This button can be used if you want to convert a tier into a new building or another building type.



The **Sloped Roof** button can be used to automatically generate tiers to create a sloped roof. Click this button to load the tool, which will allow you to specify the number of additional tiers you wish to create. Click **Generate** to create the tiers.



See Sloped Roof section for detailed instructions on how to create a sloped roof building and select examples of roof configurations.

Tier

- **Type:** This identifies the type of building being defined. If you would like to change the building type you can do so by selecting an alternate building type from the drop-down list.

Reference Point (SW Corner)

- **X Coordinate:** Here you must enter the X coordinate for the SW corner of the selected building tier in meters.
- **Y Coordinate:** Here you must enter the Y coordinate for the SW corner of the selected building tier in meters.

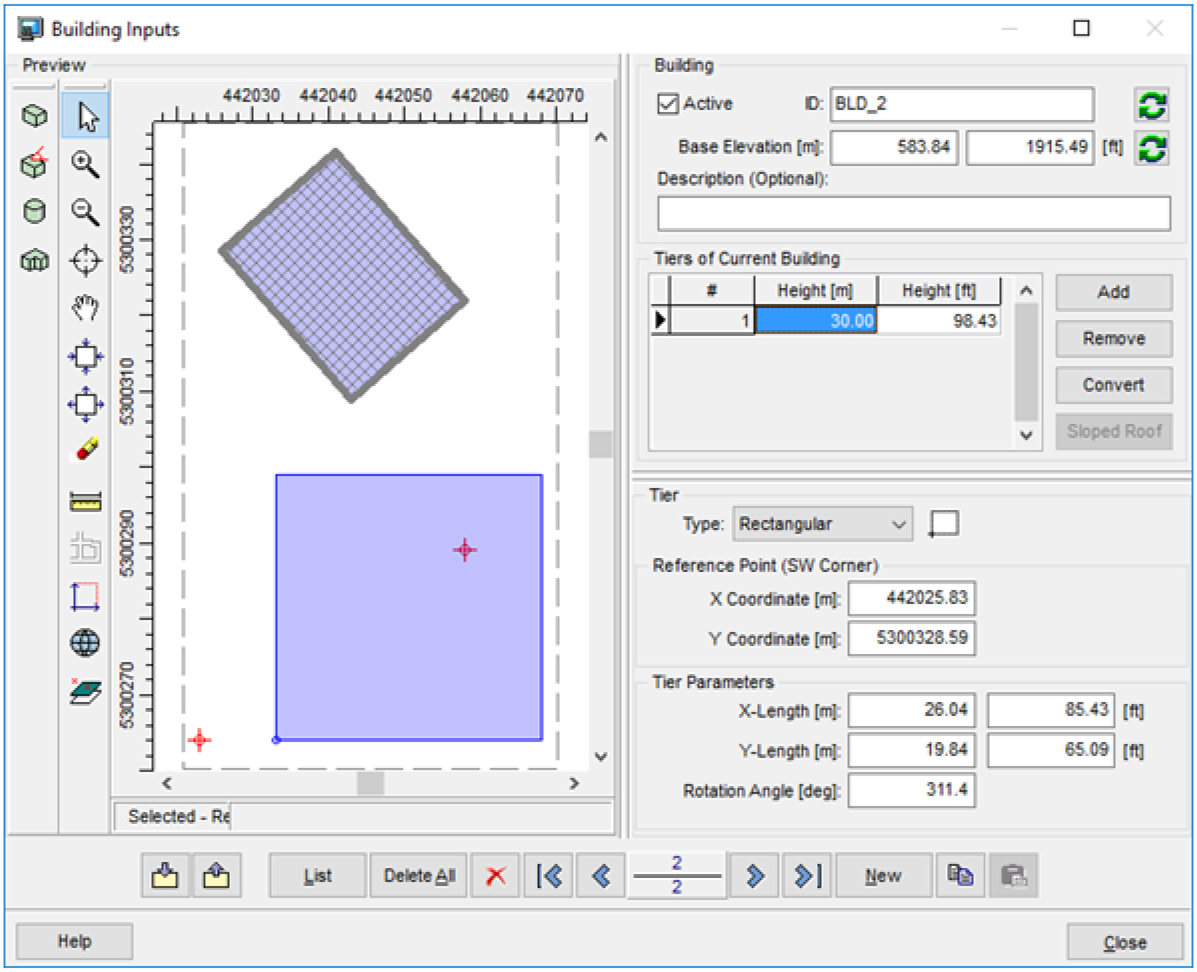
Tier Parameters

- **X-Length:** In this field you must enter the length of the selected building tier in the X direction in meters [m] or in feet [ft].
- **Y-Length:** In this field you must enter the length of the selected building tier in the Y direction in meters [m] or in feet [ft].
- **Rotation Angle:** If you wish to rotate your building, enter the rotation angle in degrees. The default is zero.

At the bottom of the dialog you will find the Record Navigator. Clicking the **List** button in the navigator will bring up the **Building List** dialog.

Angled Rectangular Building Inputs

The following parameters are requested in the **Building Inputs** dialog to define an angled rectangular building:



Building Inputs - Angled Rectangular Building Inputs

Building

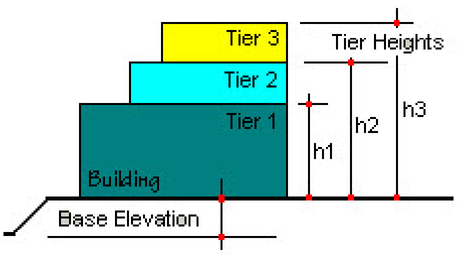
- **Active:** You have the option of changing your buildings' active status. By default this box will be checked, making your buildings active. If you do not want the building to be active, then you should uncheck this box. Inactive buildings are outlined on the drawing area in light gray and are not considered in the modeling analysis.

- **ID:** AERMOD View automatically places a default ID for the building, BLD_1, BLD_2, ..., or, BLD_n. You can change this default ID to any name you prefer. The name can have a maximum of 8 characters including spaces. If you prefer to use the default ID click on the  button.

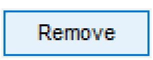
- **Base Elevation:** In this field you must input the building base elevation in meters [m] or in feet [ft]. You can click on the  button to display the default base elevation.
- **Description:** In this optional field you can provide a description of the building.

Tiers of Current Building

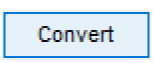
A tier is defined as the section of a building with a constant height. You must define at least one tier and specify the height of the tier in relation to the building base elevation in meters [m] or in feet [ft]. The tier height is defined as the distance from the base of the building to the top of the building tier. The following buttons are found in this panel:



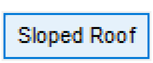
This button allows you to add a tier to your building.



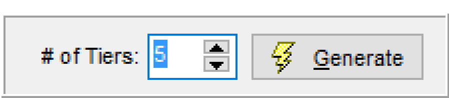
This button allows you to delete the currently selected tier. You will be prompted to confirm deletion.



This button can be used if you want to convert a tier into a new building or another building type.



The **Sloped Roof** button can be used to automatically generate tiers to create a sloped roof. Click this button to load the tool, which will allow you to specify the number of additional tiers you wish to create. Click **Generate** to create the tiers.



See Sloped Roof section for detailed instructions on how to create a sloped roof building and select examples of roof configurations.

Tier

- **Type:** This identifies the type of building being defined. If you would like to change the building type you can do so by selecting an alternate building type from the drop-down list.

Reference Point (SW Corner)

- **X Coordinate:** Here you must enter the X coordinate for the SW corner of the selected building tier in meters.

- **Y Coordinate:** Here you must enter the Y coordinate for the SW corner of the selected building tier in meters.

Tier Parameters

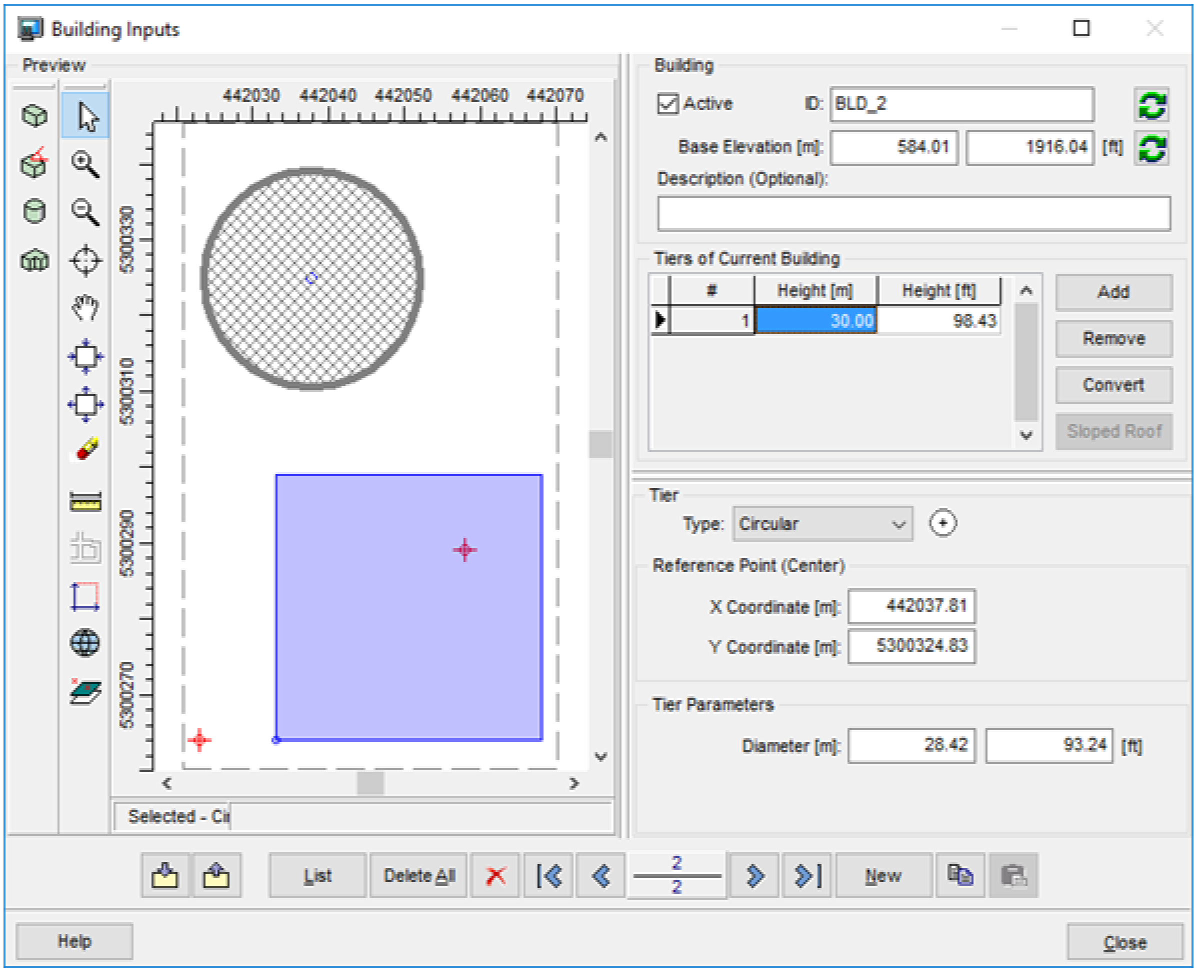
- **X-Length:** In this field you must enter the length of the selected building tier in the X direction in meters [m] or in feet [ft].

- **Y-Length:** In this field you must enter the length of the selected building tier in the Y direction in meters [m] or in feet [ft].

- **Rotation Angle:** If you wish to rotate your building, enter the rotation angle in degrees. The default is zero for a rectangular building and for an angled rectangular building it is as you draw.

Circular Building Inputs

The following parameters are requested in the **Building Inputs** dialog to define a circular building:



Building Inputs dialog - Circular Building Inputs

Building

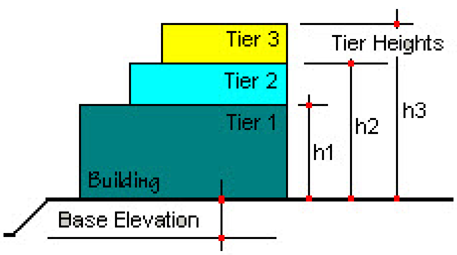
- **Active:** You have the option of changing your buildings' active status. By default this box will be checked, making your buildings active. If you do not want the building to be active, then you should uncheck this box. Inactive buildings are outlined on the drawing area in light gray and are not considered in the modeling analysis.

- **ID:** AERMOD View automatically places a default ID for the building, BLD_1, BLD_2, ..., or, BLD_n. You can change this default ID to any name you prefer. The name can have a maximum of 8 characters including spaces. If you prefer to use the default ID click on the  button.

- **Base Elevation:** In this field you must input the building base elevation in meters [m] or in feet [ft]. You can click on the  button to display the default base elevation.
- **Description:** In this optional field you can provide a description of the building.

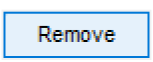
Tiers of Current Building

A tier is defined as the section of a building with a constant height. You must specify the height of the tier in relation to the building base elevation in meters [m] or in feet [ft]. The tier height is defined as the distance from the base of the building to the top of the building tier.

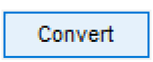


The following buttons are found in this panel:

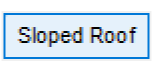
This button allows you to add a tier to your building.



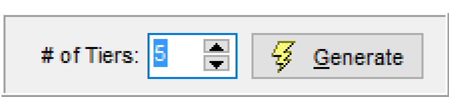
This button allows you to delete the currently selected tier. You will be prompted to confirm deletion.



This button can be used if you want to convert a tier into a new building or another building type.



The **Sloped Roof** button can be used to automatically generate tiers to create a sloped roof. Click this button to load the tool, which will allow you to specify the number of additional tiers you wish to create. Click **Generate** to create the tiers.



See Sloped Roof section for detailed instructions on how to create a sloped roof building and select examples of roof configurations.

Tier

- **Type:** This identifies the type of building being defined. If you would like to change the building type you can do so by selecting an alternate building type from the drop down list.

Reference Point (Center)

- **X Coordinate:** Here you must enter the X coordinate for the center of the circular building. If you have defined your building graphically then this value will be automatically displayed here. You may adjust this coordinate at any time.

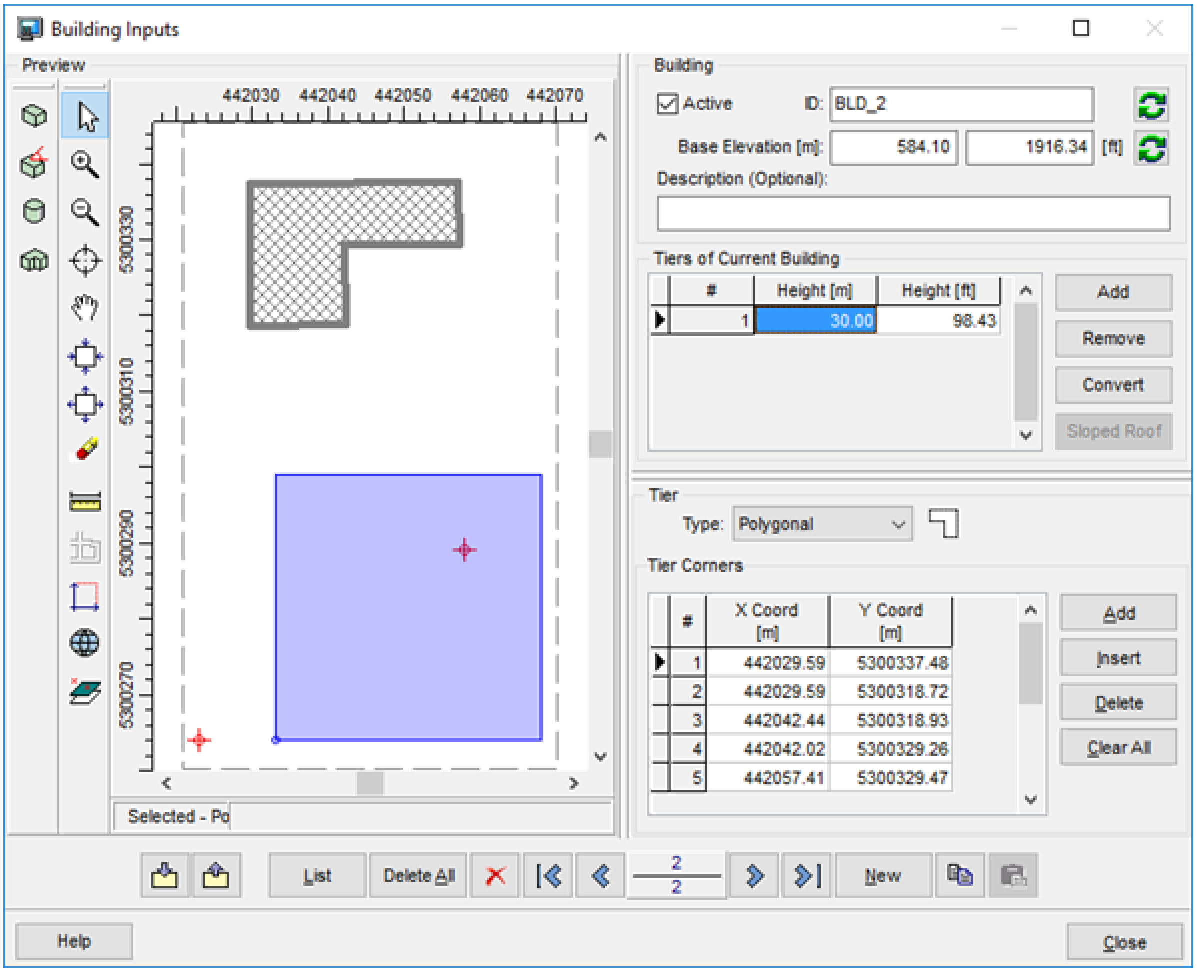
- **Y Coordinate:** Here you must enter the Y coordinate for the center of the circular building. If you have defined your building graphically then this value will be automatically displayed here. You may adjust this coordinate at any time.

Tier Parameters

- **Diameter:** In this field you must enter the diameter of the building in meters [m] or in feet [ft]. If you have defined your building graphically, then this value will be automatically displayed here. You may adjust the diameter at any time.

Polygonal Building Inputs

The following parameters are requested in the **Building Inputs** dialog to define a polygonal building:



Building Inputs dialog - Polygonal Building Inputs

Building

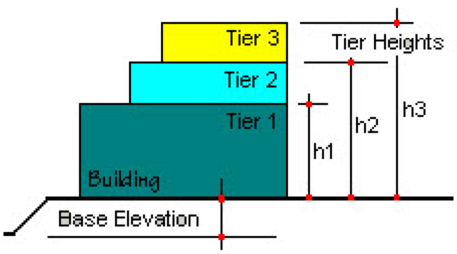
- **Active:** You have the option of changing your buildings' active status. By default this box will be checked, making your buildings active. If you do not want the building to be active, then you should uncheck this box. Inactive buildings are outlined on the drawing area in light gray and are not considered in the modeling analysis.

- **ID:** AERMOD View automatically places a default ID for the building, BLD_1, BLD_2, ..., or, BLD_n. You can change this default ID to any name you prefer. The name can have a maximum of 8 characters including spaces. If you prefer to use the default ID click on the  button.

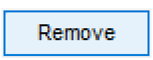
- **Base Elevation:** In this field you must input the building base elevation in meters [m] or in feet [ft]. You can click on the  button to display the default base elevation.
- **Description:** In this optional field you can provide a description of the building.

Tiers of Current Building

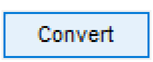
A tier is defined as the section of a building with a constant height. You must define at least one tier and specify the height of the tier in relation to the building base elevation in meters [m] or in feet [ft]. The tier height is defined as the distance from the base of the building to the top of the building tier. The following buttons are found in this panel:



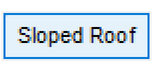
This button allows you to add a tier to your building.



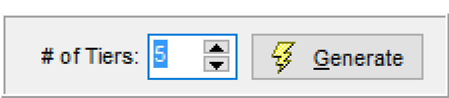
This button allows you to delete the currently selected tier. You will be prompted to confirm deletion.



This button can be used if you want to convert a tier into a new building or another building type.



The **Sloped Roof** button can be used to automatically generate tiers to create a sloped roof. Click this button to load the tool, which will allow you to specify the number of additional tiers you wish to create. Click **Generate** to create the tiers.



See Sloped Roof section for detailed instructions on how to create a sloped roof building and select examples of roof configurations.

Tier

- **Type:** This identifies the type of building being defined. If you would like to change the building type you can do so by selecting an alternate building type from the drop down list.

Tier Corners

- **X Coordinate:** In this table you must enter the X coordinate in meters for each corner of the building. If you have defined your building graphically then this value will be automatically displayed here. You may adjust the coordinates at any time.

- **Y Coordinate:** In this table you must enter the Y coordinate in meters for each corner of the building. If you have defined your building graphically then this value will be automatically displayed here. You may adjust the coordinates at any time.

The following functions are available to manipulate the tier corners:

Add This button allows you to create another row to add coordinates for a new corner.

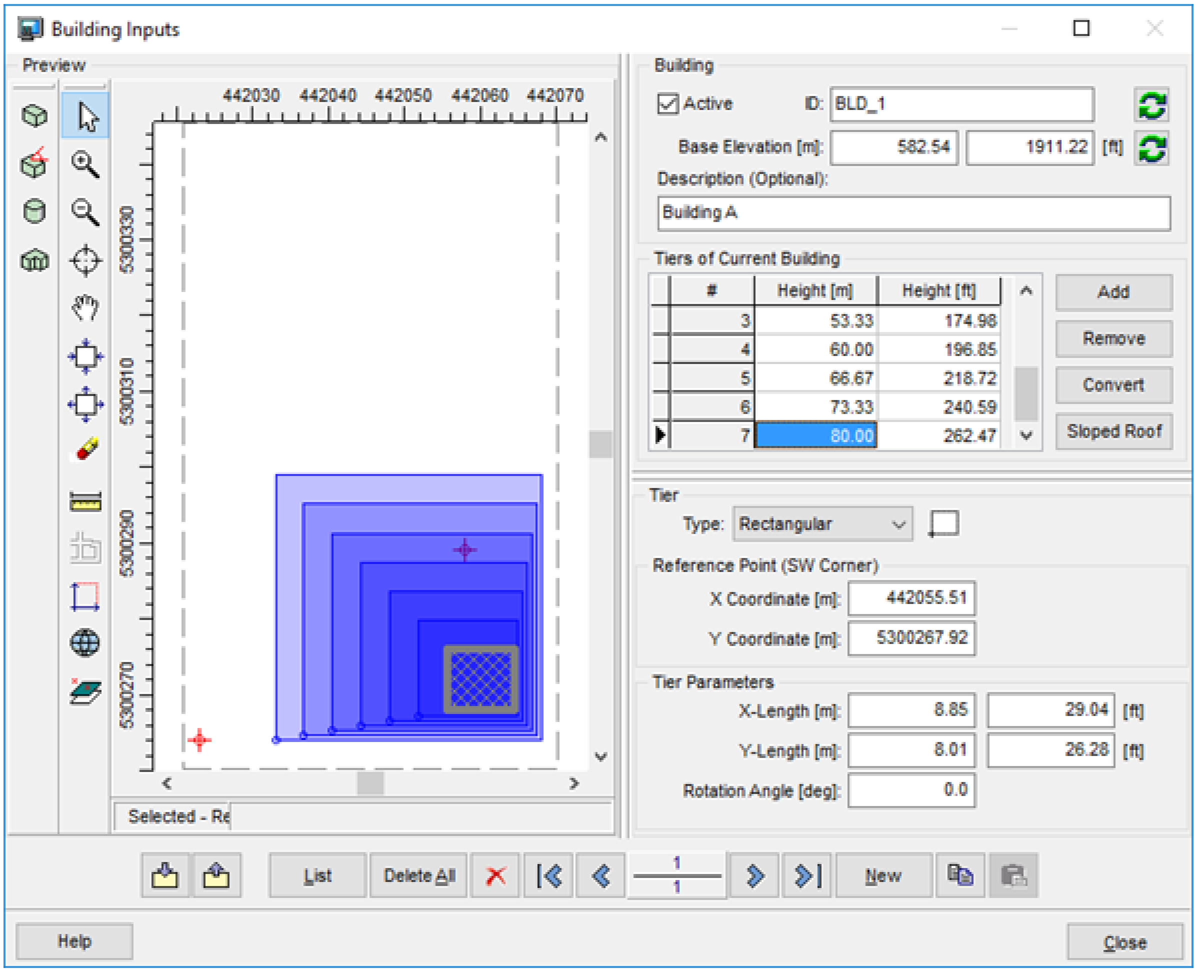
Insert This button allows you to insert a row above the one currently selected to add coordinates for a new corner.

Delete This button allows you to delete a row of coordinates.

Clear All This button allows you to delete all coordinate rows.

Sloped Roof

The **Slope Roof** feature in **Building Inputs** allows you to automatically create multiple tiers based on the defined top and bottom tier, to create an appearance of a sloped roof.

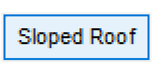


Building Inputs - Rectangular Building Sloped Roof

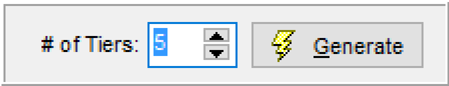
How to Define Slope Roof

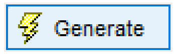
1. Completely define the bottom tier of your building.

2. Completely define the top tier of your building. If your building comes to a point, define the smallest possible tier or 0.1 x 0.1 m.

3. Click 

button. The following tool will load:



4. Type in the **# of Tiers** and click the  button.

The tiers will be created equally distributed along the range of dimensions specified.

> The sloped roof tiers will only be created if the bottom and top tiers are completely defined.

> If AERMOD View is unable to create sloped roof tiers for a complex polygonal building, break the polygon into subsections to simplify the tiers.

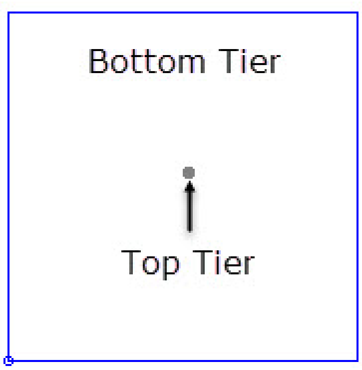
Sloped Roof Building Examples

Below are some examples of buildings you can create using the **Sloped Roof** feature.

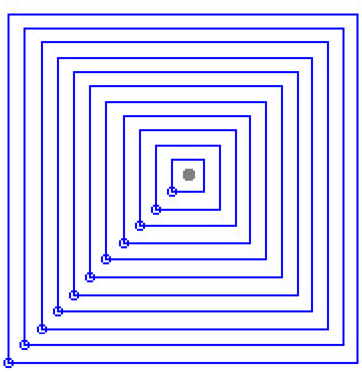
Click on an entry to expand the description.

Basic Tapered Building

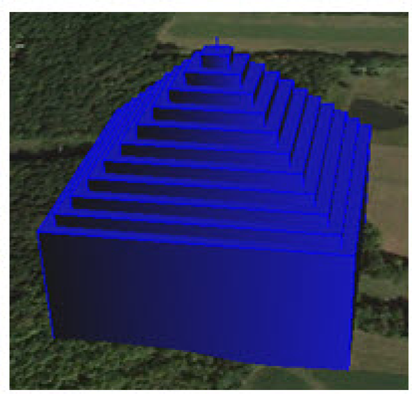
To create a simple tapered building, follow the instructions above.



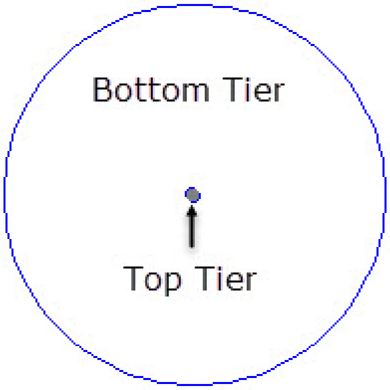
User Defined Tiers



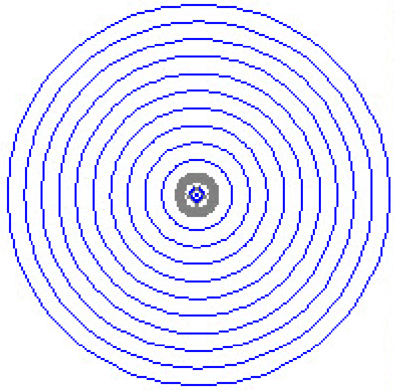
Generated Tiers

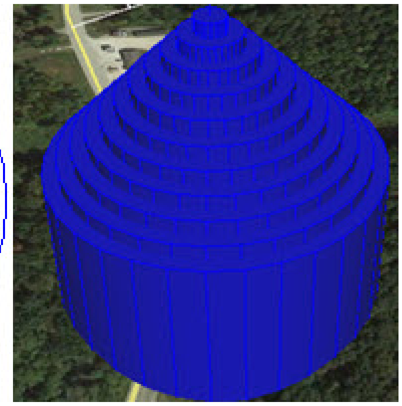


3D View



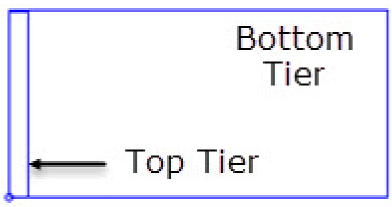
User Defined Tiers


Generated Tiers

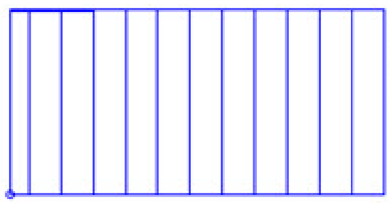

3D View

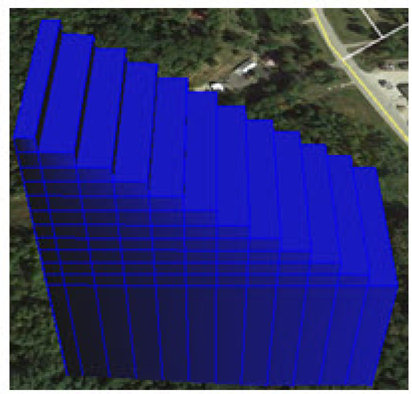
Sloped and Double-Sloped Roof

You can create a sloped roof by creating a long thin tier along one side of the building.

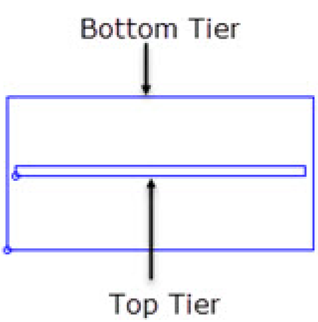


User Defined Tiers

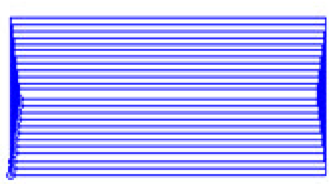

Generated Tiers

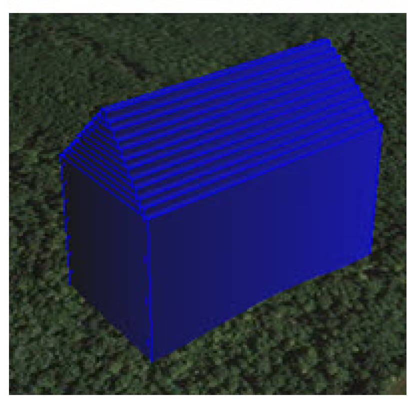

3D View

You can create a double-sloped roof by putting a long thin tier in the middle of the building.



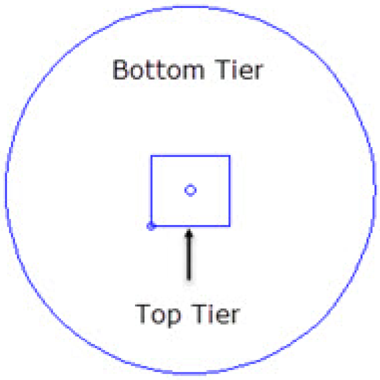
User Defined Tiers


Generated Tiers

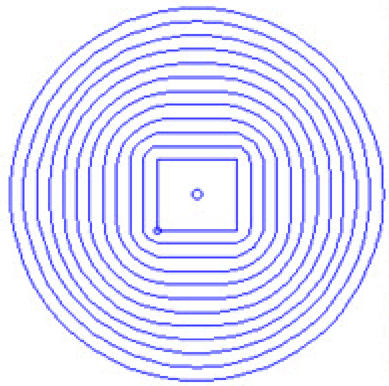

3D View

Mixed Tier Shape Building

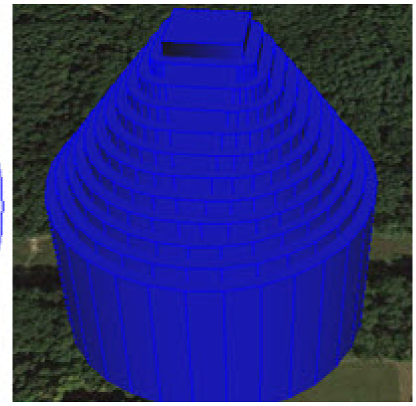
If the bottom and top tiers are not the same shape, you can still create a sloped roof.



User Defined Tiers



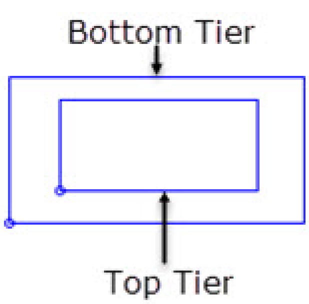
Generated Tiers



3D View

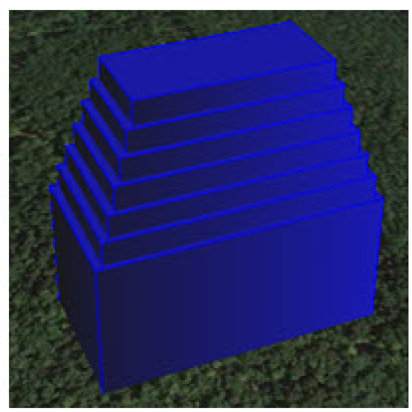
Varied Slope Roof Building

You can generate a building with a varied slope roof by creating it in multiple stages. The utility will always generate the tiers between the top tier and the tier immediately under it. Therfore you must create the multi-sloped roof from bottom up.
First, create the bottom portion of the roof:



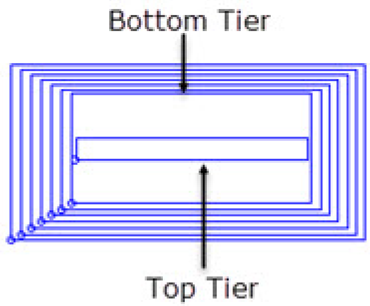
User Defined Tiers

Generated Tiers

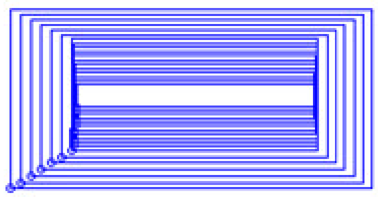


3D View

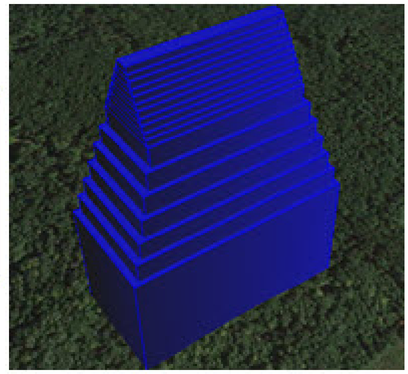
Then, create the next section of sloped roof:



User Defined Tiers



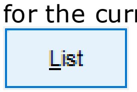
Generated Tiers



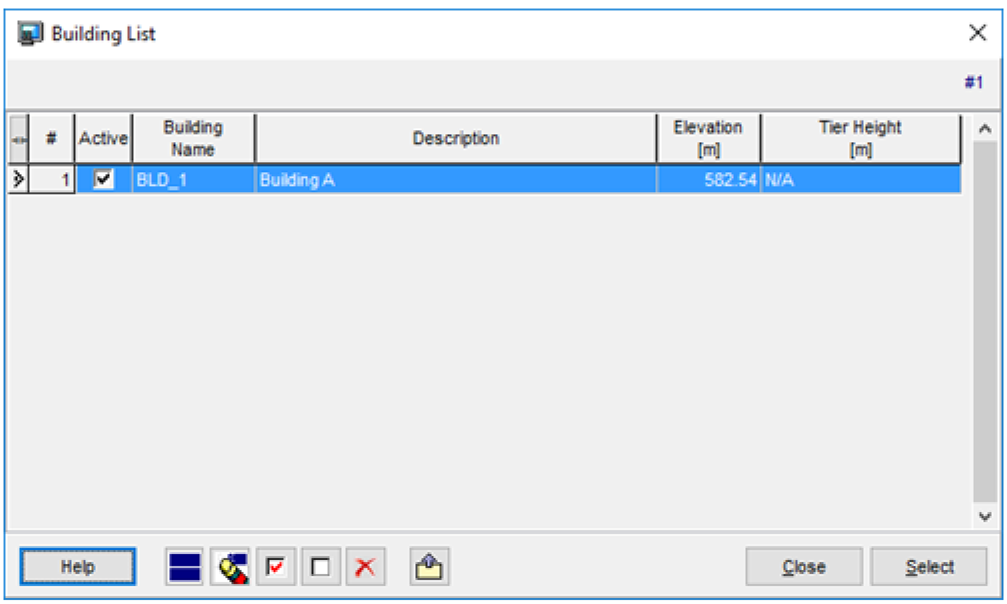
3D View

Building List

The **Building List** dialog contains a list of all the buildings already defined in the Building Inputs dialog for the current run in the order they were created. This dialog is displayed by clicking on the



button of the Record Navigator located in the **Building Inputs** dialog.

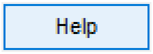


Building List dialog

You have the option of changing your buildings' active status through the Active column. By default all boxes will be checked, making your buildings active. To make a building inactive, uncheck this box.

Inactive buildings are outlined on the drawing area in light gray and are not considered in the modeling analysis.

A number of buttons are found at the bottom of this dialog. Their functions are described below:

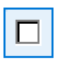


Opens AERMOD View Help for the Building List dialog.

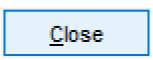

Selects all buildings for the list being displayed.


Unselects all selected buildings in the list.

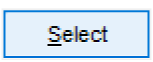

Checks the **Active** box for all selected buildings in the list.


Unchecks the **Active** box for all selected buildings in the list.


Deletes selected buildings from the list.



Closes the Building List dialog.



Opens the Building Inputs dialog for the selected building.

Importing Buildings from DXF

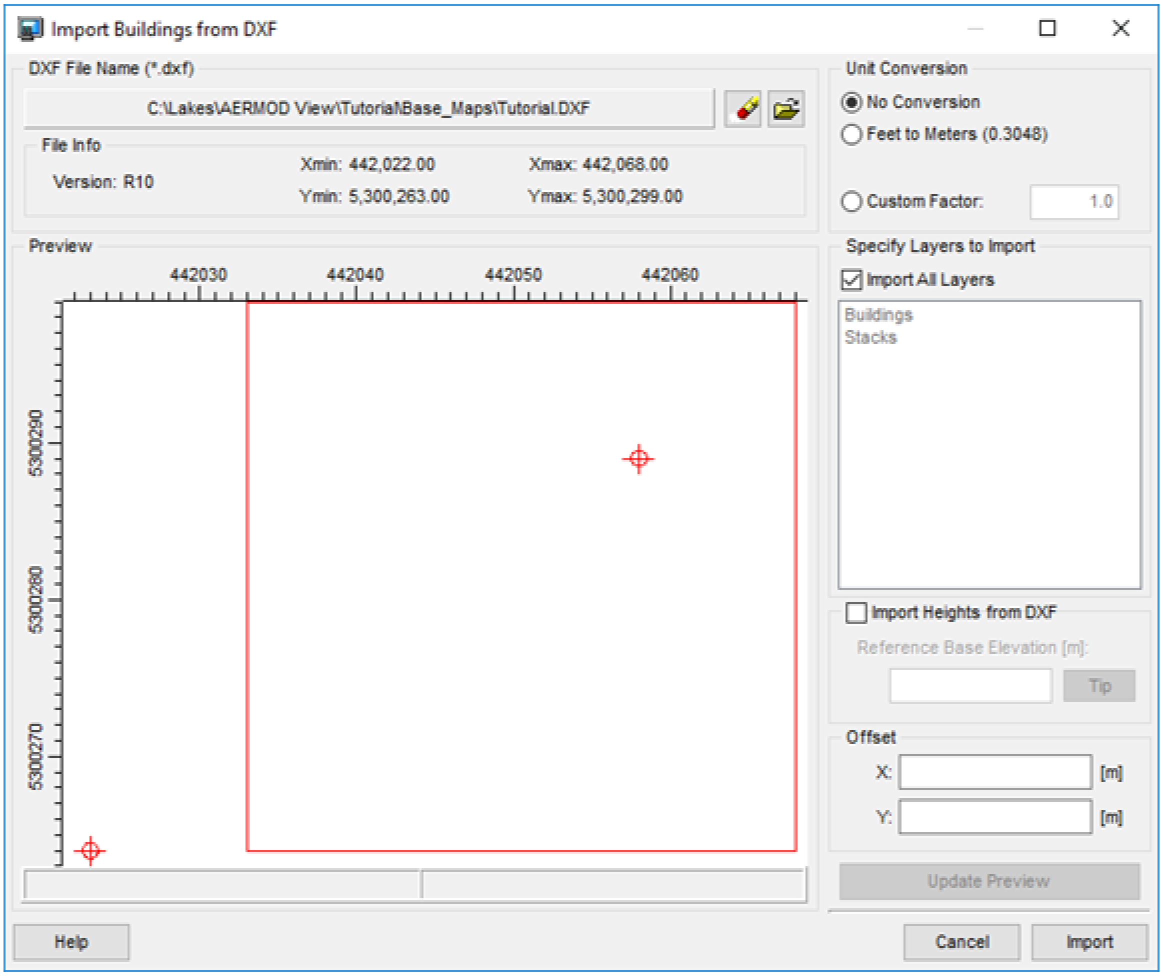
To minimize the time spent to draw buildings, AERMOD View offers the option of importing building locations and dimensions from an AutoCAD DXF file.

> The **Import Buildings from DXF** utility can only read DXF objects that are consistent with DXF Release 12 format or earlier. If your AutoCAD DXF file was created in a more recent release than R12, please re-save it as an R12 file before you try importing the file.

How to Import Buildings from an AutoCAD DXF file:

1. Select **Import | Buildings from DXF...** from the menu to display the **Import Buildings from DXF** dialog. Click on the  button to specify the name and location of the DXF file containing the

buildings to be imported. The name and path of the imported DXF file will be displayed in the **DXF File Name (*.dxf)** panel. The DXF map is displayed in the **Preview** panel.



Import Buildings from DXF dialog

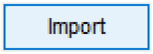
2. The **Unit Conversion to Meter** panel provides you with the option of converting the existing units of the DXF file to meters in case your DXF map was created in units other than meters. For the unit conversion, three options are available:

   - **No Conversion:** This is the default option used and the DXF will be imported without any conversions.
   - **Feet To Meters:** Will scale the DXF file by a factor of 0.3048, the equivalent conversion of feet to meters.
   - **Custom:** Allows you to specify a custom conversion factor to be applied to the DXF drawing.

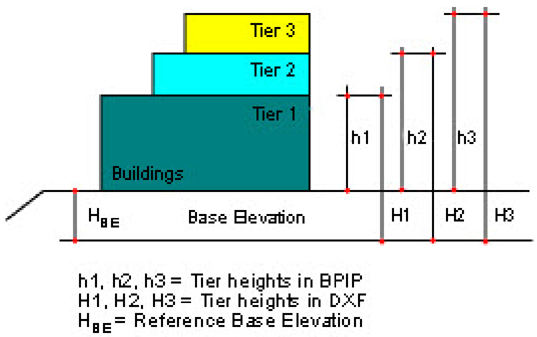
3. In the **Specify Layers to Import** section you can either **Import All Layers** by checking this box or uncheck it and select layers from the list. You can see any specific layer by selecting it in the **Specify Layers to Import** list and clicking the **Update Preview** button.

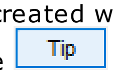
4. If your DXF file contains real world elevation values (z coordinate), this information can be imported as well. Check the **Import Heights from DXF** box and specify the **Base Elevation** for the building.

5. Use the **Offset** feature to shift your DXF in the X and Y direction, for example, to convert building coordinates from Local Cartesian coordinate system to UTM.

6. Click the 

button to import the structure(s) and close the dialog. All the imported buildings are automatically displayed in the drawing area.



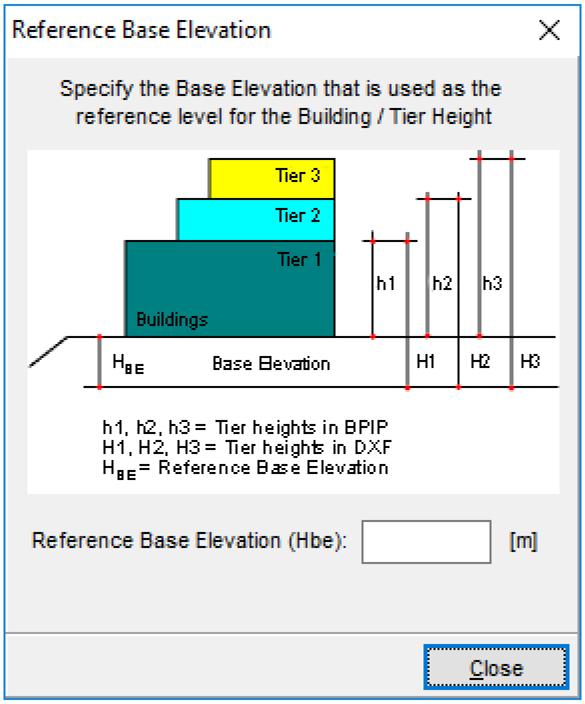
In creating your DXF, a reference base elevation may have been included (ie: elevation above MSL). To subtract this value from your imported tier heights, enter the reference base elevation on which your DXF was created. In the case your DXF was created without a reference base elevation, you can enter a value of zero. You can also click on the 

button (only available if you checked the **Import Heights from DXF** box) to display the Reference Base Elevation dialog where you can specify your reference base elevation.

> Lines or polygons with contingent end points are recognized from DXF files to be imported into projects when building data is imported from a DXF

Reference Base Elevation

You have access to the **Reference Base Elevation** dialog by clicking on the Tip button in the [Import Buildings from DXF](). In this dialog you can specify your reference base elevation.



Reference Base Elevation dialog

In creating your DXF, a reference base elevation may have been included (ie: elevation above MSL). To subtract this value from your imported tier heights, enter the reference base elevation, in meters, on which your DXF was created. In the case your DXF was created without a reference base elevation, you can enter a value of zero.

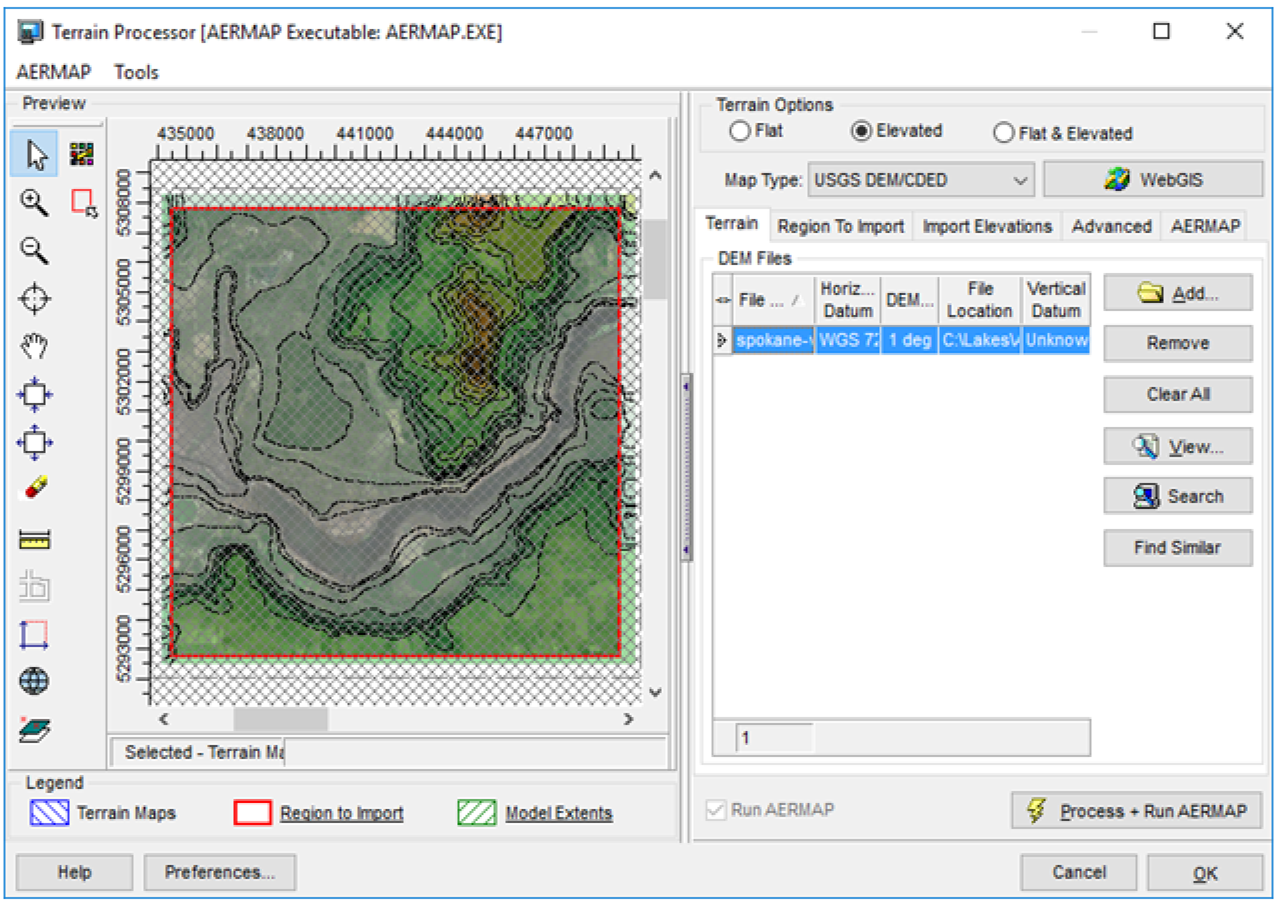
Terrain Processor

The **Terrain Processor** allows you to import and process terrain elevation data for your project. You have access to the **Terrain Processor** by selecting **Data | Terrain Processor...** from the menu or by clicking on the **Terrain** 

menu toolbar button.

AERMOD View imports terrain elevations from various digital elevation file formats for three main purposes:

- To extract terrain elevations for receptors and base elevations for sources and buildings.
- To generate the Terrain Grid (TG) file to be used in the TG Pathway (ISC only).
- To generate visualization of the surrounding terrain in 2D and 3D.



Terrain Processor

How to Use the Terrain Processor - Overview

1. In the **Terrain Options** panel, determine whether the terrain will be **Flat**, **Elevated** or **Flat & Elevated** by clicking the appropriate radio button.

> If you are running a project with flat terrain, you will not have to configure any further settings.

2. Select a **Map Type** from the drop down list. You have the following options:

   - **USGS DEM/CDED** - Digital elevation model files from the United States Geological Survey (US only) or Canadian Digital Elevation Data files (About DEM files)
   - **NED GEOTIFF** - National Elevation Dataset terrain data files from the United States Geological Survey. (About NED GEOTIFF files)
   - **SRTM1/SRTM3** - Shuttle Radar Topography Mission terrain data files (About SRTM files)
   - **GTOPO30/SRTM30** - Global digital elevation model files (About GTOPO30/SRTM30 files)
   - **XYZ File** - Comma delimited text file containing xyz coordinates (About XYZ files)
   - **AUTOCAD DXF** - AutoCAD Drawing Interchange Format file containing terrain elevations (About AUTOCAD DXF files)
   - **UK DTM** - Digital terrain model files produced by the United Kingdom Standard Ordnance Survey (About UK DTM files)
   - **UK NTF** - National transfer format files produced by the United Kingdom Standard Ordnance Survey (About UK NTF files)

3. Import the appropriate terrain data files in the Terrain tab.
4. Set the extent of the terrain files that should be used when converting the terrain data file to a GridASCII Terrain File in the Region to Import tab.
5. Assign the terrain elevations to the receptors, sources and buildings in your project in the Import Elevations tab.
6. Set advanced options for assigning and importing the terrain elevations in the Advanced tab.
7. Click the **Process + Run AERMAP** button to process the data. Click OK to close the Terrain Processor.

Interface Description

The menu bar at the top of the **Terrain Processor** dialog allows you to access various options. The **AERMAP** menu item allows you to easily view input and output files generated by the AERMAP runs. The **Tools** menu option lets you browse and edit files.

On the left side of the **Terrain Processor** dialog is the **Preview** panel, which displays a preview of the terrain elevation file boundaries (blue hatched area) you are importing in relation to the project site domain (green hatched region). This panel contains many annotation tools which allow you manipulate what is visible in the preview area, zoom in and pan on locations of interest. These tools are also found on the Annotation Toolbar in the main interface. Also available is the **Select Region to Import** tool ( ) which allows you to graphically define the region to import in the preview area. At the bottom of the **Preview** panel is the Legend which allows you to easily identify the **Terrain Maps**, **Region to Import** and **Model Extents**. If you click on either the Region to Import or the Model Extents a text file will open with information regarding the region or model extents.

On the right side of the **Terrain Processor** dialog you may specify whether the terrain in your modeling area is flat or elevated. There are four tabs where you can specify all the information to import and process the terrain elevation data. See the following sections for a description of each tab and their options:

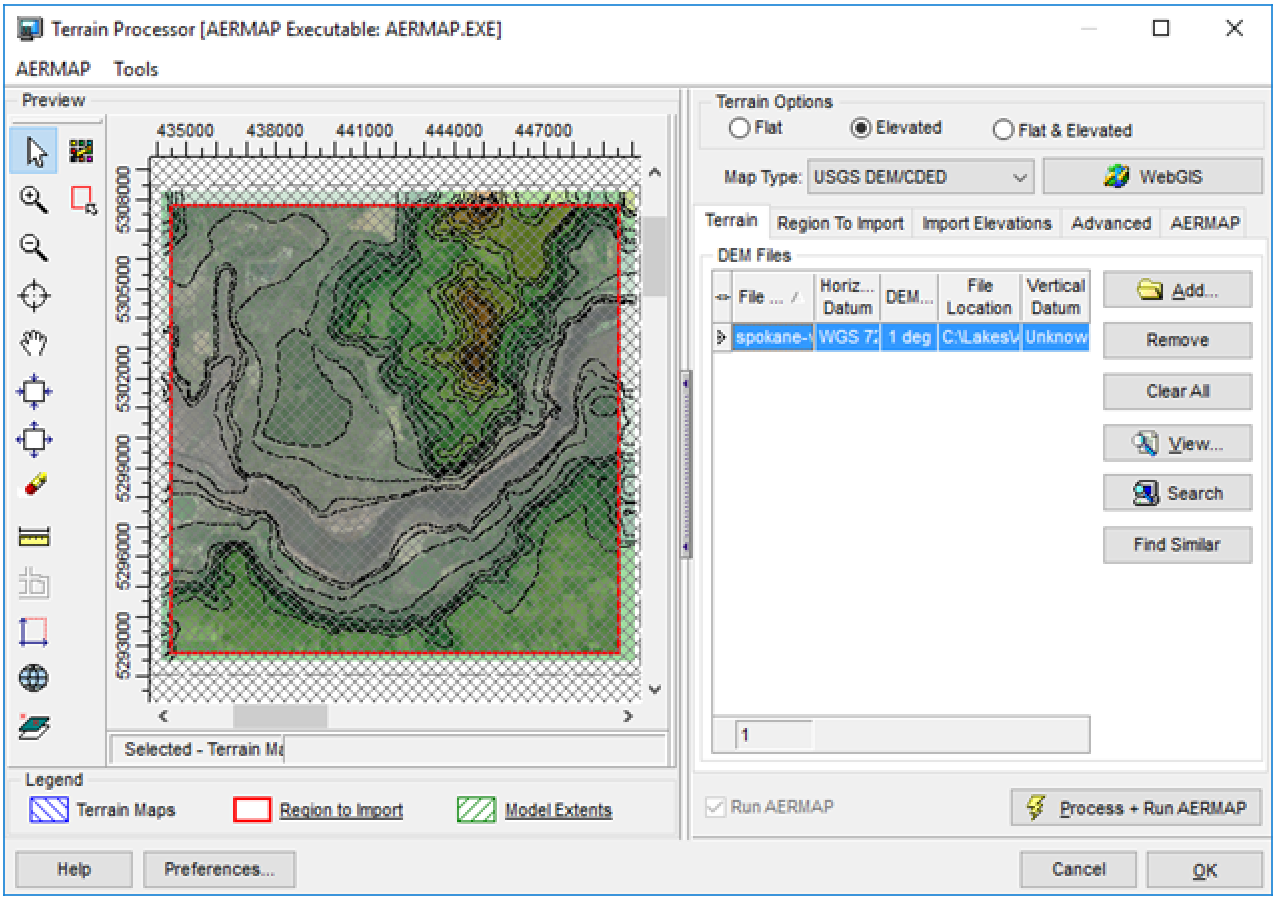
- Terrain
- Region to Import
- Import Elevations
- Advanced Options
- AERMAP

The following utilities are also found in the Terrain Processor:

Preferences... Click this button to open the AERMAP screen of the Preferences dialog.

Specify Terrain Files

The **Terrain** tab of the Terrain Processor dialog allows you to specify the terrain elevation files to be used for your project.



Terrain Processor - Terrain tab

How to Specify Terrain Files

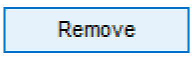
1. Select a **Map Type** from the drop down list. You have the following options:

   - **USGS DEM/CDED** - Digital elevation model files from the United States Geological Survey (US only) or Canadian Digital Elevation Data files (About DEM files)
   - **NED GeoTIFF** - National Elevation Dataset terrain data files from the United States Geological Survey. (About NED GeoTIFF files)
   - **SRTM1/SRTM3** - Shuttle Radar Topography Mission terrain data files (About SRTM files)

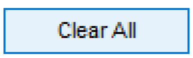
- **GTOPO30/SRTM30** - Global digital elevation model files (About GTOPO30/SRTM30 files)
- **XYZ File** - Comma delimited text file containing xyz coordinates (About XYZ files)
- **AUTOCAD DXF** - AutoCAD Drawing Interchange Format file containing terrain elevations (About AUTOCAD DXF files)
- **UK DTM** - Digital terrain model files produced by the United Kingdom Standard Ordnance Survey (About UK DTM files)
- **UK NTF** - National transfer format files produced by the United Kingdom Standard Ordnance Survey (About UK NTF files)

2. Specify the digital terrain files. The options available depend on which type of digital elevation file you will be importing:

You can add a new elevation file.



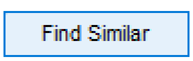
You can remove an existing elevation file.



You can remove all the elevation files.

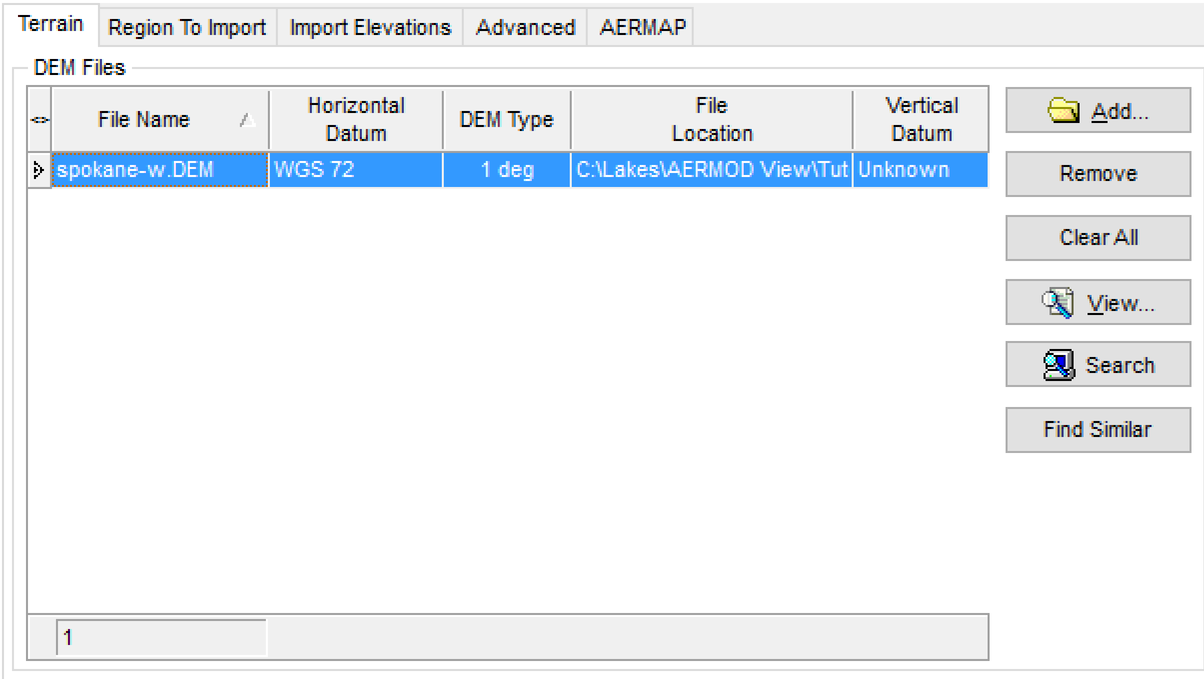
View the elevation file in WordPad.

Search for the map type based on the selected elevation file.



Search for a similar elevation file.

Specifying DEM Files



If you have not set the Map Projection information for the project, click the **Map Projection** button to set it before downloading files from WebGIS.

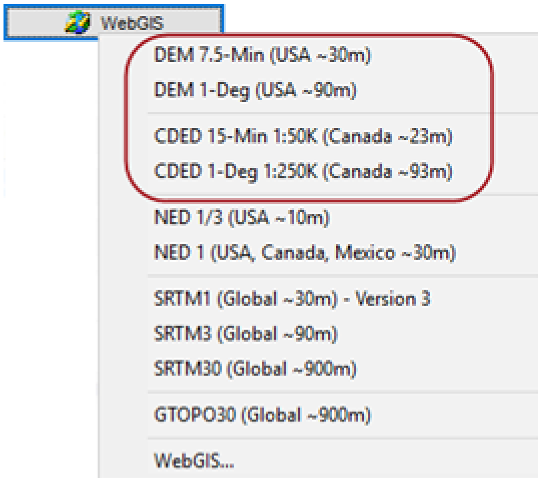
As of the AERMAP revision in version 09040, AERMAP can now process DEM files of mixed format (e.g. 7.5-minute and 1-degree DEM files) in the same run.

1. If you already have a USGS or CDED DEM file, click the 

button and specify that file. You can select multiple files.

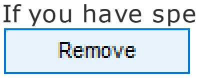
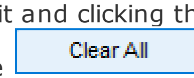
2. If you do not already have the appropriate DEM file(s), click the 

button. A sub-menu will display. Choose either: **DEM 7.5-min (USA ~30m), DEM 1-deg (USA ~90m), CDED 15-min 1:50 K (Canada ~23 m), CDED 1-Deg 1:250 K (Canada ~93 m)**. The appropriate files will be automatically downloaded from the WebGIS website.



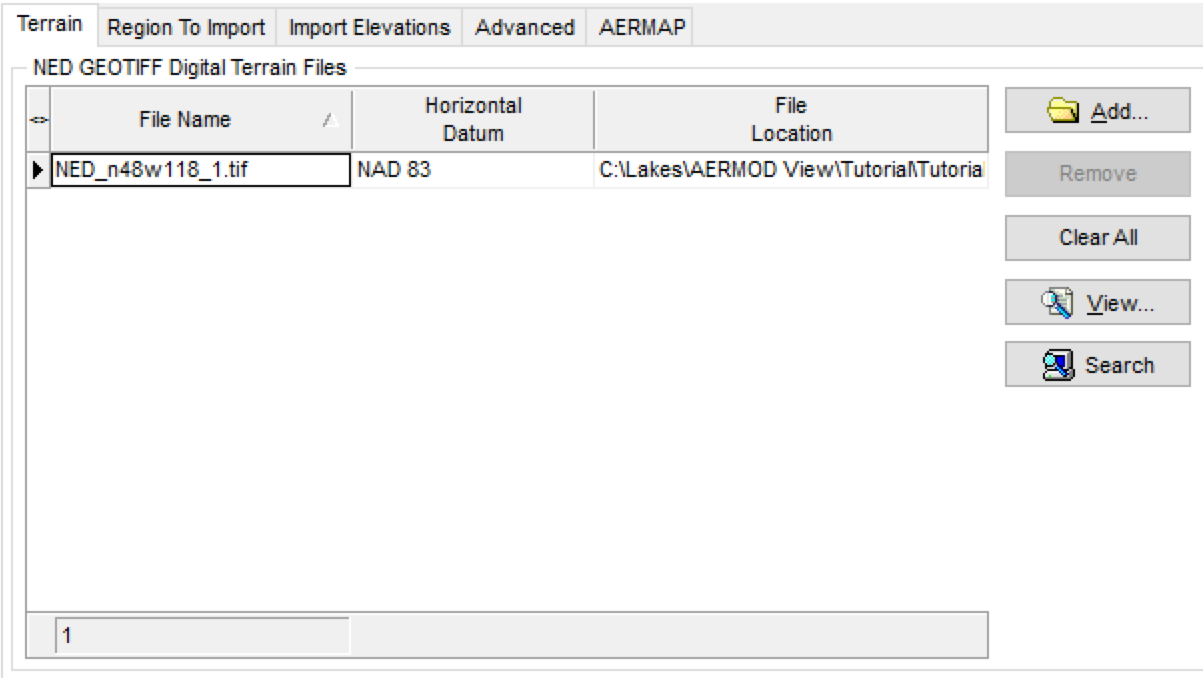
WebGIS download menu

3. Once all the files are specified, they appear in the list. Confirm that the data is correct by clicking the  button to read the file(s) in WordPad.

4. If you have specified a file in error, you can remove it by selecting it and clicking the  button. To remove all files and start again, click the  button.

5. To locate all files of the specified type on your computer, click the  button. A sub-menu will display. You can elect to search the cache folder (see Preferences - Download Settings) or specify a folder/all its sub-folders.

Specifying NED GEOTIFF Files

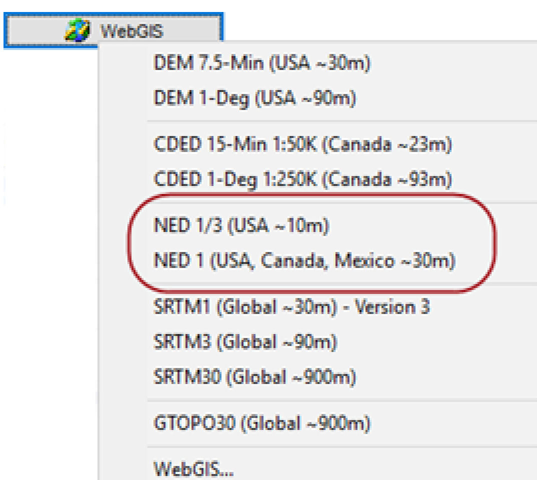


1. If you already have a NED GEOTIFF file, click the 

button and specify that file. You can select multiple files.

2. If you do not already have the appropriate NED files, click the 

button and select either **NED 1/3 (USA ~10m)** or **NED (USA, Canada, Mexico ~30m)** options.



WebGIS download menu

> **Warning!** NED files can take a while to download, depending on the size of the domain and the resolution.

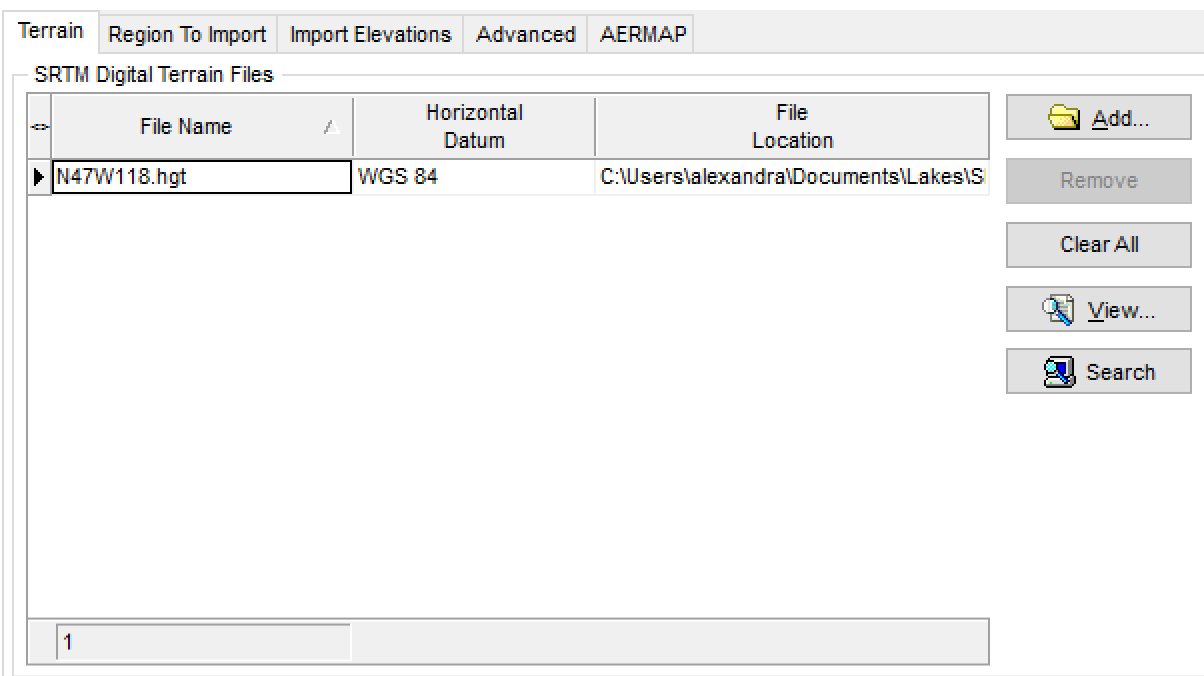
> The direct download of **NED** files through the WebGIS is **ONLY** available if you have current maintenance. If you do not have maintenance, you will need to download the files from the National Map Viewer and Download Platform.

3. Once all the files are specified, they appear in the list. Confirm that the data is correct by clicking the View... button to read the file(s) in WordPad.

4. If you have specified a file in error, you can remove it by selecting it and clicking the Remove button. To remove all files and start again, click the Clear All button.

5. To locate all files of the specified type on your computer, click the Search button. A sub-menu will display. You can elect to search the cache folder (see Preferences - Download Settings) or specify a folder/all its sub-folders.

Specifying SRTM Files



If you have not set the Map Projection information for the project, click the Map Projection button to set it before downloading files from WebGIS.

1. If you already have an SRTM file, click the Add... button and specify that file. You can select multiple files.

2. If you do not already have the appropriate SRTM files, click the WebGIS button. A sub-menu will display. Choose either **SRTM1 (Global ~30m - Version 3)** or **SRTM3 (Global ~90m)**. The appropriate files will be automatically downloaded from the WebGIS website.

WebGIS download menu

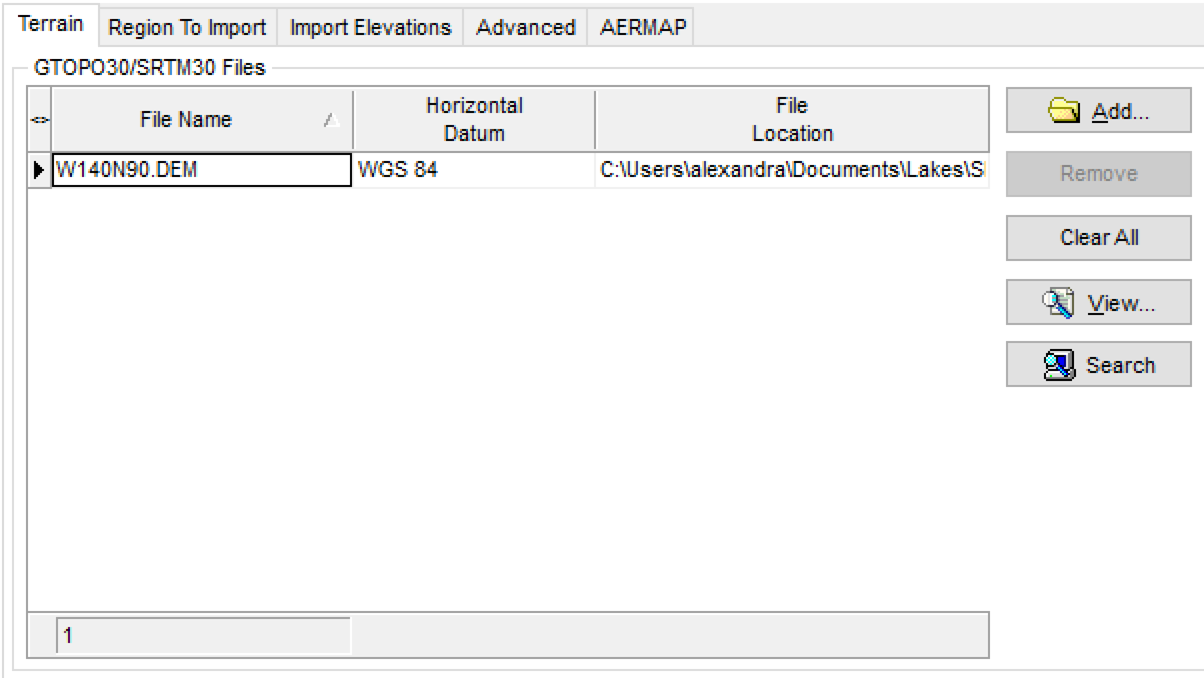
The direct download of **SRTM1 (Version 3)** files through the WebGIS is **ONLY** available if you have current maintenance. If you do not have maintenance, you will need to download the files from the USGS Global Data Explorer by selecting the **NASA SRTM 1 arcsec** layer.

3. Once all the files are specified, they appear in the list. Confirm that the data is correct by clicking the View... button to read the file(s) in WordPad.

4. If you have specified a file in error, you can remove it by selecting it and clicking the Remove button. To remove all files and start again, click the Clear All button.

5. To locate all files of the specified type on your computer, click the Search button. A sub-menu will display. You can elect to search the cache folder (see Preferences - Download Settings) or specify a folder/all its sub-folders.

Specifying GTOPO30/SRTM30 Files



If you have not set the Map Projection information for the project, click the Map Projection button to set it before downloading files from WebGIS.

1. If you already have a GTOPO file, click the 

button and specify that file. You can select multiple files.

2. If you do not already have the appropriate GTOPO file(s), click the 

button. A sub-menu will display. Choose either **GTOPO30 (Global ~900m)** or **SRTM30 (Global ~900m)**. The appropriate files will be automatically downloaded from the WebGIS website.

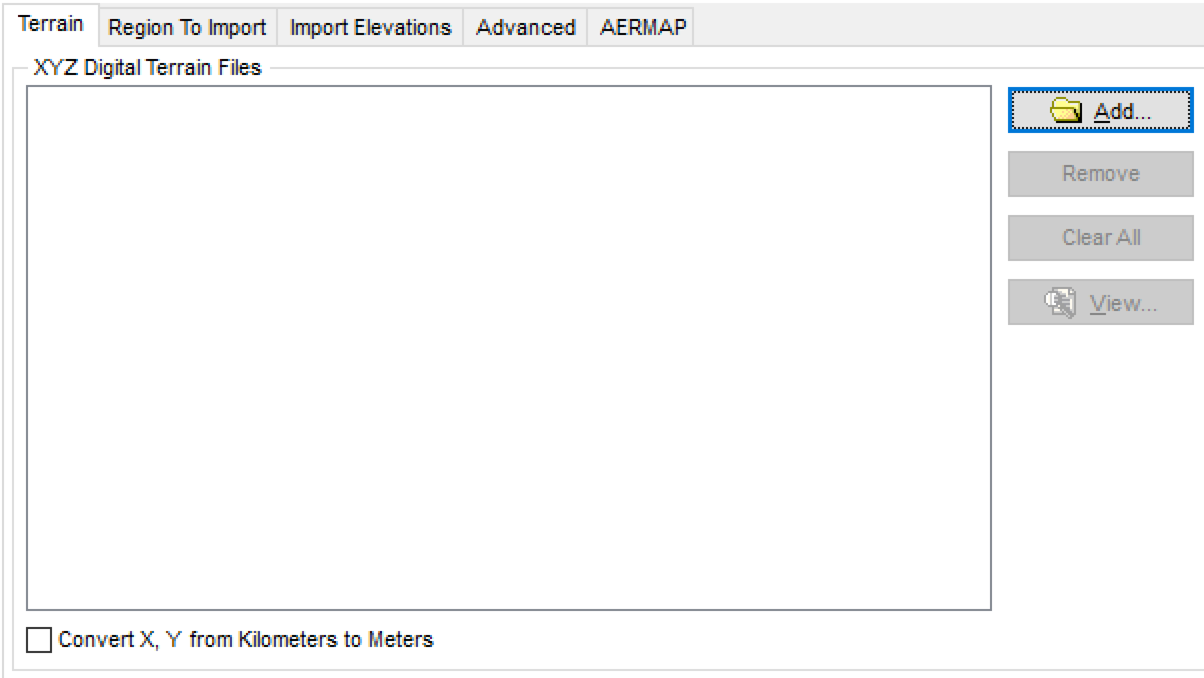
WebGIS download menu

3. Once all the files are specified, they appear in the list. The extents of the imported terrain file are displayed in the **Preview** panel in hatched blue. Confirm that the data is correct by clicking the View... button to read the file(s) in WordPad.

4. If you have specified a file in error, you can remove it by selecting it and clicking the Remove button. To remove all files and start again, click the Clear All button.

5. To locate all files of the specified type on your computer, click the Search button. A sub-menu will display. You can elect to search the cache folder (see Preferences - Download Settings) or specify a folder/all its sub-folders.

Specifying XYZ Files



1. Click the **Add...** button.

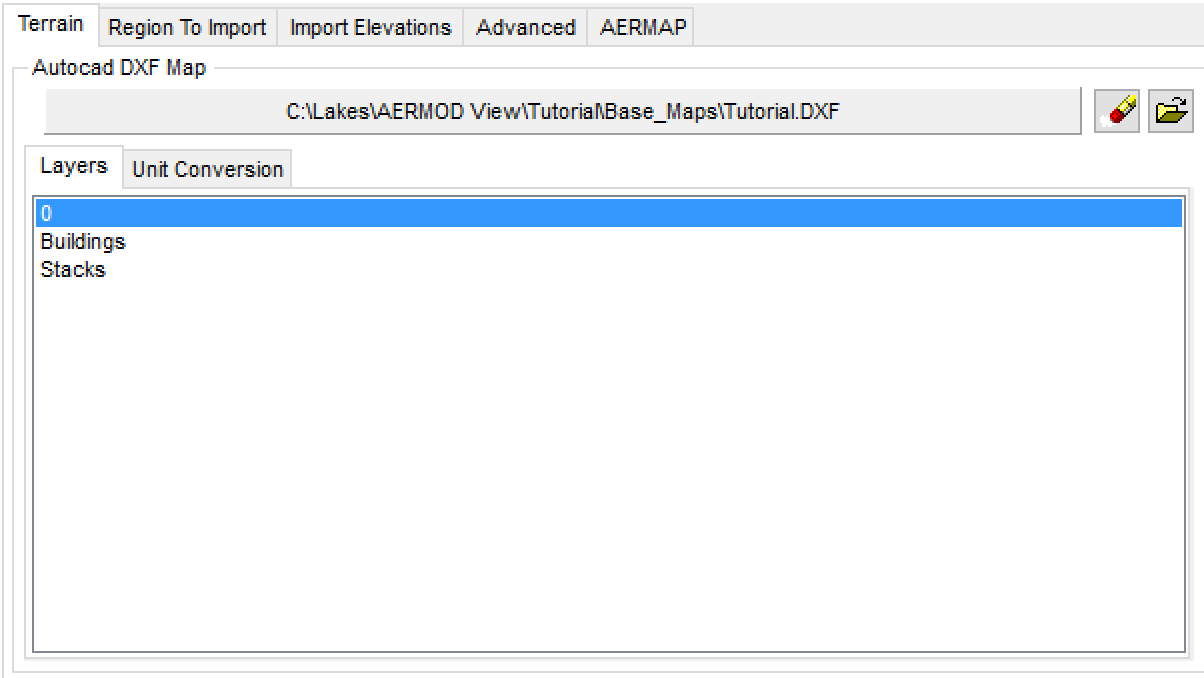
2. Select the file containing the xyz coordinates. The XYZ file can be a space, tab, or comma delimited file (*.txt,*.csv). You can select multiple terrain files.

3. Once all the files are specified, they appear in the list. Confirm that the data is correct by clicking the **View...** button to read the file(s) in WordPad.

4. If you have specified a file in error, you can remove it by selecting it and clicking the **Remove** button. To remove all files and start again, click the **Clear All** button.

5. If your XYZ file contains the x and y coordinates in kilometers, check the **Convert X, Y, from Kilometers to Meters** box.

Specifying AUTOCAD DXF Files



1. Click the **Specify File** ( ) button.

2. Select the DXF file containing the terrain data. You can only select one DXF terrain file.

3. Once the file is specified, the layers in the file are displayed in the list.

4. If you need to convert the units in the file, click the **Unit Conversion** tab and select from the following options:

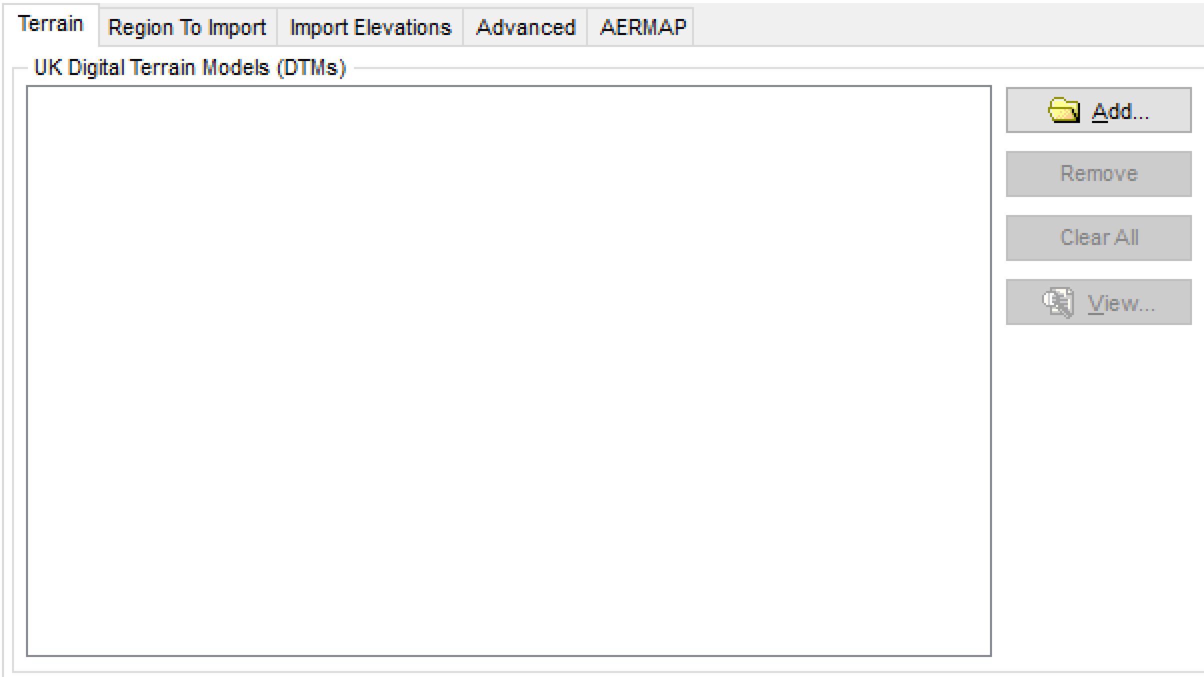
   - **No Conversion:** This is the default option used and is used if the DXF file to be imported was drawn in meters. The DXF will be imported without any conversions.

   - **Feet To Meters (0.3048):** Use this option if the units in your DXF base map are in feet. AERMOD View will scale the DXF file by a factor of 0.3048, the equivalent conversion of feet to meters.

   - **Custom Factor:** Use this option if the units in your DXF base map are not in meters or feet. Allows you to specify a custom conversion factor to be applied to the DXF file. In this case, AERMOD View converts your base map units automatically to meters using the unit conversion factor you have specified in the text box.

5. If you have specified a file in error, you can remove it by selecting it and clicking the **Clear File** button.

Specifying UK DTM or NTF Files



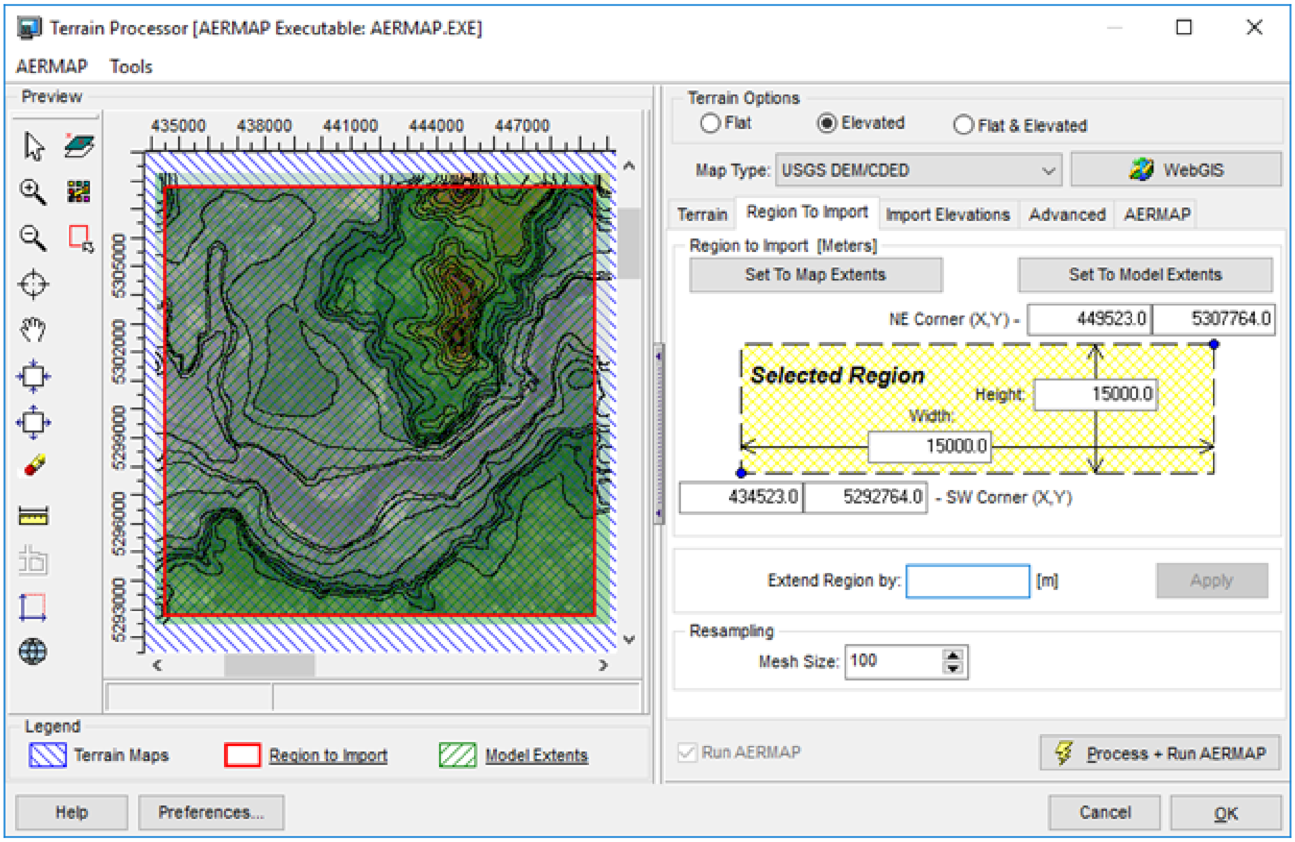
1. Click the Add... button.
2. Select the terrain data file. You can select multiple terrain files.
3. Once all the files are specified, they appear in the list. Confirm that the data is correct by clicking the View... button to read the file(s) in WordPad.
4. If you have specified a file in error, you can remove it by selecting it and clicking the Remove button. To remove all files and start again, click the Clear All button.

Once you have specified the digital terrain elevation files to be used, press the Process + Run AERMAP button to run the AERMAP model in order to process terrain data for your project.

> At least one receptor must be specified in order for AERMAP to run. If no receptors have been defined, an error will be displayed and the AERMAP run will not proceed.

Specify the Region

Once you have specified the terrain elevation files to be imported in the Terrain tab of the **Terrain Processor** dialog, you will need to specify the region to import so that AERMOD View can create a file containing only the terrain elevations for the defined region. The **Region to Import** tab allows you to select a region of the terrain elevation file to be imported.



Terrain Processor - Region to Import tab

How to Specify the Region to Import

1. Set the **Region to Import**. There are four methods available:

- Click the Set To Map Extents button to import the entire specified terrain map(s), represented in the **Preview** panel as the blue hatched region.

- Click the Set To Model Extents button to import portion of the terrain data file that is within the extents of the modeling domain, represented in the **Preview** panel as the green hatched region.

- Click the tool, and draw a rectangle in the **Preview** area to represent the region to import.
- Manually specify the X and Y coordinates for the **SW Corner**, the **Width** (X dimension), and the **Height** (Y dimension) of the region to import

> It is possible to select a region that is not completely covered by DEMs, but only if you are not running AERMAP. AERMAP requires complete coverage of your modeling area. This option may be used, for instance, when DEM coverage is not available for certain areas. A warning message will be displayed during processing and you will be responsible for specifying terrain elevation values for the receptors that are not covered by DEMs. The Elevations/Flagpole Heights tool available in the Application Toolbar allows you to easily specify terrain elevations for receptors that fall within a selected region.

2. You can elect to extend the region by specifying the number of meters to extend the region by and clicking the **Apply** button. This is particularly useful if you have chosen the **Set to Model Extents** option.

> For new projects, AERMOD View will automatically apply a default buffer zone of 500 meters around the modeling domain as the specified domain should extend to main terrain features that need to be included in the calculation of critical hill height scales.

> The **Extend Region by** function is additive; if you extend the region by 200 meters and apply the change, and then enter 250 meters in the **Extend Region by** box and apply the change again, the total number of meters that you will extend the modeling area by is 450.

> AERMOD View automatically defines the region to import (defined by the red box in the **Preview** window) by using the model domain extents. If the terrain elevation files do not cover your modeling domain at all, then the files will not be processed.

3. To resample the imported terrain data on a regular grid, which allows you to generate 2D and 3D terrain visualizations, specify a **Mesh Size** in the **Resampling** panel. A default mesh size of 40 is used, but for visualization purposes a more refined mesh size may be necessary. Please note that as mesh size increases, the terrain visualization quality improves but computational requirements increase. We recommend that you resample the terrain data for a mesh size of 40 and only use a more refined mesh if you need to increase quality for displaying your modeling results.

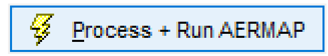
4. Once you have specified the region to import, process the terrain. Depending on the U.S. EPA model you are using, there are two ways in which the terrain is processed:

- **AERMOD:** If using the AERMOD model, terrain elevations are defined by AERMAP which is the U.S. EPA terrain pre-processor developed to process terrain data for use with the AERMOD model.

- **ISCST3/ISC-PRIME:** If using the ISCST3 or ISC-PRIME models you have two options:

- **Run AERMAP:** You can choose to run the AERMAP pre-processor to obtain your terrain elevations. In this case you will need to check the **Run AERMAP** box.

- **Lakes Algorithm:** If you do not check the Run AERMAP box, terrain elevations will be imported directly from the terrain elevation file using the Lakes Algorithm.

Depending on which of the above options you choose, click on the 

button. AERMOD View will pre-process your DEM data and create an intermediate database file containing elevations for the selected imported region. This intermediate file will be used by AERMOD View to extract terrain elevations for your receptors and is used for the visualization of your terrain in 2D and 3D.

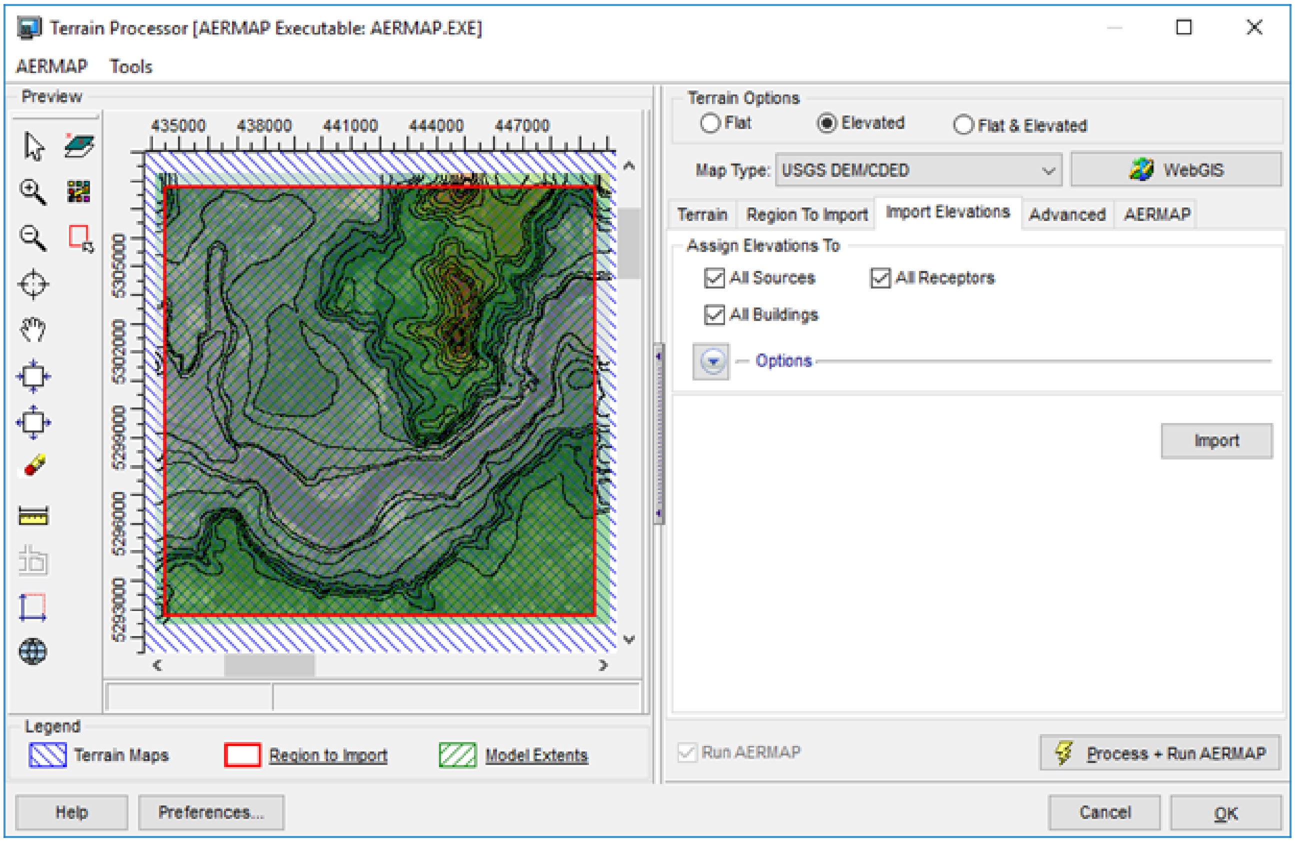
> When the intermediate database file containing elevations for the selected imported region is created, an overlay is also created called the Imported Terrain Region overlay. To be able to view this overlay in the drawing area, you should make this overlay active by checking the corresponding box in the Tree View - Overlays tab. The Imported Terrain Region overlay can help you identify whether or not all of your sources and receptors are within the imported terrain region.

> At least one receptor must be specified in order for AERMAP to run. If no receptors have been defined, an error will be displayed and the AERMAP run will not proceed.

5. Proceed to the Import Elevations tab to assign the terrain elevations to the sources, receptors and buildings in your project and import the elevations.

Import Elevations

After you have processed the terrain elevation files in the Region to Import tab of the Terrain Processor, you will need to assign the terrain elevations to the receptors, sources and buildings in your project. This can be done in the **Import Elevations** tab.

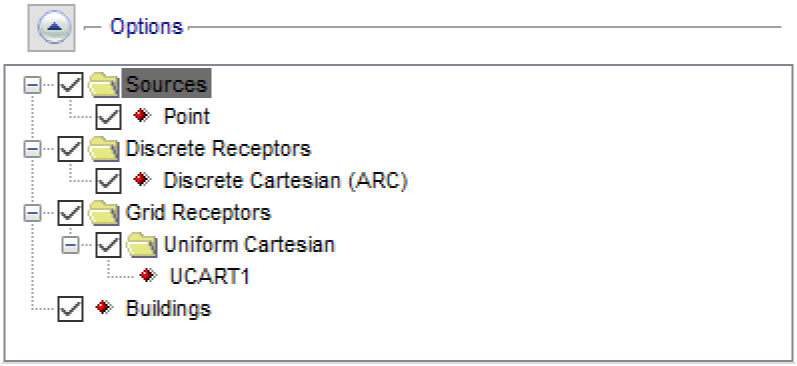


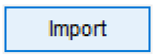
Terrain Processor - Import Elevations tab

How to Assign Elevations

1. In the **Assign Elevations To** section, choose whether you wish to assign elevations to objects by checking or unchecking the boxes next to the object type.

2. If you wish to refine your selection, click the  button to display further options. Here you can select specific types of sources/buildings/receptors to assign elevations to. Uncheck any features that you have manually entered elevations for, otherwise they will be overwritten with the calculated ones upon import.



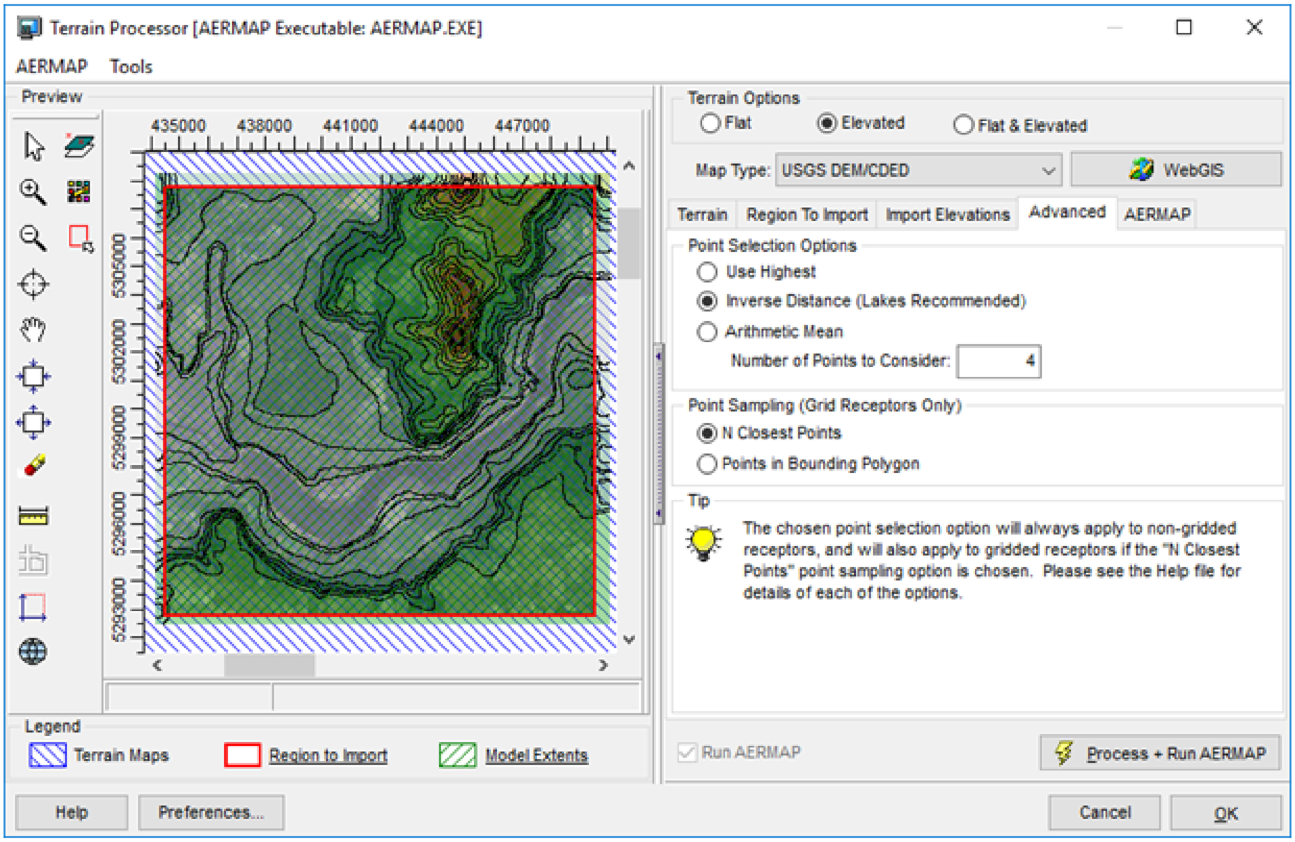
3. Click on the 

button to assign and import the terrain elevations. If you wish, you may specify some advanced options for importing and assigning the terrain elevations in the Advanced Options tab.

Set Advanced Terrain Options

Warning: Advanced Users Only

The **Advanced Options** tab of the Terrain Processor allows you to specify some advanced options for assigning and importing the terrain elevations.



Terrain Processor - Advanced Options tab

How to Set Advanced Terrain Options

1. Set the **Point Selection Options**, which are applied to all non-gridded receptors. Choose one of the following options:

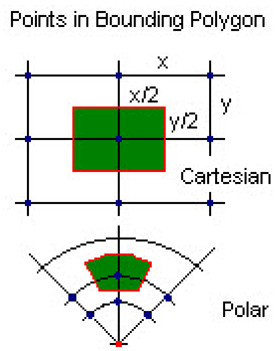
   - **Use Highest:** Uses the highest of the neighboring points as the required elevation.
   - **Inverse Distance (Lakes Recommended):** Interpolates the neighboring points using inverse distance to obtain elevation at desired point.
   - **Arithmetic Mean:** Uses a simple arithmetic mean to obtain elevation at desired point.

2. Enter a number in the **Number of Points to Consider:** field. This represents the number of points surrounding the desired location that are used on the interpolation scheme.

3. For Grid Receptors, set the Point Sampling (Grid Receptors Only) option, as follows:

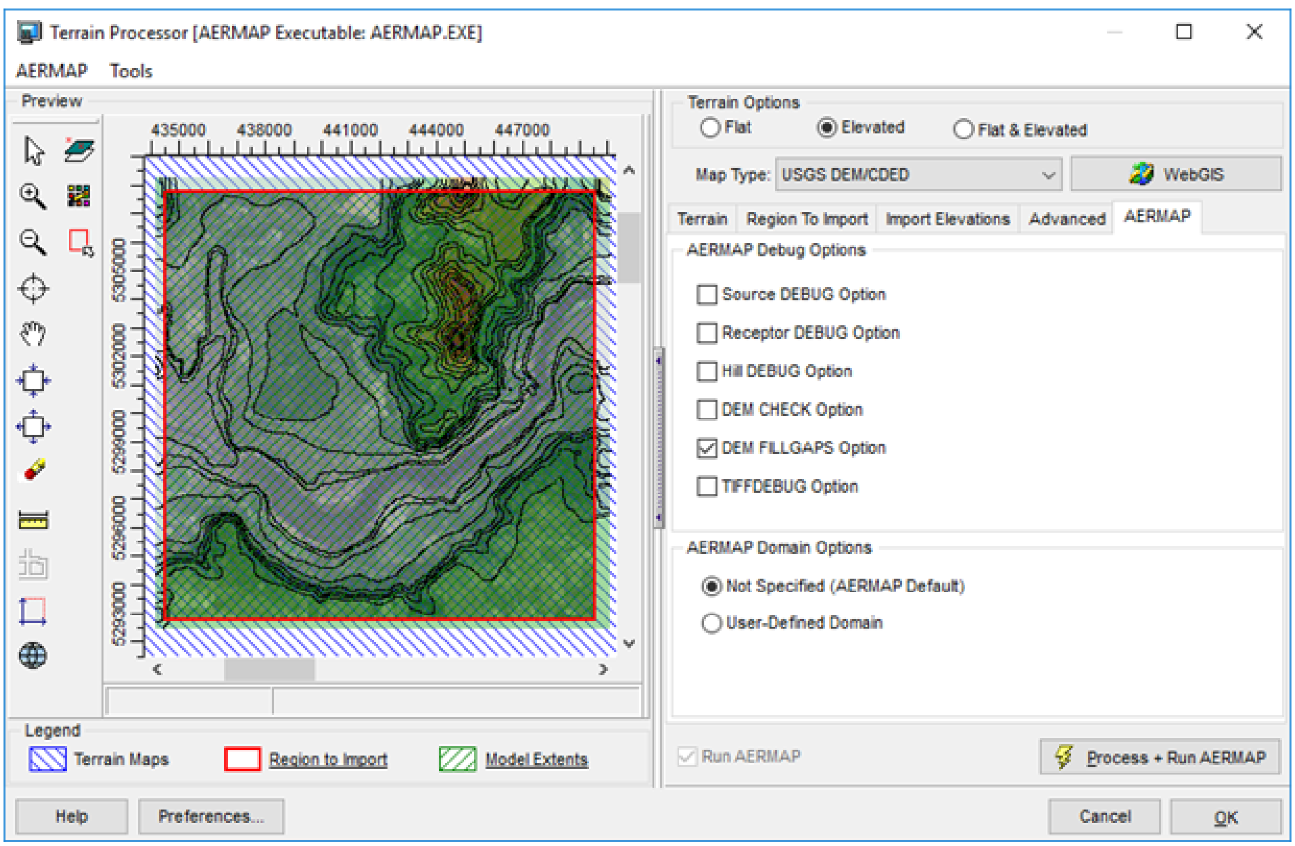
   - **N Closest Points:** The chosen Point Selection option will apply to the gridded receptors if this sampling option is selected.

- **Points in Bounding Polygon:** This option imports the highest elevation from within a bounding polygon. The bounding polygon is defined by half the distance to adjacent receptor grid nodes.



AERMAP

The **AERMAP** tab provides you with options to add supplemental debug data to the output file.



Terrain Processor - AERMAP tab

The options available in this tab are as follows:

AERMAP Debug Options

- **Source DEBUG Option:** If this option is checked, three files will be generated when AERMAP is run:
    - **ProjectName_SRC_Detail.OUT:** Contains details of NAD conversion results for sources.
    - **ProjectName_SRC_Elev.OUT:** Contains calculation information for sources.
    - **ProjectName_SRC_NDem.OUT:** Contains information about which DEM file contains which source.
- **Receptor DEBUG Option:** If this option is checked, three files will be generated when AERMAP is run:
    - **ProjectName_REC_Detail.OUT:** Contains details of NAD conversion results for receptors.
    - **ProjectName_REC_Elev.OUT:** Contains calculation information for receptors.
    - **ProjectName_REC_NDem.OUT:** Contains information about which DEM file contains which receptor.
- **Hill DEBUG Option:** If this option is checked, a ProjectName_HILL_Debug.OUT file will be generated when AERMAP is run. This file contains calculations of the critical hill height scales.
- **DEM CHECK Option:** This option allows you to run a check on the DEM files to make sure they are read correctly, as well as document the contents for informational or debugging purposes. Depending on the size and resolution of the DEM file this may take a certain amount of time.
- **DEM FILLGAPS Option:** This option allows AERMAP to fill in the gaps between different terrain map regions and allow for complete coverage of the modeling domain.

> **DEM FILLGAPS** is the only option selected by default. This is done to make sure that no model elements are left with no calculated elevations due to gaps between map regions.

- **TIFFDEBUG Option:** This option allows you to generate a TiffDebugFile_n.DBG (where "n" is the number of the NED file for which the debug file was generated) file for each NED data file which contains a listing of the TiffTags and GeoKeys in the GeoTIFF file.

AERMAP Domain Options

- **Not Specified (AERMAP Default):** This option does not limit the interpolation of elevations to any specific boundary. This prevents the exclusion of receptors on the edge of the modeling domain.

- **User-Specified Domain:** If this option is selected, only the terrain within the user-defined domain area will be used for interpolation.

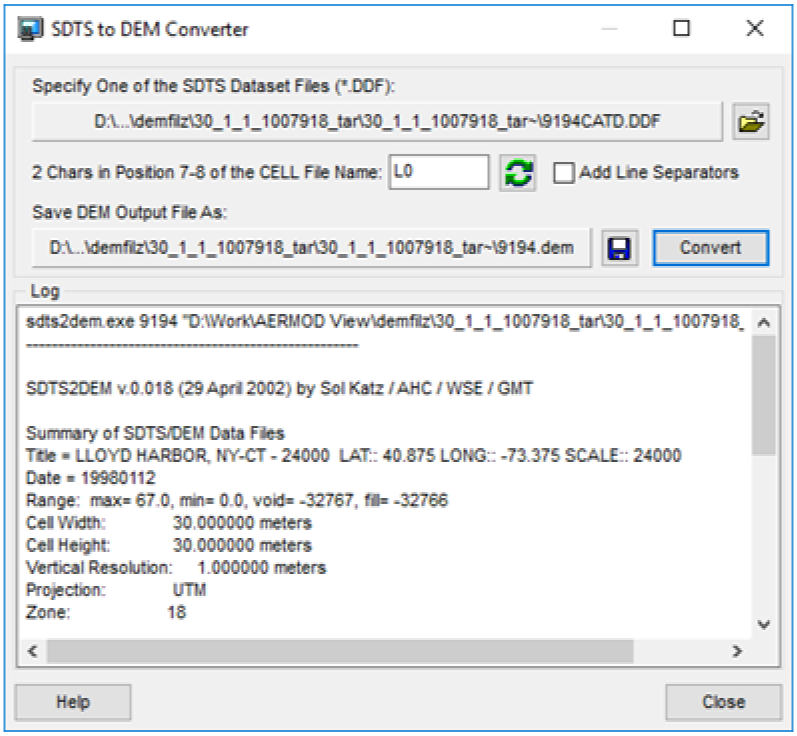
> For more information on AERMAP options please refer to the USEPA, 2011.

Convert SDTS Files to DEM Files

The U.S. EPA AERMAP does not read the new Spatial Data Transfer Standard (SDTS) formatted DEM directly; instead, the user is required to convert this format to an AERMAP-accepted DEM format. The **SDTS to DEM Converter** utility is an interface for the SDTS2DEM.EXE program that allows you to convert SDTS formatted DEMs into AERMAP-accepted DEM files for use in AERMAP. In AERMOD View, this utility can be found in the Terrain Processor dialog by selecting **Tools | SDTS to DEM...** function.

How to Convert SDTS Files to DEM Files:

1. From the Terrain Processor, select **Tools | SDTS to DEM...** to open the **SDTS to DEM Converter** dialog.



SDTS to DEM Converter dialog

2. Click on the button to specify one of the SDTS data set files (*.DDF). Each data set consists of approximately 18 files; however, you only need to specify one of the files.

3. Now enter the two characters found in the seventh and eighth position of the "CEL" file name. The two characters required are "LO".

> The "CEL" file is the largest file with the full file name being "1234CE**LO**.DDF".

> The **Add Line Separators** check box allows you to add line breaks in the newly created DEM file. This check box should only be selected if you plan on using the DEM files directly in AERMAP and outside of the AERMOD View interface. AERMOD View will accept DEMs with or without line breaks.

4. Click on the to specify a name and location for the DEM file that will be created using this utility. By default, the newly created DEM file will be saved in the same location as the *.DDF files and a default name will be given using the first four numbers of the SDTS data set files (e.g. 9194CELO.DDF will be saved as 9194.dem). If you wish you can change the name and/or location of the newly created file.

5. Click Convert to begin converting your SDTS set of files to a DEM file. The Log screen shows you how the conversion is progressing and will inform you of any errors that may have occurred. When the conversion has completed successfully, a message will appear indicating that the

conversion was successful. Click the **OK** button and then the **Close** button in the **SDTS to DEM Converter** dialog to return to the Terrain Processor dialog.

6. From the Terrain Tab, click the 

button to add the DEM file, created using the SDTS to DEM Converter utility, to your project. Make sure the Map Type field is set to USGS DEM.

> SDTS formatted DEMs are usually contained in a zip file. In order to specify SDTS files in the SDTS to DEM Converter utility, you must first extract the contents of the zip file ensuring that the files specified in the SDTS to DEM Converter are the ones with the *.DDF file extension.

Terrain File Types

The following file types can be imported into the **Terrain Processor**:

- USGS DEM/CDED
- NED GeoTIFF
- SRTM1/SRTM3
- GTOPO30/SRTM30
- XYZ File
- AutoCAD DXF
- UK DTM/UK NTF

Digital Terrain Data (DEM) Files

US Terrain Data - USGS 1-Degree DEM and USGS 7.5-Minute DEMs

USGS DEM files are digital elevation model files consisting of terrain elevations for ground positions at regularly spaced horizontal intervals produced by the USGS (U.S. Geological Survey). AERMOD View supports two distinct digital elevation data products distributed by the USGS in the standard digital elevation model (DEM) tape format:

- 7.5-Minute DEM Data
- 1-Degree DEM Data

> As of the AERMAP revision in version 09040, AERMAP can now process DEM files of mixed format (e.g. 7.5-minute and 1-degree DEM files) in the same run.

Canadian Terrain Data - 1-Degree DEM and 15-Minute DEMs

The 1-Degree and 15-Minute DEMs available automatically in WebGIS have been retrieved by using the integrated 

feature in the Terrain Processor.

WebGIS download menu

The files available are from the **Natural Resources Canada's Canadian Digital Elevation Data (CDED)** dataset, which is terrain data in USGS DEM type data format for a 1:50,000 and a 1:250,000 map scale (NAD83). CDED consists of an ordered array of ground elevations at regularly spaced intervals. This dataset is recommended by both **Guidelines For Air Quality Dispersion Modelling In British Columbia** and **Alberta Environment's Air Quality Model Guideline**.

- the 250,000 map scale data (1-Deg DEM) has a grid resolution range of 3 to 12 arc-seconds, depending on latitude. For data south of 68° N latitude, the grid resolution is roughly 90 m, depending on latitude. The data sheets are directly readable by AERMAP and TERREL and are equivalent to the one-degree USGS DEM data.

- the 1:50,000 map scale data (15-Minute DEM) has a grid resolution range of 0.75 to 3 arc-seconds. For data south of 68° N latitude, the grid resolution is roughly 20 m, depending on latitude.

The files available from the WebGIS.com website were transferred intact with no modification from the CDED dataset.

7.5-Minute DEM Data

Digital Elevation Models (DEMs) are arrays of elevations, usually at regularly spaced intervals, for a number of ground positions. Two distinct digital elevation data products are distributed by the **United States Geological Survey (USGS)** in the standard digital elevation model (DEM) tape format, 7.5-minute DEM data files and 1-degree DEM data files.

The 7.5-minute DEMs produced by the USGS correspond in coverage to standard 1:24,000-scale 7.5- x 7.5-minute quadrangles and are based on 30- by 30-meter data spacing with the **Universal Transverse Mercator (UTM)** projection. Each 7.5-minute block provides the same coverage as the standard USGS 7.5-minute topographic quadrangle map series for all of the United States and its territories except Alaska.

The **7.5-Minute DEM** has the following characteristics:

- The data consist of a regular array of elevations referred in the **Universal Transverse Mercator (UTM)** coordinate system. Elevations randomly located in an irregular arrays have been produced to date.
- The unit of coverage is the 7.5-minute quadrangle. Overedge coverage is not provided.
- The data are ordered from south to north in profiles that are ordered from west to east.
- The data are stored as profiles in which the spacing of the elevations along and between each profile is 30m.
- The profiles do not always have the same number of elevations due to the variable angle between true north and grid north of the UTM coordinate system.
- The 7.5-minute DEM data are produced in 7.5- x 7.5-minute blocks either from map contour overlays that have been digitized or from automated or manual scanning of photographs usually taken at an average height of 40,000 ft. (1:80,000-scale). The data are processed to produce a DEM with a 30m sampling interval.

> The 7.5-minute DEM data files and the 1-degree DEM data files are identical in logical data structure but differ in sampling interval, geographic reference system, areas covered, and accuracy of data. USGS 7.5-minute DEM data are available for selected quadrangles in the United States; 1-degree DEM data are available for most of the United States.

1-Degree DEM Data

Digital Elevation Models (DEMs) are arrays of elevations, usually at regularly spaced intervals, for a number of ground positions. Two distinct digital elevation data products are distributed by the **United States Geological Survey (USGS)** in the standard digital elevation model (DEM) tape format, 7.5-minute DEM data files and 1-degree DEM data files.

The 1-degree DEM data files produced by the **Defense Mapping Agency (DMA)** correspond in coverage to 1- by 1- degree (one half of standard 1:250,000-scale 1° x 2° quadrangles) and are also referred to as 3-arc second or 1:250,000 DEM data. The 1-degree DEM (3- by 3-arc-second data spacing) provides coverage for all of the contiguous United States, Hawaii, and limited portions of Alaska.

The **1-Degree DEM** in the United States (except Alaska) has the following characteristics:

- The data consist of a regular array of elevations cast on the geographic coordinate system.
- The unit of coverage is a 1° x 1° block representing one-half of a 1° x 2° 1:250,000 scale map. The unit of coverage includes profiles coincident with the neatlines of the map.
- The data are ordered as profiles ascending northward. The origin is at the southwest corner of the map.
- The data are stored as profiles in which the spacing of the elevations along and between each profile is 3 arc-seconds.
- The data comprise an array having 1,201 profiles with 1,201 elevations per profile.

For the State of Alaska, the spacing of elevations along each profile is 3 arc-seconds (1,201 elevations per profile), and the normal spacing between profiles varies from 6 arc-seconds (601 profiles per DEM) in the south to 12 arc-seconds (151 profiles per DEM) on the north slope of the State. Some Alaska sheets have a 4-arc-second spacing of the profiles.

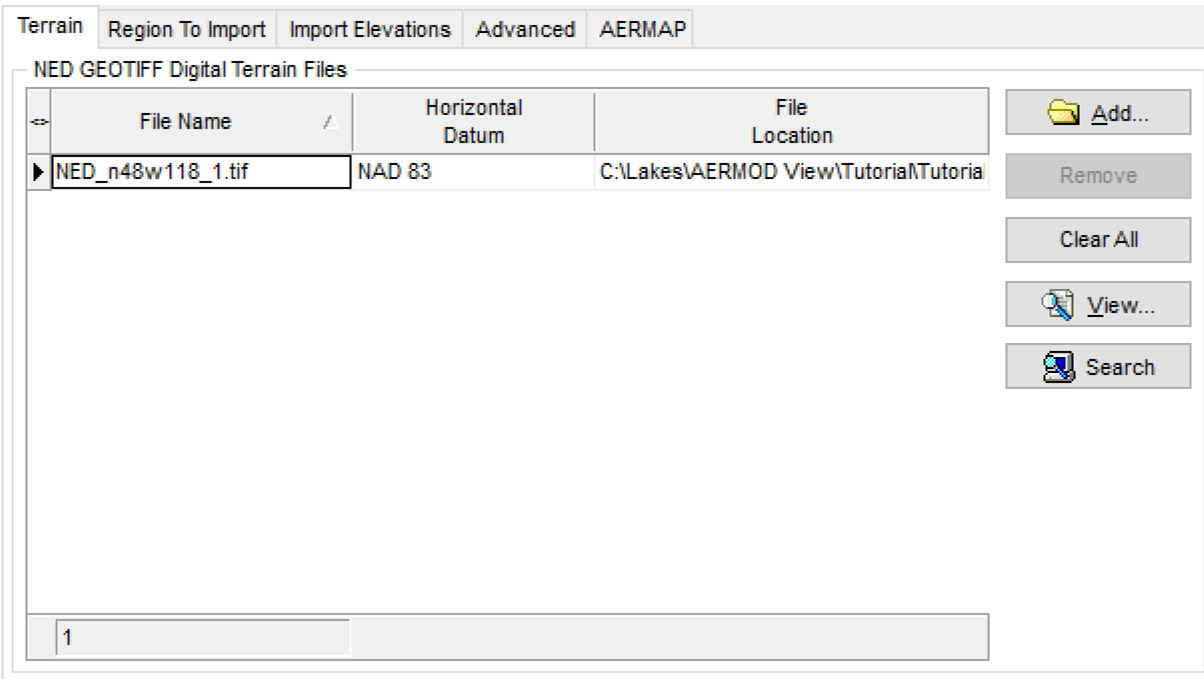
The 1-degree DEM data are produced by interpolating elevations at intervals of 3 arc-seconds from contours, ridge lines, and drains digitized from 1:250,000-scale topographic maps. Three seconds of arc represent approximately 90m in the north-south axis and a variable dimension (approximately 90m at the equator to 60m at 50° latitude). In the east-west axis due to convergence of the meridians. The area of each map is divided into an east half and a west half to accommodate the large volume of data required to cover the 1° x 2° topographic map area.

> The 7.5-minute DEM data files and the 1-degree DEM data files are identical in logical data structure but differ in sampling interval, geographic reference system, areas covered, and accuracy of data. USGS 7.5-minute DEM data are available for selected quadrangles in the United States; 1-degree DEM data are available for most of the United States.

USGS NED Files

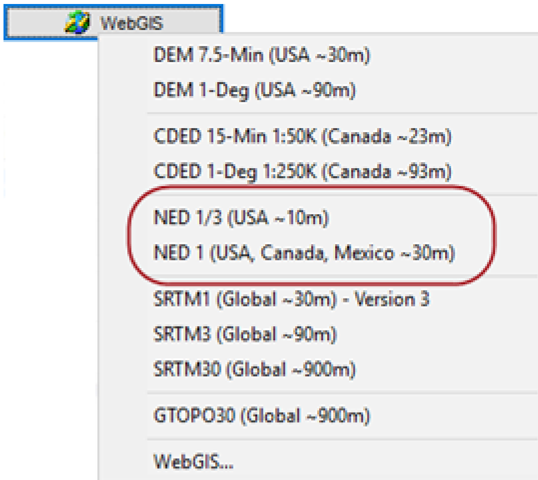
USGS NED GeoTIFF terrain data are originated from the National Elevation Dataset consisting of ground surface elevation data for the United States, Canada and Mexico produced by the USGS (U.S. Geological Survey). The NED terrain data, in GeoTIFF format, can be used directly with the US EPA AERMAP model, dated 09040 and above.

As of March 19, 2009, USGS NED GeoTIFF is the terrain data set recommended by the US EPA for use in the United States for regulatory purposes. Users outside the United States should continue to use the terrain data most appropriate to their location and regulatory requirements.



Terrain Processor - Add NED GeoTIFF files

The GeoTIFF format is a binary file that includes data descriptors and geo-referencing information in the form of "TiffTags" and "GeoKeys." AERMAP processes these TiffTags and GeoKeys to determine the type and structure of the elevation data within the NED file. NED elevation files can be obtained directly through the AERMOD View integrated WebGIS feature if you have current maintenance.



WebGIS download menu

SRTM Terrain Data Files

The **Shuttle Radar Topography Mission (SRTM)** obtained elevation data on a near-global scale to generate the most complete high-resolution digital topographic database of Earth to date. Data for nearly the whole world is now available for download. Data are divided into one by one degree latitude and longitude tiles in "geographic" projection. Heights are in meters referenced to the WGS84/EGM96 geoid.

There are three types of SRTM terrain data files:

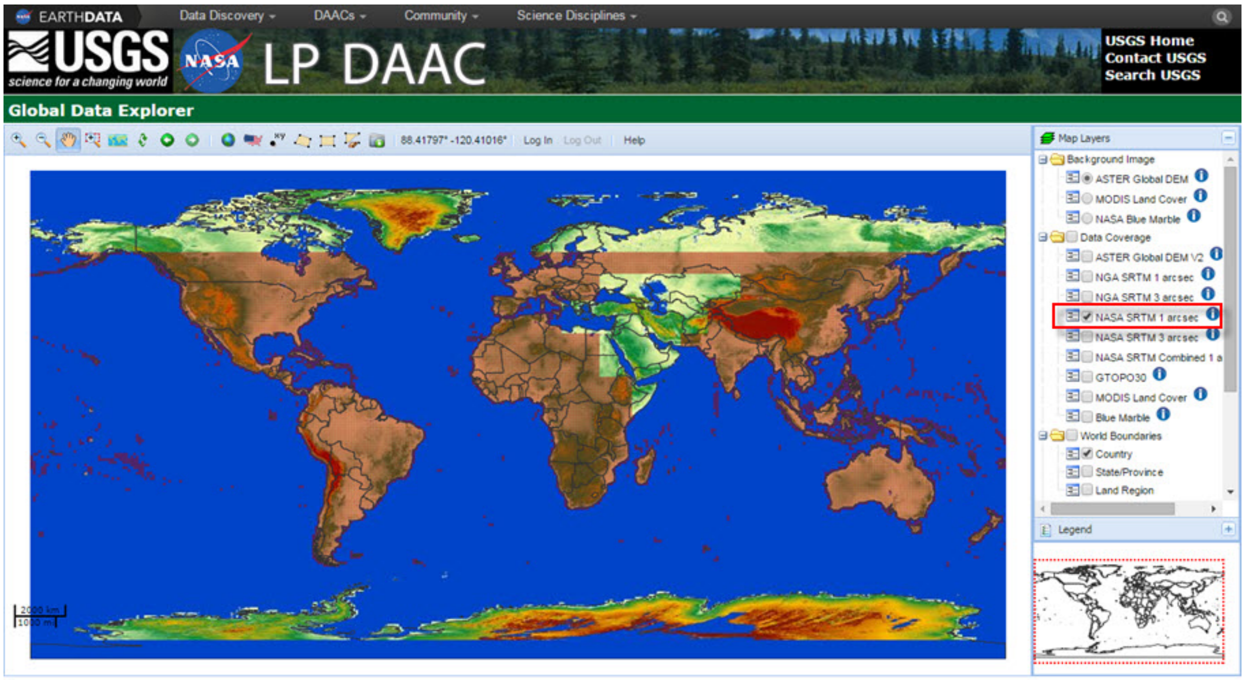
SRTM1- Version 3

- **Resolution:** ~30 m, 1 arc-sec
- **Coverage:** Global
- **Projection:** Geographic
- **Datum:** WGS84
- **Vertical Units:** Meter

SRTM1 Version 3 is the highest quality SRTM data available to date. It covers regions between 60° north and 58° south latitude with exception of a few regions in Western Asia and Northeastern Africa which may available in the near future.

> The automatic download of **SRTM1 Version 2**, which covered only the United States, has been **discontinued** for AERMOD View version 9.1 and above. You can still upload these files (*.hgt) using the **Add** button in the Terrain Processor.

> The direct download of **SRTM1 (Version 3)** files through the WebGIS is ONLY available if you have current maintenance. If you do not have maintenance, you will need to download the files from the USGS Global Data Explorer by selecting the **NASA SRTM 1 arcsec** layer (as shown below). Note that additional data will be added to WebGIS as it becomes available.



Current Coverage for SRTM1 Global Data as of Dec 2015

SRTM3

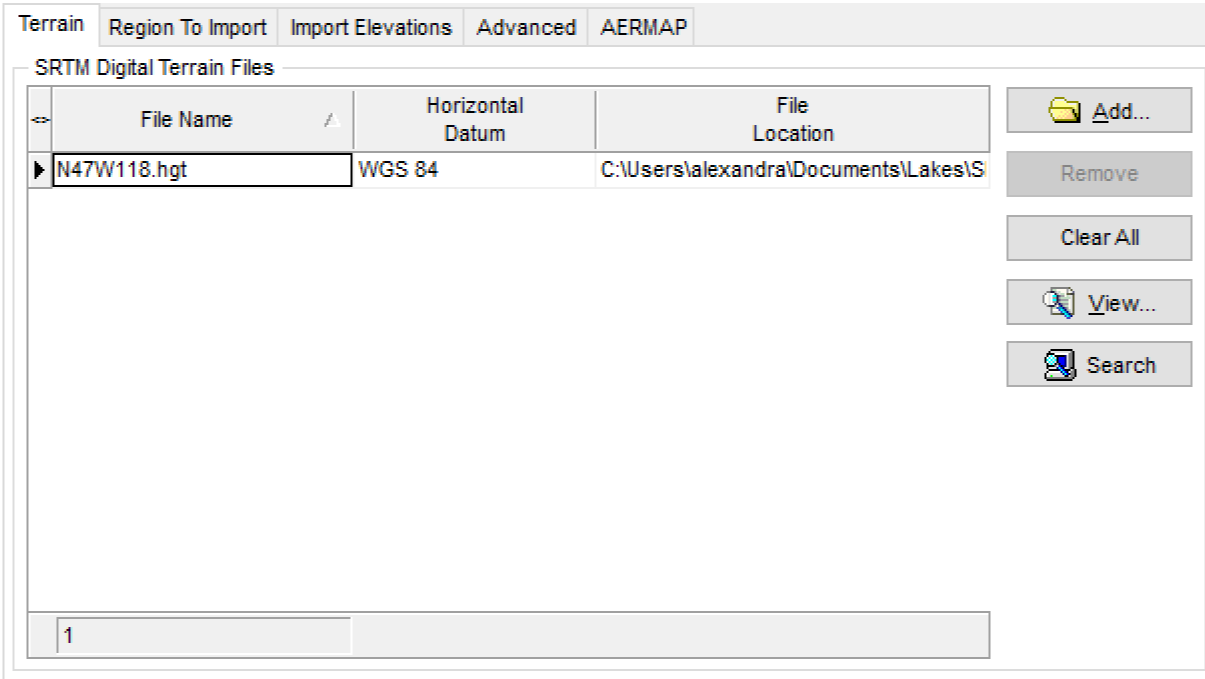
- **Resolution:** ~90 m, 3 arc-sec
- **Coverage:** Global
- **Projection:** Geographic
- **Datum:** WGS84
- **Vertical Units:** Meter

You can download the SRTM files by using the integrated WebGIS feature in the Terrain Processor.

WebGIS download menu

To learn more about the Shuttle Radar Topography Mission, please visit http://www2.jpl.nasa.gov/srtm/

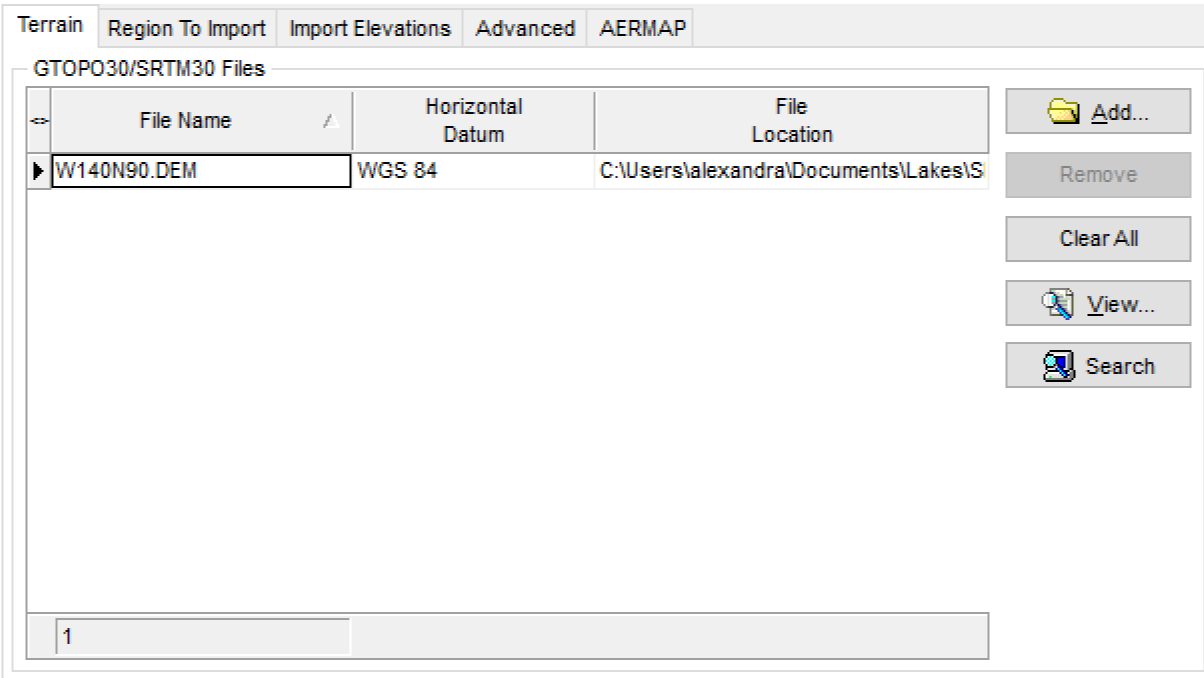
The names of individual data tiles refer to the latitude and longitude of the lower-left (southwest) corner of the tile. For example, the coordinates of the lower-left corner of tile N40W118.hgt are 40 degrees North latitude and 118 degrees West longitude.



Terrain Processor - Add SRTM files

GTOPO30/SRTM30 Terrain Files

Selecting the GTOPO30/SRTM30 option in the **Terrain** data tab allows you to specify GTOPO30 and SRTM30 terrain files for processing:



Terrain Processor - Add GTOPO30/STRM30 files

- **GTOPO30:** a global digital elevation model (DEM) with a horizontal grid spacing of 30 arc seconds (approximately 1 kilometer) and was developed to meet the needs of the geospatial data user community for regional and continental scale topographic data. GTOPO30 is a global data set covering the full extent of latitude from 90 degrees south to 90 degrees north, and the full extent of longitude from 180 degrees west to 180 degrees east.

You can download the above GTOPO30 data files, free of charge, from the USGS web site, using links provided at Lakes Environmental web site: http://www.webgis.com/terr_world.html

- **SRTM30:** The Shuttle Radar Topography Mission (SRTM) obtained elevation data on a near-global scale to generate the most complete high-resolution digital topographic database of Earth to date. Data for nearly the whole world is now available for download. Data are divided into one by one degree latitude and longitude tiles in "geographic" projection. Heights are in meters referenced to the WGS84/EGM96 geoid.

SRTM30

- **Resolution:** ~1 km, 30 arc-sec
- **Coverage:** Global
- **Projection:** Geographic

- **Datum:** WGS84
- **Vertical Units:** Meter

GTOPO30

- **Resolution:** ~1 km, 30 arc-sec
- **Coverage:** Global
- **Projection:** Geographic
- **Datum:** WGS84
- **Vertical Units:** Meter

You can download the files using the integrated WebGIS feature.

WebGIS download menu

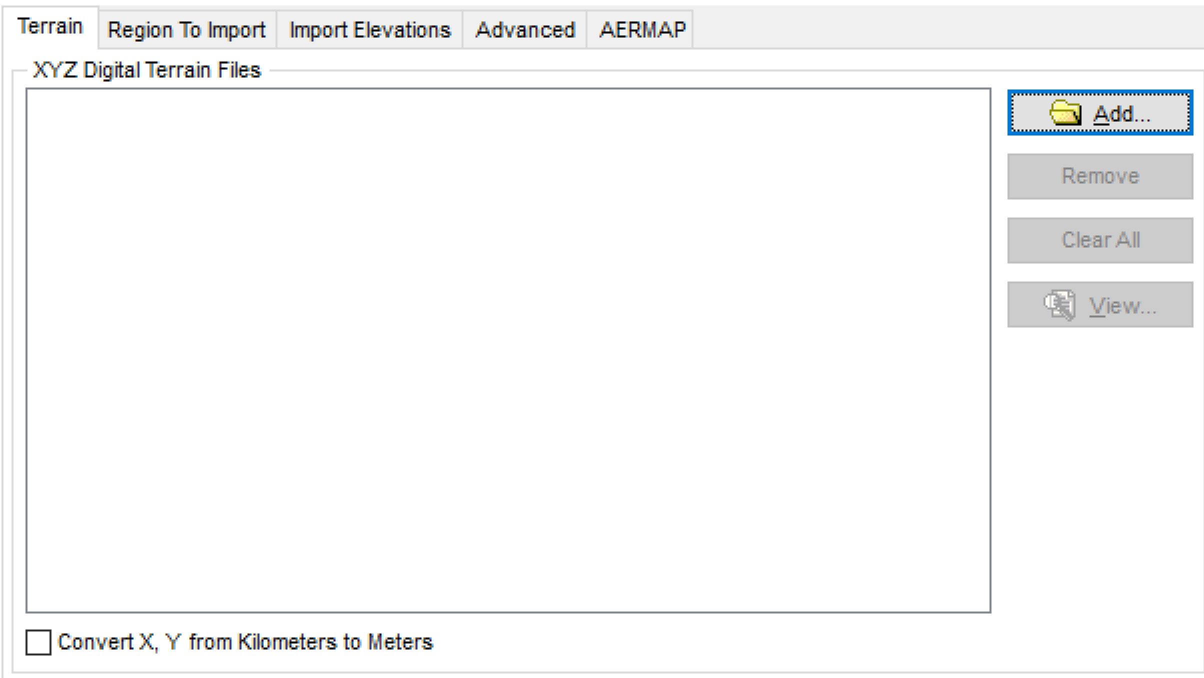
For more information about GTOPO30 please visit http://eros.usgs.gov. For more information on SRTM30 please visit http://www.src.com/datasets/SRTM_Info_Page.html.

While SRTM30 has the same resolution as GTOPO30, it can be considered a more accurate global digital data set compared to GTOPO30 because of its seamless and uniform representation, due to the fact that it was created over a short period of time from a single source rather than from the numerous sources spanning many decades that went into creating the GTOPO30 data set. However, it must be noted that the SRTM30 does not cover the poles north and south of approximately 60° latitude. Therefore, data for Antarctica, for example, must be obtained from the GTOPO30 data set.

XYZ files

An XYZ file is a comma delimited text file containing just xyz coordinates. This file type can be used for extracting the raw XYZ coordinates for elevation data.

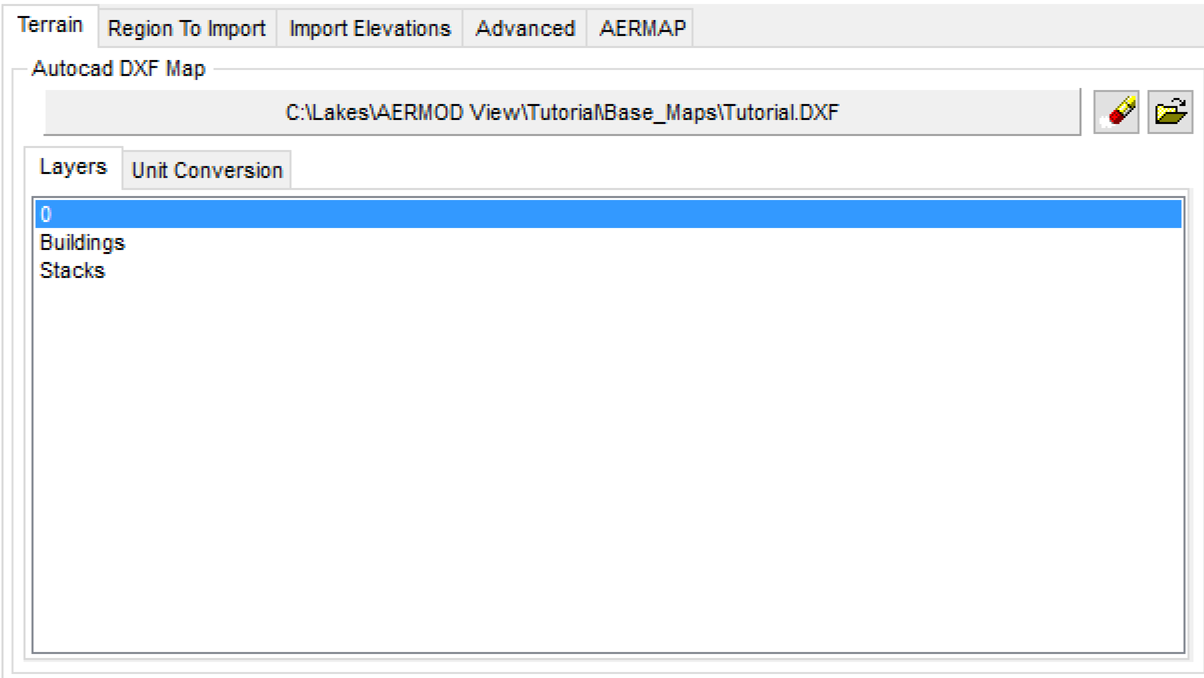
See Also: XYZ Terrain Elevations Data File Format.



Terrain Processor - Add XYZ files

AutoCAD DXF Files

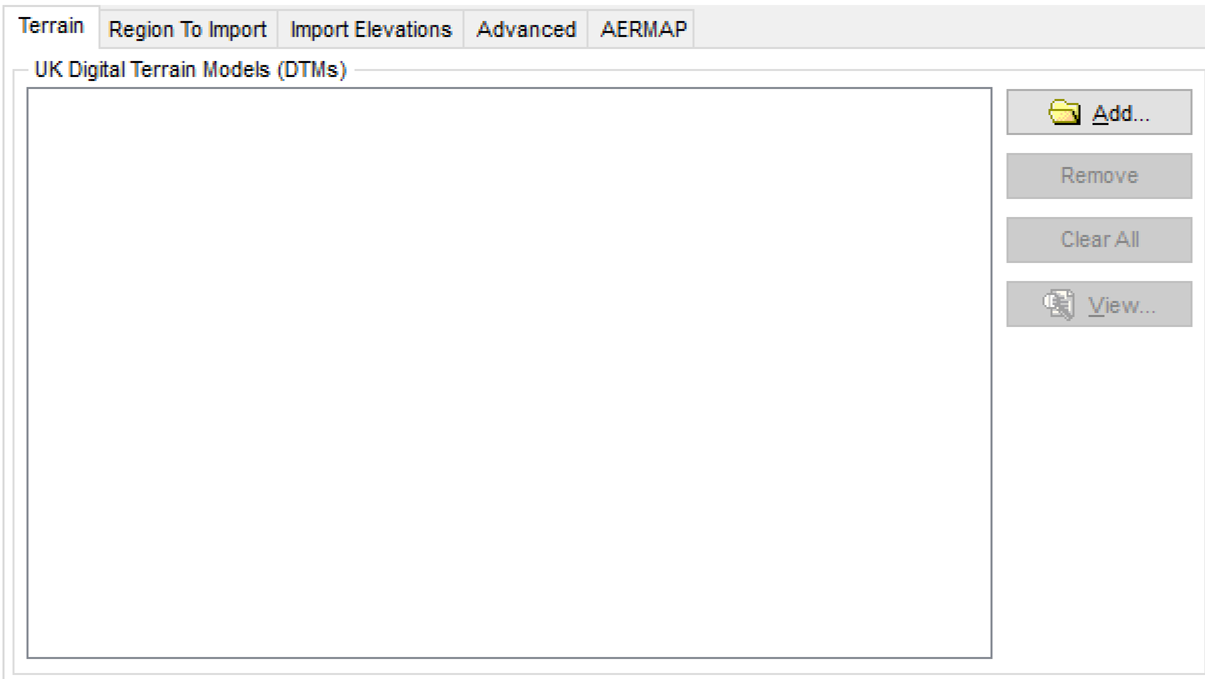
The **AutoCAD DXF (Drawing Interchange Format)** file is a standard format which can be used to import terrain elevations in AERMOD View.



Terrain Processor - Add DXF files

UK DTM and UK NTF Files

The **UK DTM (Digital Terrain Model)** and **UK NTF (National Transfer Format)** files are produced by the United Kingdom standard Ordnance Survey. AERMOD View supports both Land-Form Panorama and Profile NTFs - DTMS - as well as the DTM data format.



Terrain Processor - Add UK DTM files

RiskGen

This **Risk Mode** option is used to guide you on preparing the input files according to the **U.S. EPA OSW Human Health Risk Assessment (HHRAP)** and **Screening Level Ecological Risk Assessment (SLERAP)** Protocols.

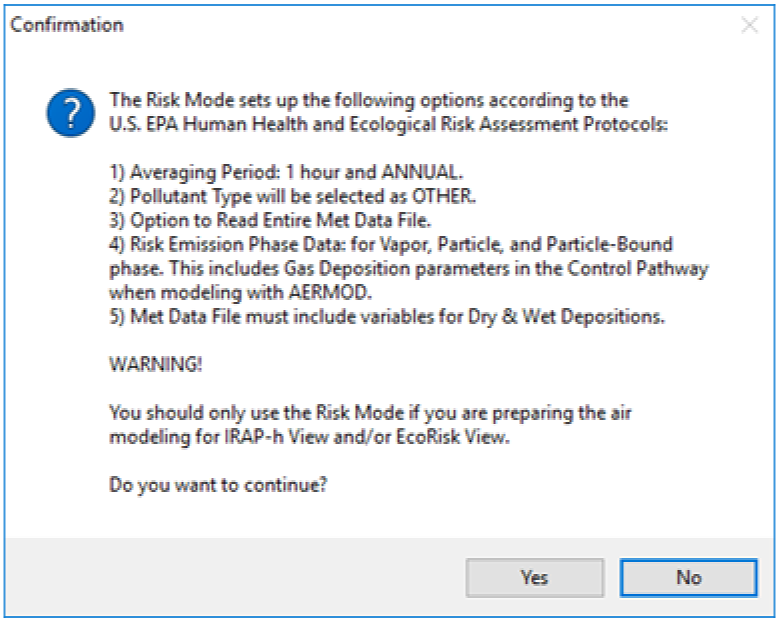
RiskGen - Risk Mode

The **Risk Mode** was implemented in AERMOD View as an add-on to users of IRAP-h View (Industrial Risk Assessment Program – Human Health) and EcoRisk View (Ecological Risk Assessment Program). The uses of the Risk Mode option allows you to quickly and easily comply with the **U.S. EPA OSW HHRAP** and **SLERAP** protocols. For more information on these programs, please contact **Lakes Environmental Software**.

How to Use the Risk Mode:

1. Select **Risk | Risk Mode (ON)** from the menu.

2. A **Confirmation** dialog is displayed asking confirmation to continue. Click **Yes** to continue.



The following options will be automatically setup in your AERMOD View project:

- A **1-hour** and **Annual** averaging period
- **Pollutant Type** will be selected as **Other**
- Option to read entire met data file selected
- Met data file must include variables for dry & wet deposition
- Specify risk emission phase data - for vapor, particle and particle-bound phase

3. Select **Risk | Risk Grid** from the menu. This will display the Multi-Tier Grid window. From this window, you can automatically setup the risk grid according to the **U.S. EPA OSW** protocols.

4. Select **Risk | Risk Emission Phase Data** from the menu. This will display the Gas & Particle Data screen in the Source Pathway. From this screen you should specify vapor phase, particle phase, and particle-bound phase data for all your sources, regardless of the output type options you selected in the Control Pathway.

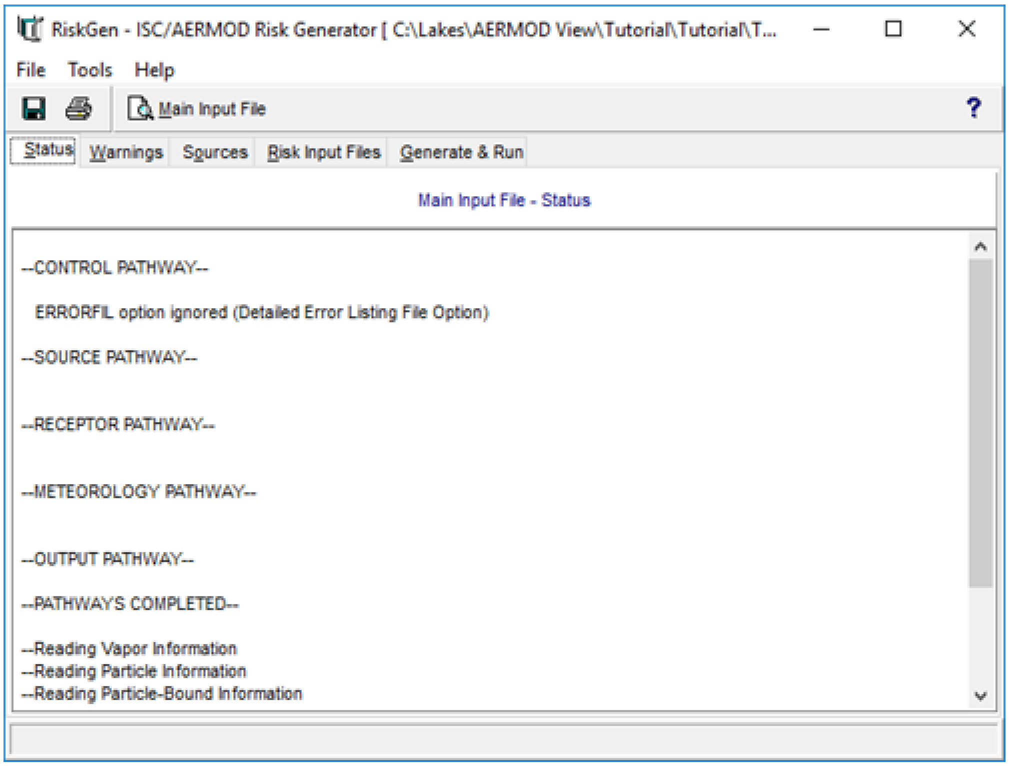
When conducting modeling with AERMOD, you will also need to enable Gas Deposition setting sin the Control Pathway.

See Also: The Risk Mode - Gas & Particle Data for more information.

5. Select **Risk | Run RiskGen** from the menu. This will display the **RiskGen** utility. This utility allows you to setup all input files you will need for your risk project.

RiskGen - Interface Overview

The components of the RiskGen dialog are shown and briefly described below:



1. **Menu bar:** Displays the menu names (see Menu Options).

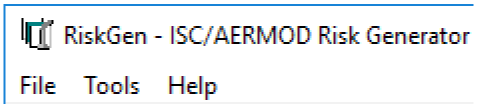
2. **Toolbar buttons:** Provide quick access to some menu commands.

3. **Tabs:** Provide access to the data inputs required to process the data.

4. **Input Area:** This area displays the content of each input tab.

RiskGen - Menu Options

The following menu options are available in **RiskGen**:



File

- **Save:** Saves the current project

- **Print:** Brings up a dialog with options to print the information currently displayed in the **RiskGen** window

- **Exit:** Closes RiskGen window

Tools

- **Browse...:** Opens a browsing window

- **Editor...:** Opens WordPad text editor

Help

- **Contents:** Opens on-line help file

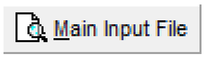
RiskGen - Menu Toolbar

The toolbar buttons are shortcuts to some of the menu commands.

Each button's function is explained below:

 - **Save** button. Saves the current RiskGen project file to your AERMOD View project file.

 - **Print** button. Displays the Print dialog, which will print the current contents of the RiskGen window.



- **Main Input File** button. Opens the ISCST3 input file in WordPad.

 - **Help** button. Opens on-line help file.

RiskGen - Getting Started

How to Start a Project:

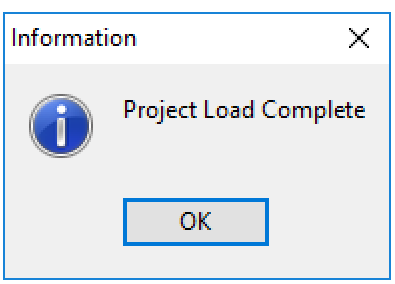
1. Create a project in AERMOD View. RiskGen is available for projects created in any of the models.

2. Select **Risk | Risk Mode (ON)** from the AERMOD View menu. Select **Yes** for the confirmation message. Complete all the required data for your risk project (e.g., source parameters, risk grid, met files, etc.).

3. Select **Risk | Risk Emission Phase Data**, from the AERMOD View menu, and complete the required Gas & Particle data for all your sources (vapor phase, particle phase, and particle-bound phase).

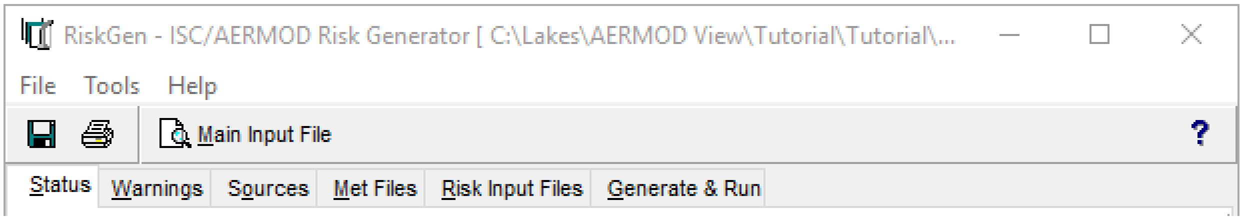
4. Check your project data for any missing information by selecting **Run | Details** from the AERMOD View menu. Complete any required information and then select **Risk | Run RiskGen** from the AERMOD View menu.

5. The **RiskGen** main window displays. Wait a few seconds as your **Master Risk File** loads and is checked. The **Master Risk File** is taken from the project that is currently open in AERMOD View. A message is displayed to advise you that your project load was completed.



6. Note that information about your **Master Risk File** is displayed in the **Status** tab (see Status Tab for more information on this topic).

7. The **RiskGen** main window contains 6 tabs. The contents of each one of these tabs can be found in the following sections:



- Status
- Warnings
- Sources
- Met Files
- Risk Input Files
- Generate & Run

RiskGen - Risk Mode - Gas & Particle Data

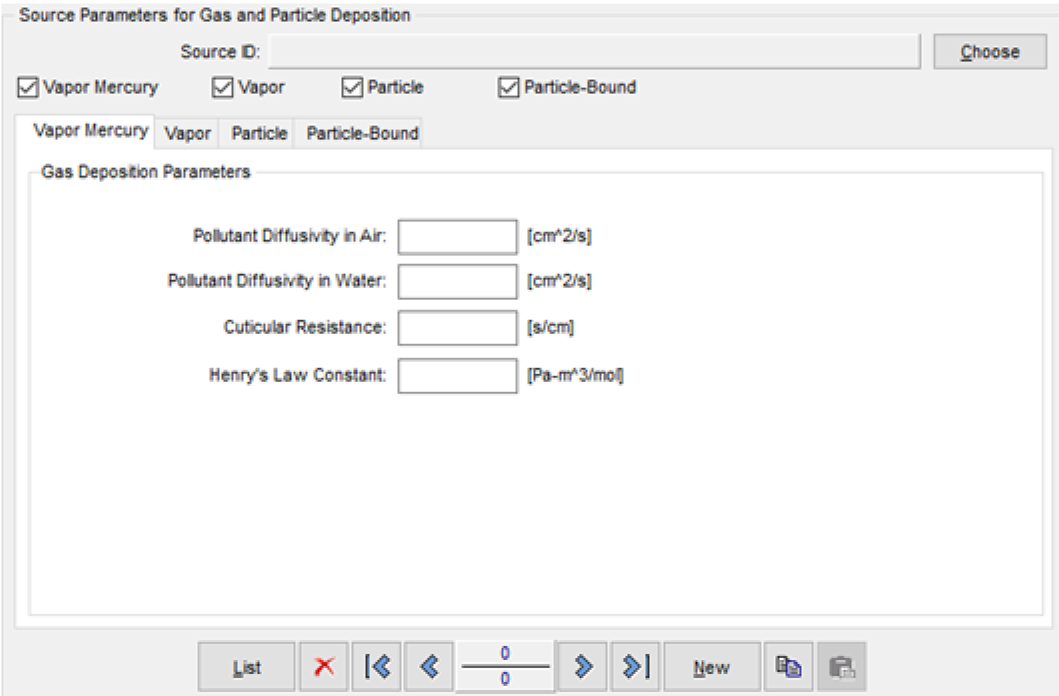
If you are using the **Risk Mode**, you must specify the vapor phase, particle phase, and particle-bound phase data for all your sources in the **Gas & Particle Data** screen of the **Source Pathway**.

The parameters requested in the **Gas & Particle Data** screen depend on the source phase being modeled. There are three tabs available for the Risk Mode, the options available depend on which model is being used:

AERMOD

The AERMOD model requires the following options to be specified:

Vapor Mercury and Vapor Phases



- **Pollutant Diffusivity in Air:** This is the diffusivity in air for the pollutant being modeled.
- **Pollutant Diffusivity in Water:** This is the diffusivity in water for the pollutant being modeled.
- **Cuticular Resistance:** This is the cuticular resistance to uptake by lipids for individual leaves.
- **Henry's Law Constant:** This is the Henry's Law constant.

The gas dry deposition parameters for the pollutant being modeled may be found in chemical engineering handbooks and various publications, such as the Air/Superfund National Technical Guidance Study Series (EPA, 1993).

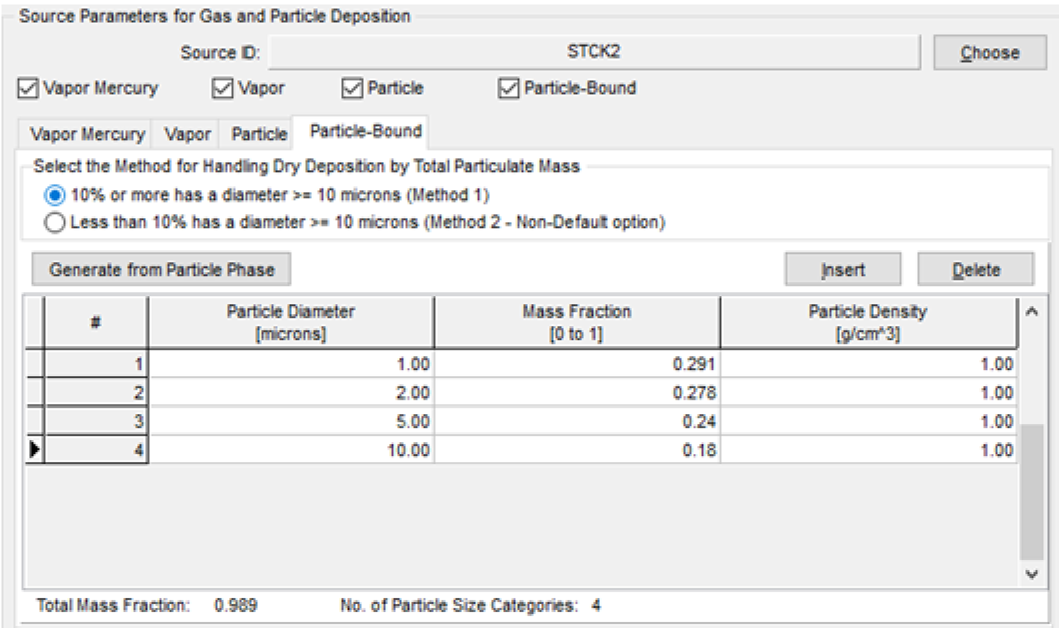
In order to execute the Vapor Mercury and Vapor phases, AERMOD requires gaseous deposition parameters. This information is input to the Control Pathway by enabling the Non-Default Gas Deposition option.

Particle & Particle-Bound Phases

The AERMOD model includes two methods for handling dry deposition of particulate emissions -

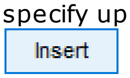
Method 1

This method is used when a significant fraction (greater than 10 percent) of the total particulate mass has a diameter of 10 microns or larger. The particle size distribution must be known reasonably well in order to use Method 1.



Particle Phase Options - Method 1

With Method 1 you must specify the following parameters:

- **No.:** Identifies the number of particle size categories specified for a particular source. You can specify up to a maximum of 20 categories. You may add a category by clicking on the 

button after you have entered information for the previous category.

- **Particle Diameter:** Enter in this column the particulate diameter in microns for each particle size category, up to a maximum of 20.

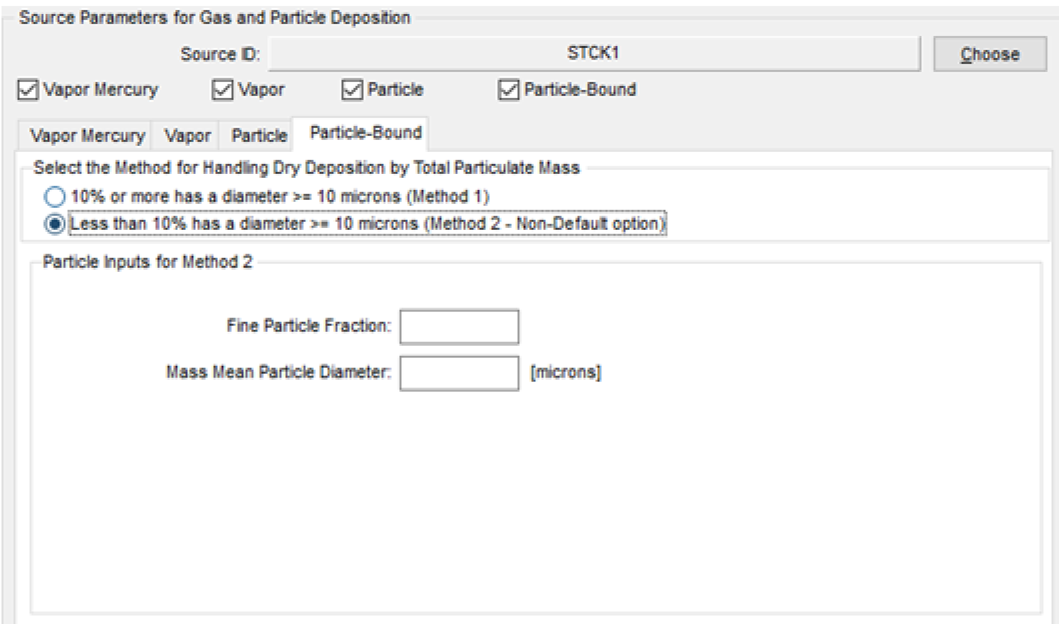
- **Mass Fraction:** In this column, you should define the mass fractions (between 0 and 1) for each of the categories you have defined. The mass fraction for each source must add up to 1.0 (within 2%). Note that the current total for the mass fraction is displayed on the bottom of the table.

- **Particle Density:** Here you can define the particle density in grams per cubic centimeter for each of the categories you have defined.

Method 2

This method is used when the particle size distribution is not well known and a small fraction (less than 10 percent of the mass) is in particles with a diameter of 10 microns or larger. The deposition velocity for Method 2 is given as the weighted average of the deposition velocity for the coarse mode, that is, greater than 2.5 microns but less than 10 microns in diameter.

> Method 2 is a non-default option and is available only if have selected the **Non-Default** radio button in the Dispersion Options window.



Particle Phase Options - Method 2

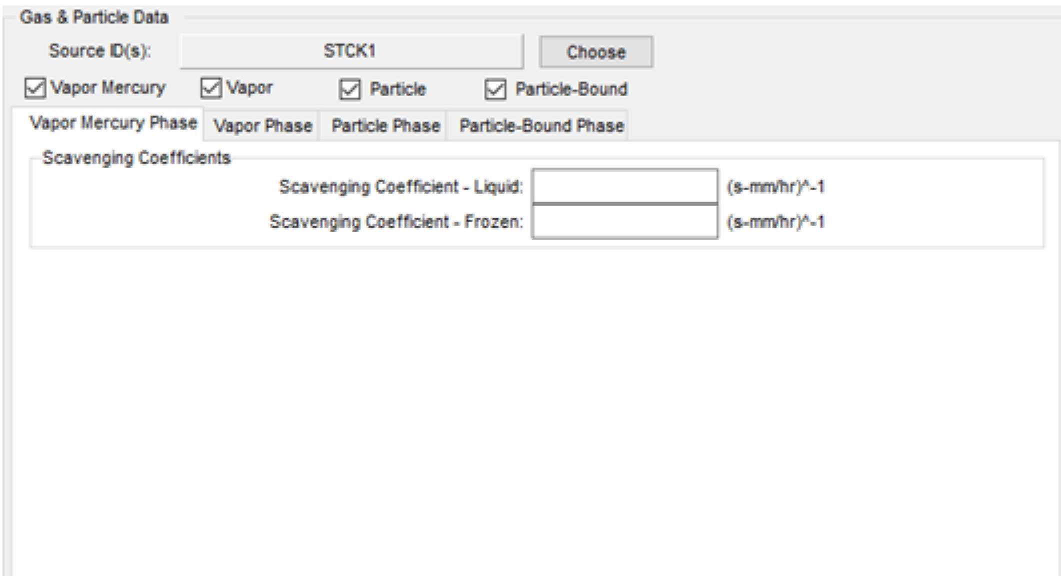
With Method 2 you must specify the following parameters:

- **Fine Particle Fraction:** Enter in this column the fraction of particle mass emitted in the fine mode, less than 2.5 microns.
- **Mass Mean Particle Diameter:** Specify here the representative mass mean particle diameters in microns.

ISCST3/ISC-PRIME

Vapor Mercury and Vapor Phases

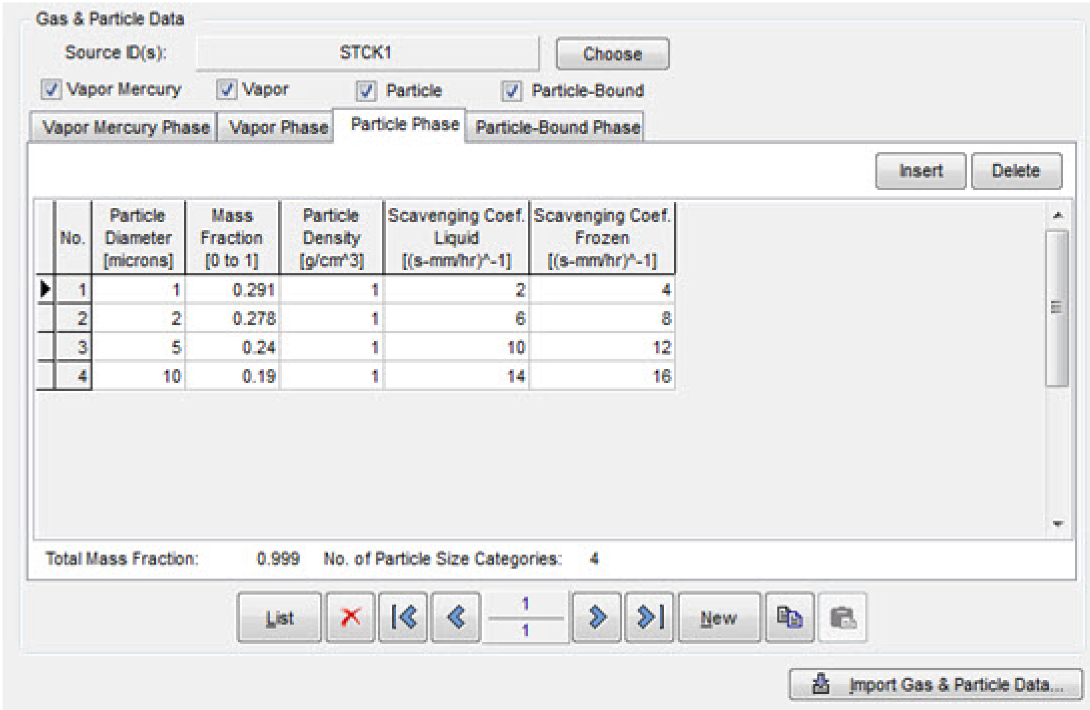
These two panels have the same input parameters. Here you must specify the liquid and frozen scavenging coefficients.

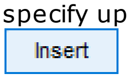


- **Scavenging Coefficient - Liquid:** Specify the scavenging coefficient for liquid precipitation.
- **Scavenging Coefficient - Frozen:** Specify the scavenging coefficient for frozen precipitation.

Particle Phase

You must specify the following information for the particle-phase of a source:



- **No.:** Identifies the number of particle size categories specified for a particular source. You can specify up to a maximum of 20 categories. You may add a category by clicking on the 

button after you have entered information for the previous category.

- **Particle Diameter:** Enter in this column the particulate diameter in microns for each particle size category, up to a maximum of 20. Note that the current total number of particle size categories is displayed on the bottom of the table.

- **Mass Fraction:** In this column, you should define the mass fractions (between 0 and 1) for each of the categories you have defined. The mass fraction for each source must add up to 1.0 (within 2%). Note that the current total for the mass fraction is displayed on the bottom of the table.

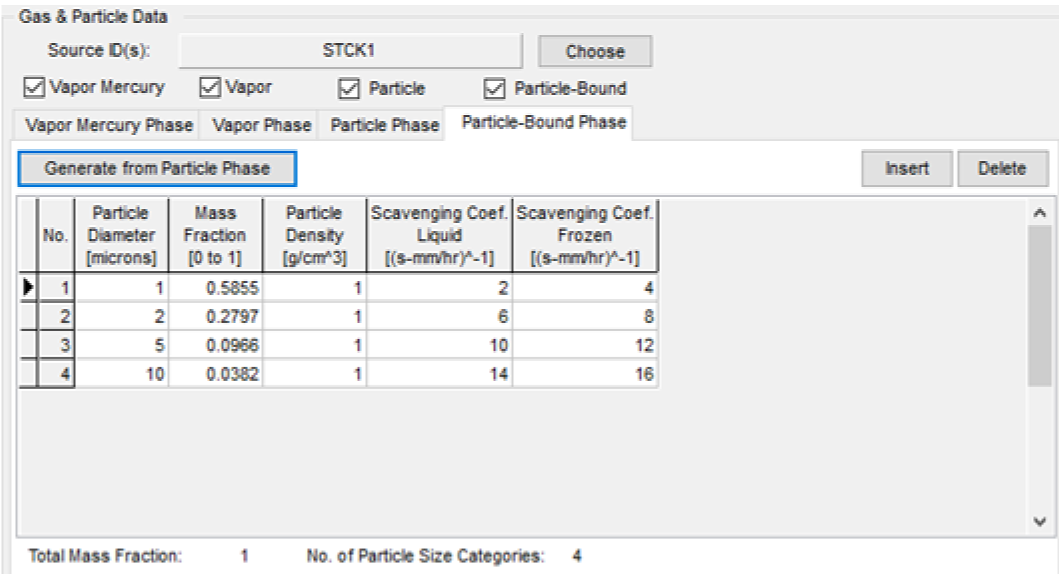
- **Particle Density:** Here you can define the particle density in grams per cubic centimeter for each of the categories you have defined.

- **Scavenging Coefficient - Liquid:** Enter in this column the particulate scavenging coefficient for liquid precipitation. The liquid scavenging coefficient should be entered for each particle size category.

- **Scavenging Coefficient - Frozen:** Enter in this column the particulate scavenging coefficient for frozen precipitation. The frozen scavenging coefficient should be entered for each particle size category.

Particle-Bound

The information to be specified in the **Particle-Bound Phase** tab is the same information as in the **Particle Phase** tab except for the **Mass Fraction**. You can press the Generate from Particle Phase button to copy the information from the **Particle Phase** tab.



In the Particle-Bound phase, the Mass Fraction is equivalent to the Fraction of Total Surface Area that is available. The **Fraction of Total Surface Area** is calculated in the following way:

Fraction of Total Surface Area = Pi / SUM

where:

Pi = (6 / Di) x Mass Fraction in Particle Phase

Di = particle diameter in the particle phase for particle category i

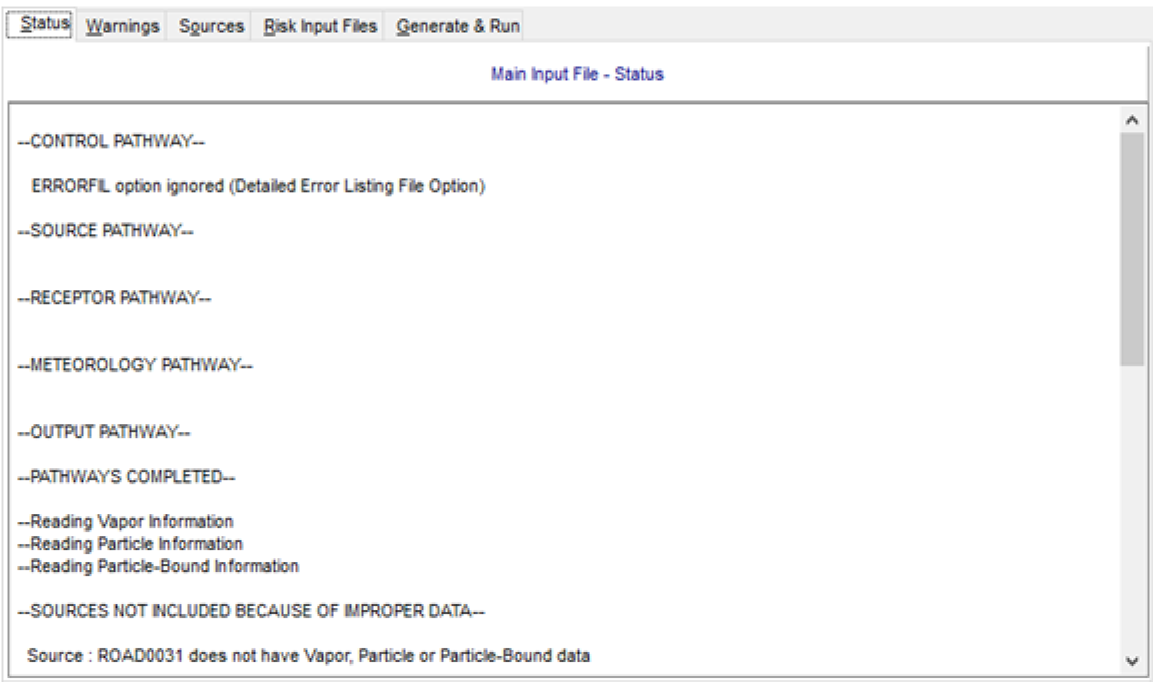
SUM = The sum of all Pi

Gas & Particle data can be specified for more than one source. The buttons at the bottom of the Gas & Particle Data screen called the Record Navigator buttons will help you manage information for multiple sources defined.

RiskGen - Status Tab

Once you open an ISCST3 Input File, RiskGen displays information about this input file in the **Status** tab. The information is related to whether options present in your ISCST3 Input File are considered or are supported when generating the necessary Risk Input Files. For easy reference, the information is grouped under the five Pathways:

- Control Pathway
- Source Pathway
- Receptor Pathway
- Meteorology Pathway
- Output Pathway

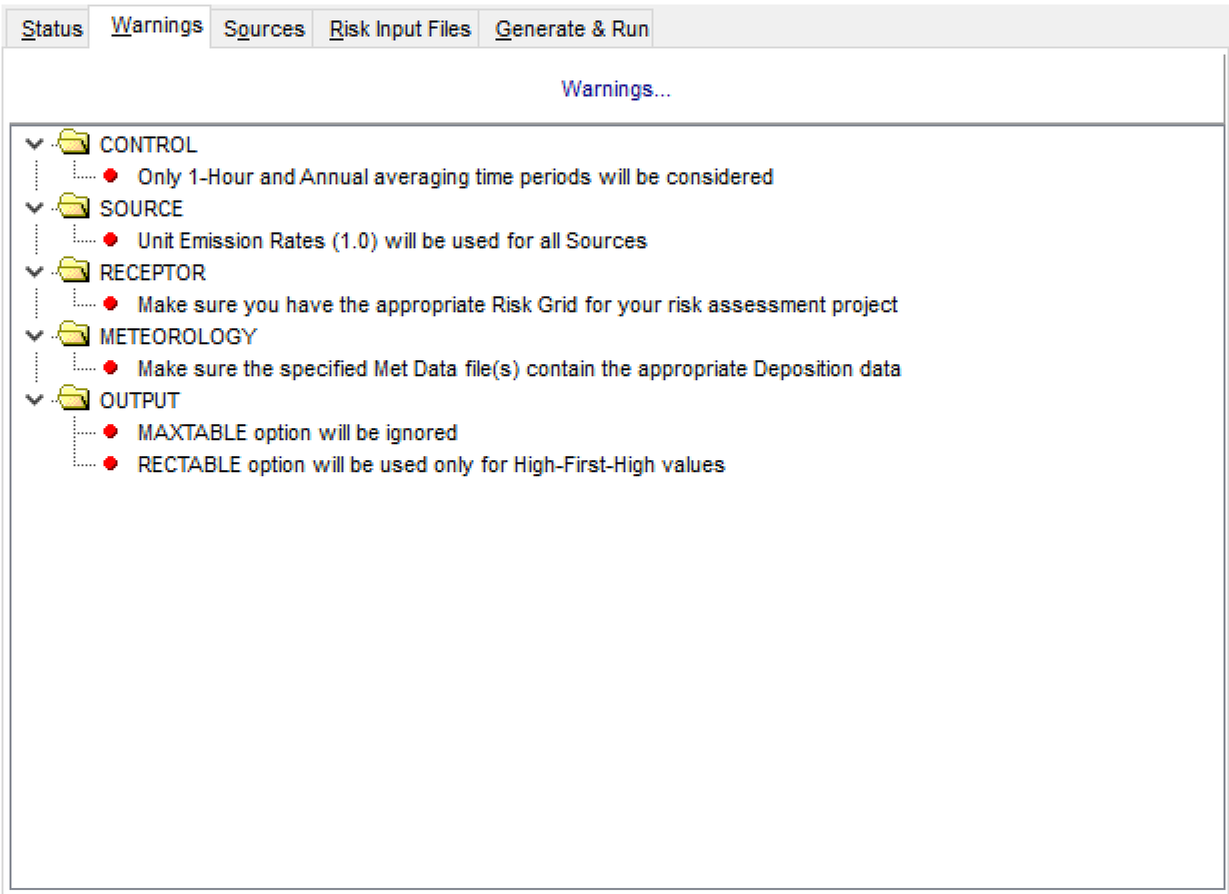


RiskGen - Warning Tab

The **Warning** tab displays information and options supported in RiskGen. For easy reference, the information is grouped under the five Pathways:

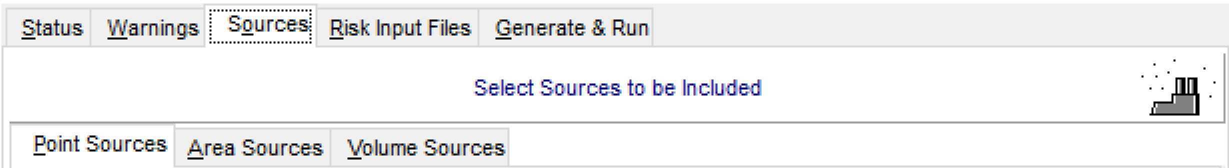
- Control Pathway
- Source Pathway

- Receptor Pathway
- Meteorology Pathway
- Output Pathway



RiskGen - Sources Tab

Three types of sources are currently being supported under the **1998 U.S. EPA Human Health Risk Assessment Protocol**: point, area, and volume sources. The information read from the Risk Input File for these three source types will be available in the following tabs:



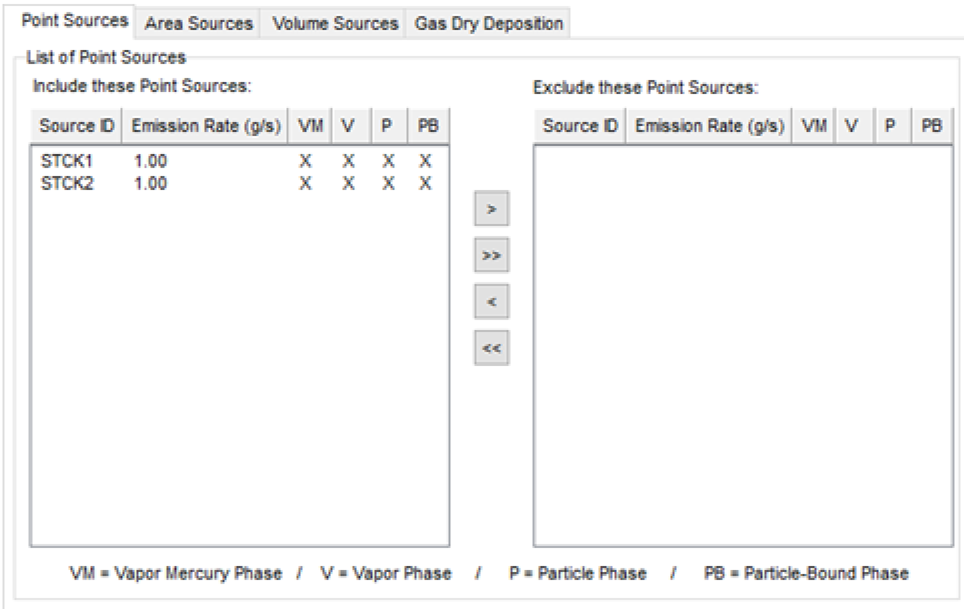
- Point Sources tab
- Area Sources tab
- Volume Sources tab
- Gas Dry Deposition

Three types of sources are currently being supported under the 1998 U.S. EPA Human Health Risk Assessment Protocol: **Point**, **Area**, and **Volume** sources.

Sources created in your AERMOD View project are checked to see if they have required vapor, particle, and particle-bound data. Sources with at least one of these parameters are then placed in the appropriate field in **RiskGen**. The **Status** tab provides a warning about any sources without vapor, particle, and/or particle-bound data.

RiskGen - Point Sources tab

All the point sources that are present in your project and that contain the required information for at least one emission phase (vapor mercury, vapor, particle, or particle-bound) are displayed in the **Include these Point Sources** list.



The **Exclude these Point Sources** list can be used to store the sources that, for any reason, should not be included into your final **Risk Input Files**. Use the buttons located between both lists to transfer sources from one list to the other.

For each source included in any one of these lists, the following will be displayed:

- **Source ID**: the source ID specified in your ISC project.
- **Emission Rate (g/s):** the unit emission rate of 1.0 g/s is used regardless of what you have specified in your ISC project. The HHRA Protocol specifies that a unit emission rate be used for the ISCST3 runs.
- **VM:** represents the **Vapor Mercury** emission phase. If RiskGen found the necessary information to set up a run for this phase, a mark is placed in this column.
- **V:** represents the **Vapor** emission phase. If RiskGen found the necessary information to set up a run for this phase, a mark is placed in this column.
- **P:** represents the **Particle** emission phase. If RiskGen found the necessary information to set up a run for this phase, a mark is placed in this column.
- **PB:** represents the **Particle-Bound** emission phase. If RiskGen found the necessary information to set up a run for this phase, a mark is placed in this column.

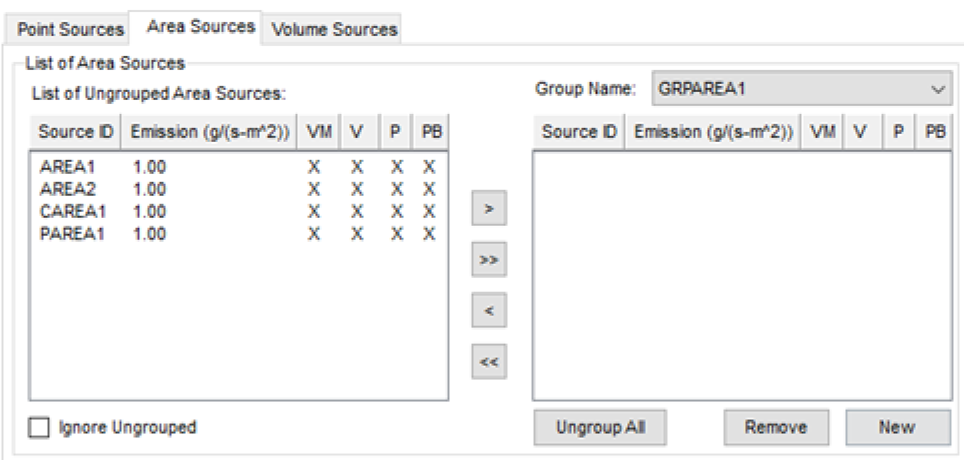
When checking for the emission phase information for a source, RiskGen tries to find the following information:

Emission	Particle Information	Scavenging Coeff. Gases	Scavenging Coeff. Particulates
Phase			
Vapor		✘	
Particle	✘		✘
Particle-Bound	✘		✘

> The emission phase information required by RiskGen is only available if you have specified emission data by selecting **Risk | Risk Emission Phase Data** from the AERMOD View menu.

RiskGen - Area Sources tab

Fugitive emission sources to be evaluated should be represented as either **Area** or **Volume** source types. All the **Area** sources that are present in your project and that contain the required information for at least one emission phase (vapor mercury, vapor, particle, or particle-bound) will be displayed in the **Area Sources** tab under the **List of Ungrouped Area Sources**.



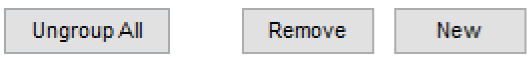
When setting up ISCST3 runs for a risk assessment following the U.S. EPA OSW HHRAP, one run is required for each emission source. Some fugitive emission sources are modeled as a combination of Area sources. In these cases, however, these sources should be included in a single run. RiskGen allows you to specify which Area sources must be combined in a single ISCST3 run by placing them into a group. A group name is automatically set up (e.g. GRPAREA1).

How to Group Area Sources:

1. In the **List of Ungrouped Area Sources**, select the sources you want group together (e.g., they represent the same fugitive source). To select more than one source from the list hold down the Shift key and select with the mouse pointer all the sources.

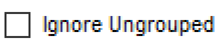
2. Click the single arrow button (>) to move the selected sources to the **Group** list. As you move area sources to the **Group** list, RiskGen redefines a new emission rate for each source This emission rate is proportionately allocated for each Area source based on the area of each individual source.

3. To define a new group press the New button. To ungroup an existing group press the **Remove** button. To ungroup all the existing groups, press the **Ungroup All** button.



4. For fugitive sources represented by only one area source, you can do one of the following: leave this source in the **List of Ungrouped Area Sources** or set up a group containing only this source.

5. If you want to exclude certain area sources from the final **Risk Input Files** than you have to leave these sources in the **List of Ungrouped Area Sources** and check the **Ignore Ungrouped** box.



The following information is available for Area sources contained in any one of the lists:

- **Source ID:** this is the source ID you specified in your ISC project.

- **Emission Rate (g/(s-m2)):** the unit emission rate of 1.0 g/s is used regardless of what you have specified in your ISC project. The HHRA Protocol specifies that a unit emission rate be used for the ISCST3 runs.

- **VM:** represents the **Vapor Mercury** emission phase. If RiskGen found the necessary information to set up a run for this phase, a mark is placed in this column.

- **V:** represents the **Vapor** emission phase. If RiskGen found the necessary information to set up a run for this phase, a mark is placed in this column.

- **P:** represents the **Particle** emission phase. If RiskGen found the necessary information to set up a run for this phase, a mark is placed in this column.

- **PB:** represents the **Particle-Bound** emission phase. If RiskGen found the necessary information to set up a run for this phase, a mark is placed in this column.

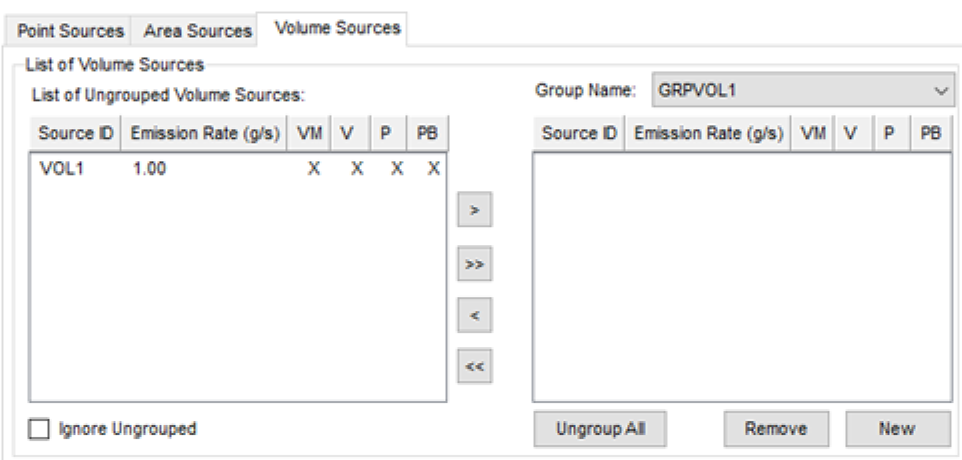
When checking for the emission phase information for a source, RiskGen tries to find the following information:

Emission	Particle Information	Scavenging Coeff. Gases	Scavenging Coeff. Particulates
Phase			
Vapor		✘	
Particle	✘		✘
Particle-Bound	✘		✘

The emission phase information required by RiskGen is only available if you have specified emission data by selecting **Risk | Risk Emission Phase Data** from the AERMOD View menu.

RiskGen - Volume Sources tab

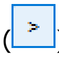
Fugitive emission sources to be evaluated should be represented as either **Area** or **Volume** source types. All the **Volume** sources that are present in your project and that contain the required information for at least one emission phase (vapor mercury, vapor, particle, or particle-bound) will be displayed in the **Volume Sources** tab under the **List of Ungrouped Volume Sources**.



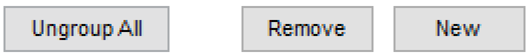
When setting up ISCST3 runs for a risk assessment following the U.S. EPA OSW HHRAP, one run is required for each emission source. Some fugitive emission sources are modeled as a combination of **Volume** sources. In these cases, however, the sources should be included in a single run. **RiskGen** allows you to specify which **Volume** sources must be combined in a single ISCST3 run.

How to Group Volume Sources:

1. In the **List of Ungrouped Volume Sources**, select the sources you want group together (e.g., they represent the same fugitive source). To select more than one source from the list hold down the Shift key and select with the mouse pointer all the sources.

2. Press the single arrow button () to move the selected sources to the **Group** list. As you move Volume sources to the **Group** list, **RiskGen** redefines a new emission rate for each source This emission rate is proportionately allocated for each **Volume** source based on the area of each individual source.

3. To define a new group, click the **New** button. To ungroup an existing group. click the **Remove** button. To ungroup all the existing groups, press the **Ungroup All** button.



4. For fugitive sources represented by only one Volume source, you can do one of the following: leave this source in the **List of Ungrouped Volume Sources** or set up a group containing only this source.

5. If you want to exclude certain Volume sources from the final Risk Input Files than you have to leave these sources in the **List of Ungrouped Volume Sources** and check the **Ignore Ungrouped** box.

The following information is available for Volume sources contained in any one of the lists:

- **Source ID:** this is the source ID you specified in your ISC project.

- **Emission Rate (g/(s-m2)):** the unit emission rate of 1.0 g/s is used regardless of what you have specified in your ISC project. The HHRA Protocol specifies that a unit emission rate be used for the ISCST3 runs.

- **VM:** represents the **Vapor Mercury** emission phase. If RiskGen found the necessary information to set up a run for this phase, a mark is placed in this column.

- **V:** represents the **Vapor** emission phase. If RiskGen found the necessary information to set up a run for this phase, a mark is placed in this column.

- **P:** represents the **Particle** emission phase. If RiskGen found the necessary information to set up a run for this phase, a mark is placed in this column.

- **PB:** represents the **Particle-Bound** emission phase. If RiskGen found the necessary information to set up a run for this phase, a mark is placed in this column.

When checking for the emission phase information for a source, **RiskGen** tries to find the following information:

Emission	Particle Information	Scavenging Coeff. Gases	Scavenging Coeff. Particulates
Phase			
Vapor		✘	
Particle	✘		✘
Particle-Bound	✘		✘

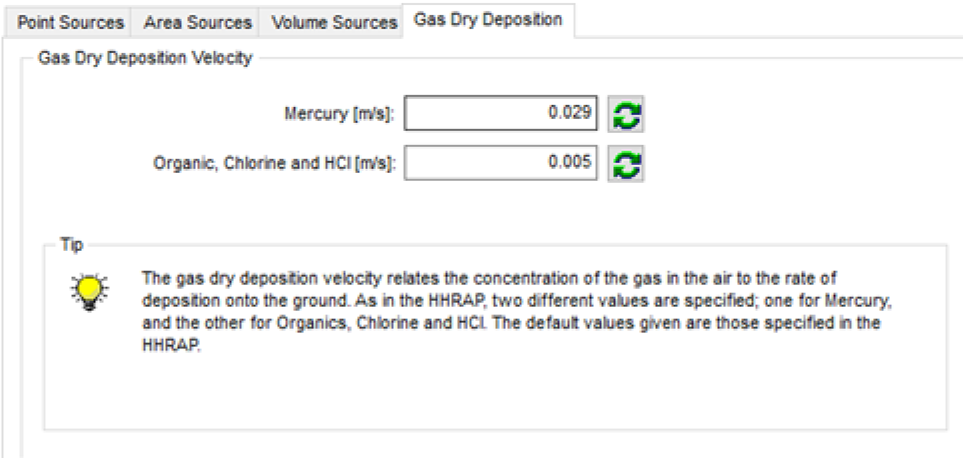
> The emission phase information required by RiskGen is only available if you have specified emission data by selecting **Risk | Risk Emission Phase Data** from the AERMOD View menu.

RiskGen - Gas Dry Deposition tab

The gas dry deposition velocity of Mercury and of Organics, Chlorine and HCI can be specified if they are different from the values specified in the **Human Health Risk Assessment Protocol (HHRAP)**. By default, **HHRAP** specifies the following gas dry deposition velocities:

- 0.029 meters per second (m/s) for Mercury
- 0.005 m/s for Organics, Chlorine, and HCI

RiskGen supplies these default values automatically.



- To change the gas dry deposition velocity of either mercury or organics, chlorine and HCI, enter the value in the appropriate field.

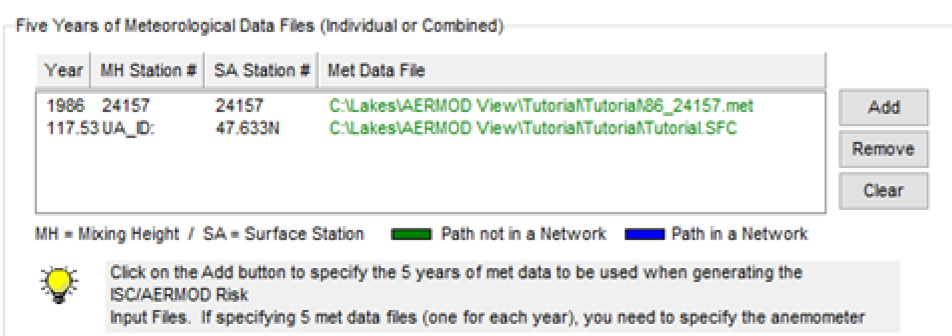
- To revert to the default values specified in the **HHRAP**, click the revert button ( ) to the right of the appropriate field.

RiskGen - Met Files Tab

In the **Met Files** tab, you specify the 5 years of meteorological data to be used for your risk assessment project. This is specified in the format of meteorological data files specified in the AERMOD View Master Risk Project File. When you specify the file(s), RiskGen places them in the **Met Data Files** list. In RiskGen, you can specify one of the following:

- **Combined:** You can specify only one met data file if it contains the combined 5-years of met data (each year must be complete with a full year of data).

- **Individual:** You can also specify five individual met data files, one for each year. In this case, you will need to add each file to the **Met Data Files** list. To add one or more met data files press the **Add** button (see **Anemometer Height** below).



Met Data Files list

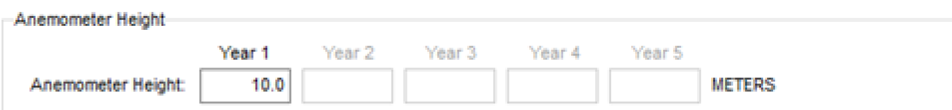
In the **Met Data Files** list the following information is available for each specified met data file:

- **Year:** This is the year for the specified met data file.
- **MH Station:** This is the mixing height station number for the specified met data file.
- **SA Station:** This is the surface air station number for the specified met data file.
- **Met Data File:** This is the location and name for the specified met data file.

> You can view the contents of any specified met data file using Windows WordPad by selecting the met data file from the list and double-clicking it.

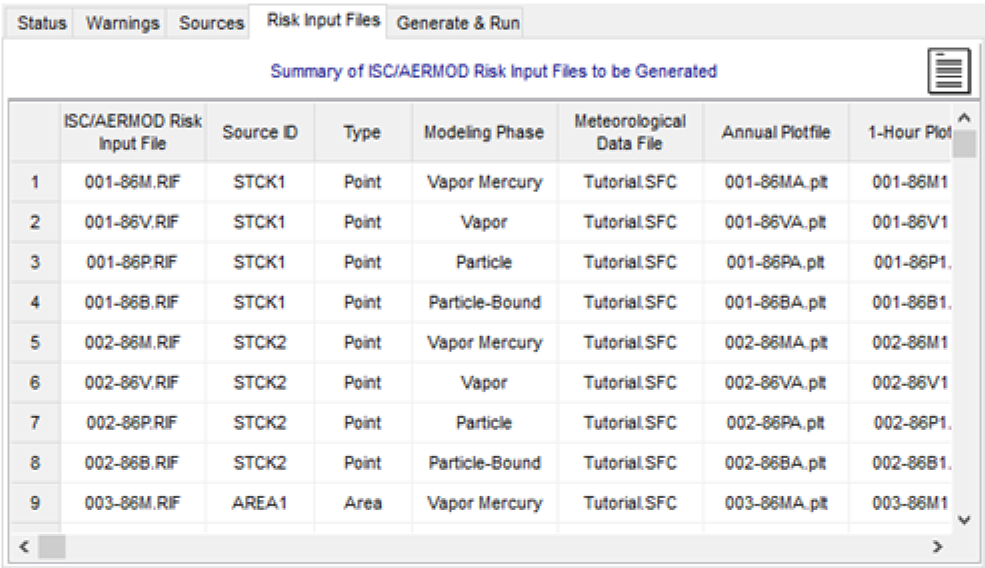
Anemometer Height

For sites where the anemometer height changed during the 5-year period, you should use the five years of meteorological data separately so you can specify the correct anemometer height for each year.



RiskGen - Risk Input File

In the **Risk Input Files** tab, a table details the ISC/AERMOD Risk Input Files that RiskGen will generate. Each column is explained below.



	ISC/AERMOD Risk Input File	Source ID	Type	Modeling Phase	Meteorological Data File	Annual Plotfile	1-Hour Plot
1	001-86M.RIF	STCK1	Point	Vapor Mercury	Tutorial.SFC	001-86MA.plt	001-86M1
2	001-86V.RIF	STCK1	Point	Vapor	Tutorial.SFC	001-86VA.plt	001-86V1
3	001-86P.RIF	STCK1	Point	Particle	Tutorial.SFC	001-86PA.plt	001-86P1.
4	001-86B.RIF	STCK1	Point	Particle-Bound	Tutorial.SFC	001-86BA.plt	001-86B1.
5	002-86M.RIF	STCK2	Point	Vapor Mercury	Tutorial.SFC	002-86MA.plt	002-86M1
6	002-86V.RIF	STCK2	Point	Vapor	Tutorial.SFC	002-86VA.plt	002-86V1
7	002-86P.RIF	STCK2	Point	Particle	Tutorial.SFC	002-86PA.plt	002-86P1.
8	002-86B.RIF	STCK2	Point	Particle-Bound	Tutorial.SFC	002-86BA.plt	002-86B1.
9	003-86M.RIF	AREA1	Area	Vapor Mercury	Tutorial.SFC	003-86MA.plt	003-86M1

- **ISC/AERMOD Risk Input File:** This column displays the name of the Risk Input File to be created (see Name Convention - Risk Input Files).

- **Source ID:** This column displays the source being specified for the Risk Input File. Each Risk Input File must contain information for a single source. Fugitive emissions are modeled either as "area" or "volume" source types. In many cases, to model these emissions sources, it is required that the area be subdivided into more than one area or volume source. In those cases, the group of sources that make up one fugitive emission should be included into the same Risk Input File.

- **Type:** This column displays the source type (Point, Volume, or Area) being specified for the Risk Input File.

- **Modeling Phase:** This column displays the modeling phase being considered for the Risk Input File. Three sets of Risk Input Files are required for each source (Vapor phase, Particle phase, and Particle-Bound phase).

- **Meteorological Data File:** This column displays the met data file being used for each Risk Input File. If 5 years of meteorological data were specified in individual files instead of in a combined 5-year met data file, then for each source, modeling phase, and meteorological data file, an ISC Risk Input File must be created. For example, for each source, 3 Risk Input Files are created (1 source x 3 modeling phases x 1 met data file = 3 input files). On the other hand, 15 Risk Input Files are created if using five data files for the 5 years of meteorological data (1 source x 3 modeling phases x 5 met data files = 15 input files).

- **Annual Plotfile:** The annual average plotfile must be specified for each Risk Input File (see Name Convention - Annual Plotfiles).

- **1-Hour Plotfile:** The 1-hour average plotfile must be specified for each Risk Input File (see Name Convention - 1-Hour Plotfiles).

Name Convention - ISC/AERMOD Risk Input Files

The following name convention is applied to all ISC/AERMOD Risk Input Files produced by RiskGen:

Example: 001-88V.RIF

- **Characters 1 to 3:** indicate the source number; sources will be numbered from 001 to 999 in the following order: Point sources, Area sources, and Volume sources. The numbering scheme will follow the order that these sources appear in the source list (see Sources Tab).
- **Character 4:** hyphen (-). For projects that contain between 999 and 9999 total sources, the 4th character will be part of the source number, rather than a hyphen.
- **Characters 5 & 6:** indicate the last two digits of the year for the met data (e.g., 88 for 1988). For a combined 5 year met data file the two digits will be for the first year of met data.
- **Character 7:** indicates the modeling phase (V=Vapor Phase, P=Particle Phase, and B=Particle-Bound Phase).
- **File Extension:** RIF (Risk Input File).

Name Convention - Annual Plotfiles

The following name convention is applied to all Annual Plotfiles produced by RiskGen:

Example: 001-88VA.PLT

- **Characters 1 to 3:** indicate the source number. Sources will be numbered from 001 to 999 in the following order: Point sources, Area sources, and Volume sources. The numbering scheme will follow the order that these sources appear in the source list (see Sources Tab).
- **Character 4:** hyphen (-). For projects that contain between 999 and 9999 total sources, the 4th character will be part of the source number, rather than a hyphen.
- **Characters 5 & 6:** indicate the last two digits of the year for the met data (e.g., 88 for 1988). For a combined 5 year met data file the two digits will be for the first year of met data.
- **Character 7:** indicates the modeling phase (V=Vapor Phase, P=Particle Phase, and B=Particle-Bound Phase).
- **Character 8:** indicates the annual average, and is always the character "A".
- **File Extension:** PLT (Plotfile).

Name Convention - 1-Hour Plotfiles

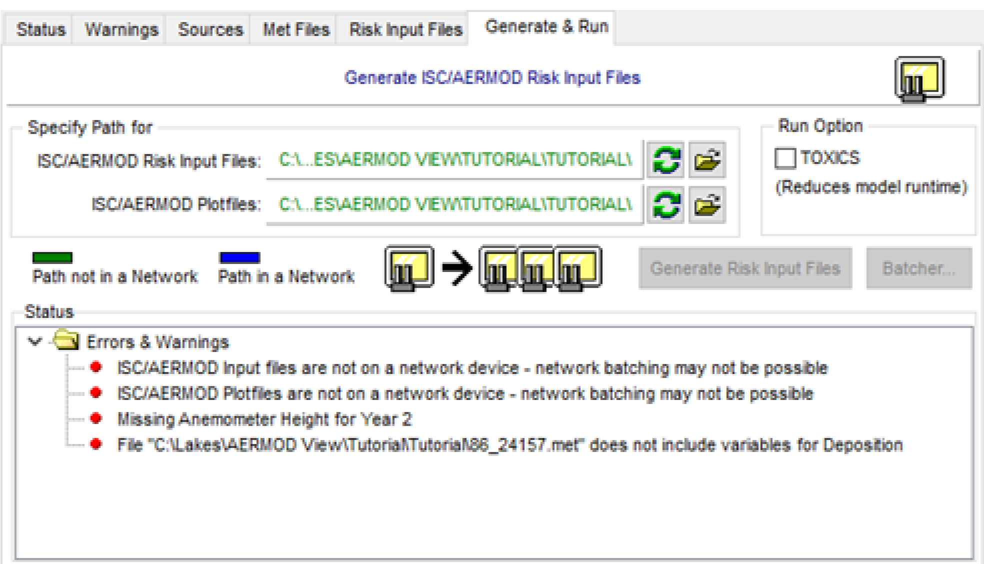
The following name convention is applied to all 1-Hour Plotfiles produced by RiskGen:

Example: 001-88V1.PLT

- **Characters 1 to 3:** indicate the source number. Sources will be numbered from 001 to 999 in the following order: Point sources, Area sources, and Volume sources. The numbering scheme will follow the order that these sources appear in the source list (see Sources Tab).
- **Character 4:** hyphen (-). For projects that contain between 999 and 9999 total sources, the 4th character will be part of the source number, rather than a hyphen.
- **Characters 5 & 6:** indicates the last two digits of the year for the met data (e.g., 88 for 1988). For a combined 5 year met data file the two digits will be for the first year of met data.
- **Character 7:** indicates the modeling phase (V=Vapor Phase, P=Particle Phase, and B=Particle-Bound Phase).
- **Character 8:** indicates the 1-hour average, and is always the character "1".
- **File Extension:** PLT (Plotfile).

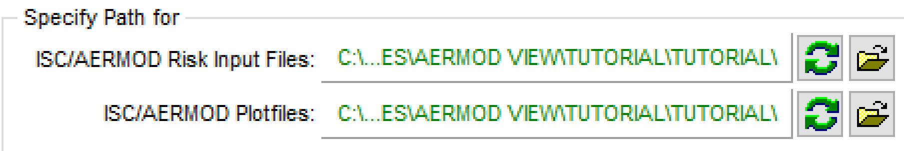
RiskGen - Generate & Run Tab

The **Generate** tab contains the following options:



Specify Path for

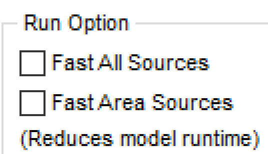
You must identify the location where you want the **ISC/AERMOD Risk Input Files** and **ISC/AERMOD Plotfiles** to be placed after they are generated. If you intend to run your **ISC/AERMOD Risk Input Files** from a network, you should specify a network path. By default, ISC/AERMOD Risk Generator will use the same path as your Risk Master File.



> For example, you want to use two machines on the network to run your **Risk Input Files**, one file from John's machine and one from Mary's. John originally set up the **Risk Input Files** using **RiskGen**. He specified a path for the generated **Input Files** and **Plotfiles** on a shared resources drive (e.g., z:). When Mary sets up the **AERMOD Batcher** to run some of the **Risk Input Files**, the path of these files will be automatically recognized. If you do not know how to create a shared resources drive, contact your network manager.

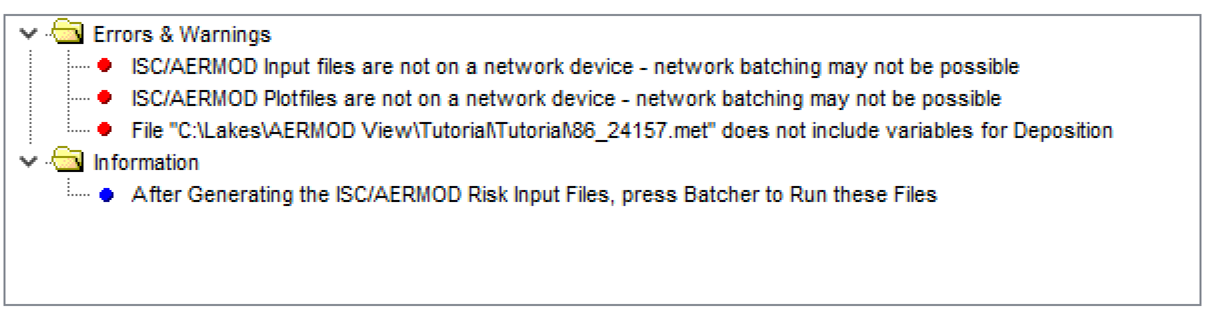
Run Option

The **Run Option** reduces model runtime; it is optional.



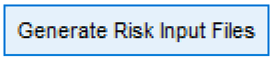
Status

The **Status** displays information on the status of your project, and warns you if any information is missing.



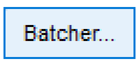
Generate Input Files

When all requirements are met, the **Generate Input Files** button is enabled. Click the button to generate all the necessary **Risk Input Files**.



Batcher

The Batcher button starts the AERMOD Batcher, which allows you to specify Input Files to be run in an unattended mode. The Batcher button is enabled only after the necessary Risk Input files are generated. When the AERMOD Batcher is opens, the **Risk Input Files** are automatically set up and can be run immediately.

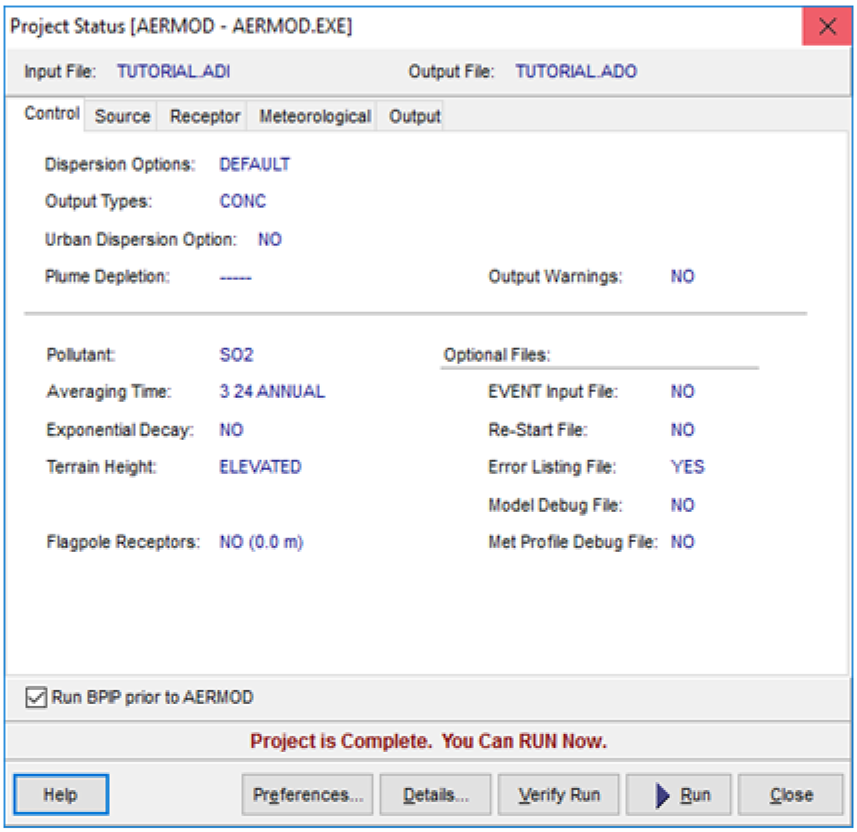


Running

To run an AERMOD View project, you must first determine that you have a complete project, and then run the project.

Project Status

The **Project Status** dialog provides you with a concise way of viewing all the options selected in your project. This dialog can be displayed by clicking on the Run menu toolbar button or by selecting **Run | Run Model...** from the menu.



Project Status dialog

The **Project Status** dialog contains a summary of the inputs and options selected for the current project.

On the top section of the dialog, the name of the input file and output file for the selected model is displayed. The **Project Status** dialog is composed of six tabs: **Control**, **Source**, **Receptor**, **Meteorology**, **Terrain Grid**, and **Output**. Each tab summarizes the options you have selected for the specific pathway for the current project run. Options not defined or not being used will appear as a dashed line (----).

The **Run BPIP prior to running AERMOD** option allows you to run BPIP automatically if buildings are present. This provides a means to avoid using out of date building wash data. This option can be disabled (unchecked) if desired.

At the bottom of the dialog, a message identifies if your project is complete or not. It will also display a message if the project is complete, but with warnings. If your project is incomplete, you should go to the Details dialog to discern the missing information.

The following buttons are available in the **Project Status** dialog:

Help Displays the **Help** contents for the **Project Status** dialog.

Preferences... Click this button to open the AERMAP screen of the Preferences dialog.

Details... Displays the Details dialog where you can view any missing project run information.

Verify Run Click this button to run the U.S. EPA model in the DO NOT RUN mode. This mode allows you to process the input file through the U.S. EPA model just to check for any errors in the input data. This option can save you time if a large run is performed with some incorrect input data.

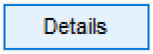
Run Click here to run the U.S. EPA model (ISCST3, ISC-PRIME, or AERMOD).

> If you added or moved any sources, buildings, or receptors at your site since you last ran AERMAP, a message will be displayed urging you to re-run it. AERMAP will calculate any changes in elevation to ensure that the emission values calculated by the model are accurate.

Close Closes the **Project Status** dialog.

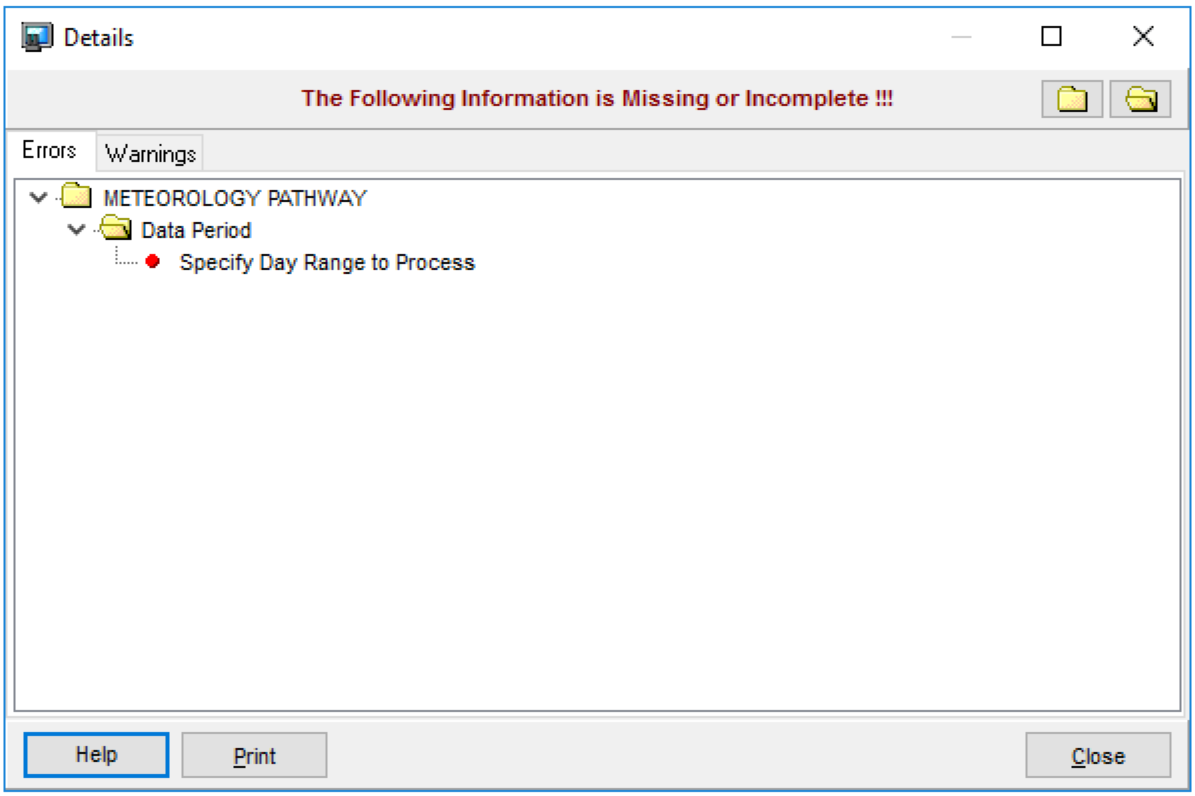
> We strongly recommend installing your software and operating all projects from a single local hard drive. Since this is database driven software, there is consistently a large exchange of information from the program to the project files. Running the software or projects on a network can significantly slow down the run time of the software, and can slow down your network for other network users.

Details

The **Details** dialog is displayed by selecting **Run | Details** from the menu, or by pressing the 

button from the Project Status dialog.

Before running the model, you can check the input to make sure that nothing is missing or incomplete. An example of missing info would be if somehow no sources were specified in the project. An example of incomplete information would be if elevated terrain was specified in the project, but no hill heights were specified for discrete receptors. The **Details** dialog contains a list of all missing information or errors for the current project. AERMOD View will detect most of the data that is missing or incorrect, however, it is advisable that you always verify your project input data. The list is subdivided into pathways, with all the missing inputs for each pathway listed under each pathway heading.



Details dialog

If there is any missing or incomplete data, you must fix it before running the project.

How to Check Project Completeness

1. Select **Run | Details** from the menu.

2. The **Details** dialog opens with a status message displayed:

- If the project is complete, the dialog displays **ALL Data Completed!**
- If the project is incomplete, the dialog displays **The Following Information is Missing or Incomplete !!!**

3. If there is missing or incomplete data in the project, you can print the **Details** report, in order to have a list of items to fix, by clicking the **Print** button.

4. Close the dialog:

   - If the project is complete, you can run the project.
   - If the project is incomplete, fix the missing or incomplete data, and recheck the project.

Running the U.S. EPA Model

Once you have ascertained there is no information missing or incomplete using the **Project Status** dialog you may view the **Input Files** and run the model.

How to View the Input Files

1. Select **Run | View Input File** and then select the model you wish to run. The summary of all input information will open in a text editor.

2. Go through the information to see if is is correct. The **Project Status** dialog will provide a warning about missing information, but it may not catch wrong values.

3. Close the text editor when you are done reviewing the inputs.

How to Run the Model

1. Click the **Run** 

button and select the model you wish to run or click the Run menu item and select the model you wish to run

2. The Project Status dialog will load. If all the information is correct, the message **Project is Complete. You can RUN Now.** will be displayed.

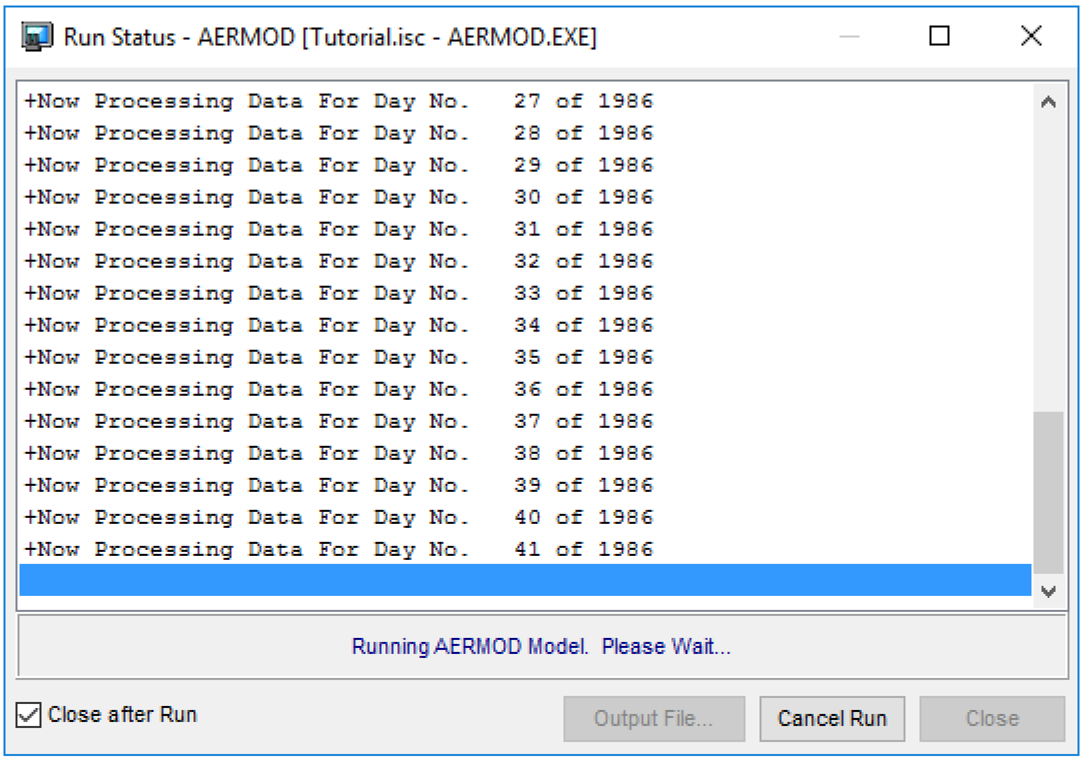
> If there is any missing or incomplete information you will not be able to run the model. Please refer to Project Status and Details sections for more information.

You may click Verify Run button to verify the input files. This option will not run the model itself, but will only check the input files for presence of errors.

3. Click **Run** Run button to run the model

4. If you have previously run the model, a message informing you that the current run will clear existing contours and delete old plot files will be displayed (unless you previously checked the **Do not show this message again** box). Click **Yes** in this message.

5. The model will run and **Run Status** dialog will show progress. Wait for the run to finish, or, if you wish to end the run early, click the **Cancel Run** Cancel Run button.



Run Status dialog

The Cancel Run button will be disabled if AERMOD View detects that the U.S. EPA model executable version cannot be cancelled during the run.

6. If the run finishes successfully the message stating so will be displayed. Click **OK** to close the message.

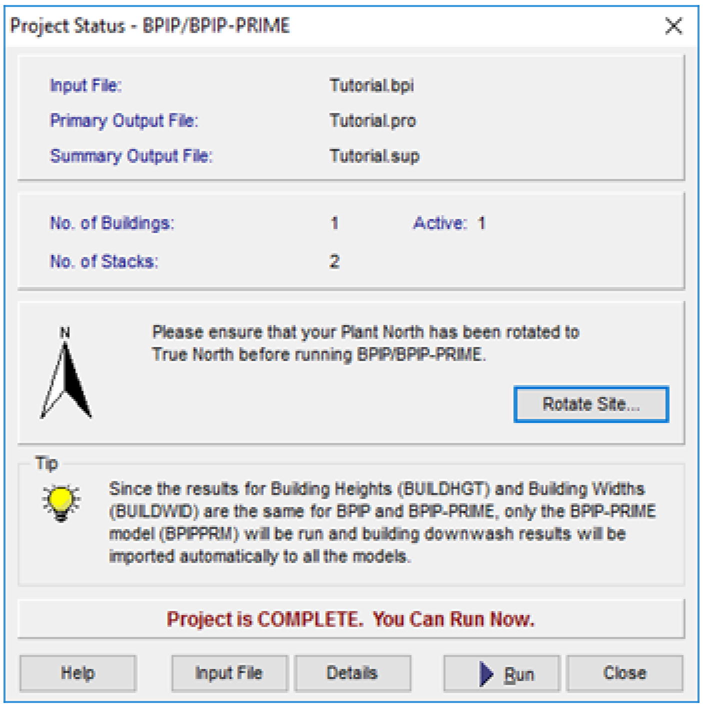
7. Close the **Run Status** dialog if it does not close automatically after the run.

You can set the **Run Status** dialog to close automatically after a run by checking the box **Close after Run** at the bottom of the dialog

8. The calculated contours will be displayed.

BPIP/BPIP-Prime Project Status

The **Project Status** dialog provides you with a concise way of viewing all the options selected for your BPIP run. The **Project Status** dialog can be displayed by selecting **Run | Status | BPIP...** from the menu.



Project Status - BPIP/BPIP-PRIME dialog

The **Project Status - BPIP/BPIP-PRIME** dialog contains a summary of the inputs and options selected for the BPIP run. The following information is displayed in this dialog:

- The first panel of the **Project Status - BPIP/BPIP-PRIME** dialog displays the name of the input, primary and the summary output files, for the current project.
- The second panel displays the number of buildings and stacks you have defined for the current run and the number of active buildings in your project. You must have at least one building and one stack to run the EPA BPIP model.

- The third panel is a reminder that the parameters of your project must be specified to True North orientation before running **BPIP/BPIP-PRIME**. If necessary, the Rotate Site tool allows you to perform this function.

- On the bottom panel of the dialog, a message identifies if your project is complete or not. If your project is incomplete, you should go to the Details dialog to discern the missing information.

> With the release of BPIPPRM (dated 04274) on December 9, 2004, flags to either run BPIP with prime algorithms or without prime algorithms were introduced (P = flag for PRIME and ST or NP = flag for non-PRIME). When you press the **Run** button, the BPIP model (BPIPPRM.EXE) will be run only once with the Flag "P" for PRIME. Because the results for BPIP and BPIP-PRIME are the same for the parameters BUILDHGT (Building Height) and BUILDWID (Projected Building Width), there is no need to run BPIP twice and, therefore, only BPIP-PRIME is run. AERMOD View will extract the necessary building downwash results from the BPIP-PRIME output file for each model.
> **SO BUILDHGT** (BPIP and BPIP-PRIME parameters – Necessary for all models)
> **SO BUILDWID** (BPIP and BPIP-PRIME parameters – Necessary for all models)
> **SO BUILDLEN** (BPIP-PRIME parameter – Necessary for ISC-PRIME and AERMOD)
> **SO XBADJ** (BPIP-PRIME parameter – Necessary for ISC-PRIME and AERMOD)
> **SO YBADJ** (BPIP-PRIME parameter – Necessary for ISC-PRIME and AERMOD)

The following buttons are available in this dialog:

Help Displays the Help contents for the Project Status - BPIP/BPIP-PRIME dialog.

Input File Click this button to view the BPIP Input File.

Details Displays the Details dialog where you can view any missing BPIP run information.

Run Click here to run the EPA BPIP model.

Close Closes the Project Status - BPIP/BPIP-PRIME dialog.

Running the U.S. EPA EVENT Model

The **EVENT** model is specifically targeted to perform short-term (less than 24 hour) analysis of source contributions for specific limiting events. A good example of the use of the **EVENT** model would be to examine possible violations of an air quality standard from a source contribution over a short-term time period.

This is an optional process. In order to use this model select the EVENT Input File option from the Control Pathway.

The **EVENT** model works in conjunction with all the models (**ISCST3**, **ISC-PRIME**, and **AERMOD**). The models actually generate the input file needed to run the **EVENT** model as the input file for the **EVENT** model is based on the options selected for the model.

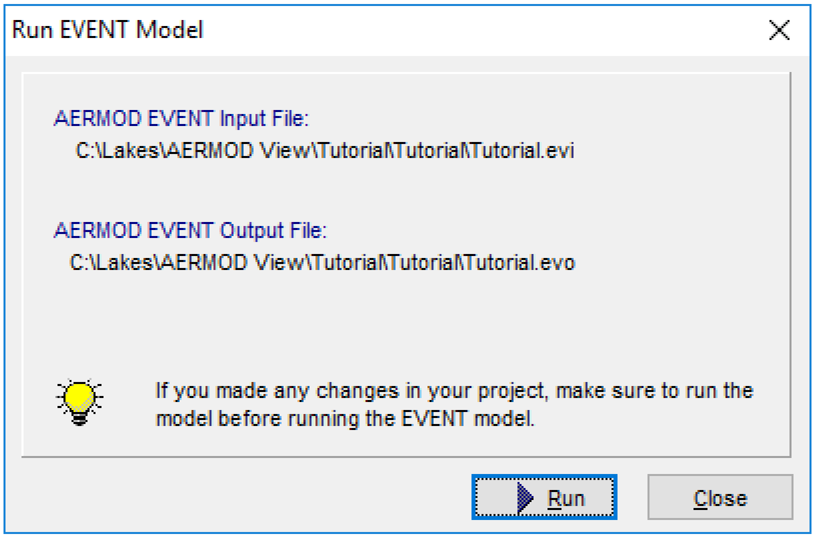
The results contained in the EVENT output file will be affected by the selections made in the Tabular Outputs window, such as the highest values. You can also specify threshold violation files that contain all the information on occurrences where concentrations of a substance violated the threshold limit for the particular substance. These files are optional and are specified in the Threshold Violation Files window.

> The EVENT model may produce very large files for runs involving a large number of receptors if a significant percentage of the results exceed the threshold value. These files can get extremely large in certain circumstances, even up to several hundred megabytes. Therefore, please be sure you have adequate space on your hard drive.

How to Run the U.S. EPA EVENT Model:

1. Select the EVENT Input File option from the Control Pathway.
2. Select **Run | Run Model...** from the menu. After the model run has finished successfully, go to the menu and select **Run | Run EVENT**.
3. The **Run EVENT Model** dialog will appear giving you status information. If all the necessary options are satisfied, press the 

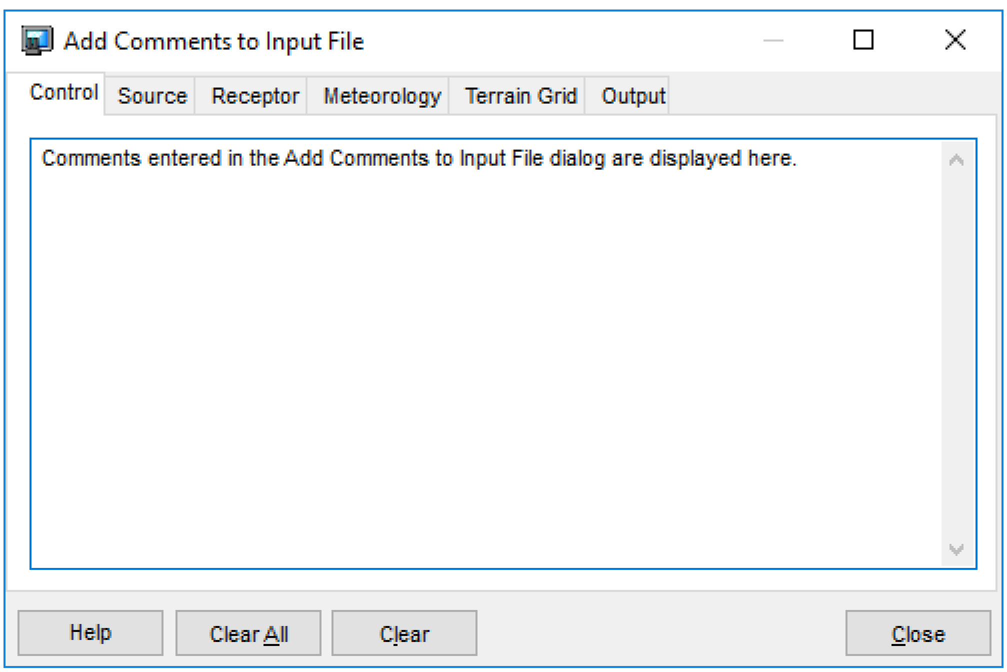
button to run the EVENT model.
4. Once the model run is complete, another dialog will appear asking you if you want to see the EVENT output file.



Run EVENT model dialog

Add Comments to Input File

You have access to the **Add Comments to Input File** dialog by selecting **Run | Add Comments...** from the menu. This dialog allows you to comments to the input file for each pathway.



Add Comments to Input File dialog

Each tab represents a pathway in the input file. Select the tab representing the pathway for which you wish to enter comments. The comments that you type are added to the input file at the start of each pathway section.

```
**
*****************************************
**
** AERMOD Input Produced by:
** ISC-AERMOD View Ver.
** Lakes Environmental Software Inc.
** Date: 5/16/2005
** File: C:\ISC\Tutorial\tutorial.ADI
**
*****************************************
**
**
*****************************************
** AERMOD Control Pathway
*****************************************
**
**
** Comments entered in the Add Comments to Input File dialog are
** displayed here.
**
CO STARTING
   TITLEONE C:\ISCView4\Tutorial\tutorial.isc
   MODELOPT DFAULT CONC
   AVERTIME 1
   URBANOPT 0 Urban_Area_Name 1.00000
   POLLUTID SO2
   RUNORNOT RUN
CO FINISHED
**
```

The following buttons are found in this dialog -

Help Click this button to load the help file at this page.

Clear All Click this button to clear comments for all pathways.

Clear Click this button to clear comments for the displayed pathway.

Close Click this button to close the dialog.

Use the Multi-Chemical Run Utility

The **Multi-Chemical Utility** is designed to allow each source in a given project to emit multiple chemicals with each at a specified emission rate, allowing you to model the contributions from each pollutant quickly and concisely. These chemicals, and emission rates, can be different for each source. This feature is very powerful, as the alternative is to run separate projects for each different emitted chemical.

The **Multi-Chemical Utility** works by creating individual batch/input files for each source, running them through the specified model (**ISC**, **ISC-PRIME** or **AERMOD**), and combining them into a single result to be viewed. The output is the contribution from each individual pollutant for all the sources. It should also be noted that output files are created for each source, with unitized emission rates, and that chemical specific output files are not created.

While it is very useful and can save a lot of time, the **Multi-Chemical Utility** does have limitations, namely that it does not support all of the features of the ISC and AERMOD models.

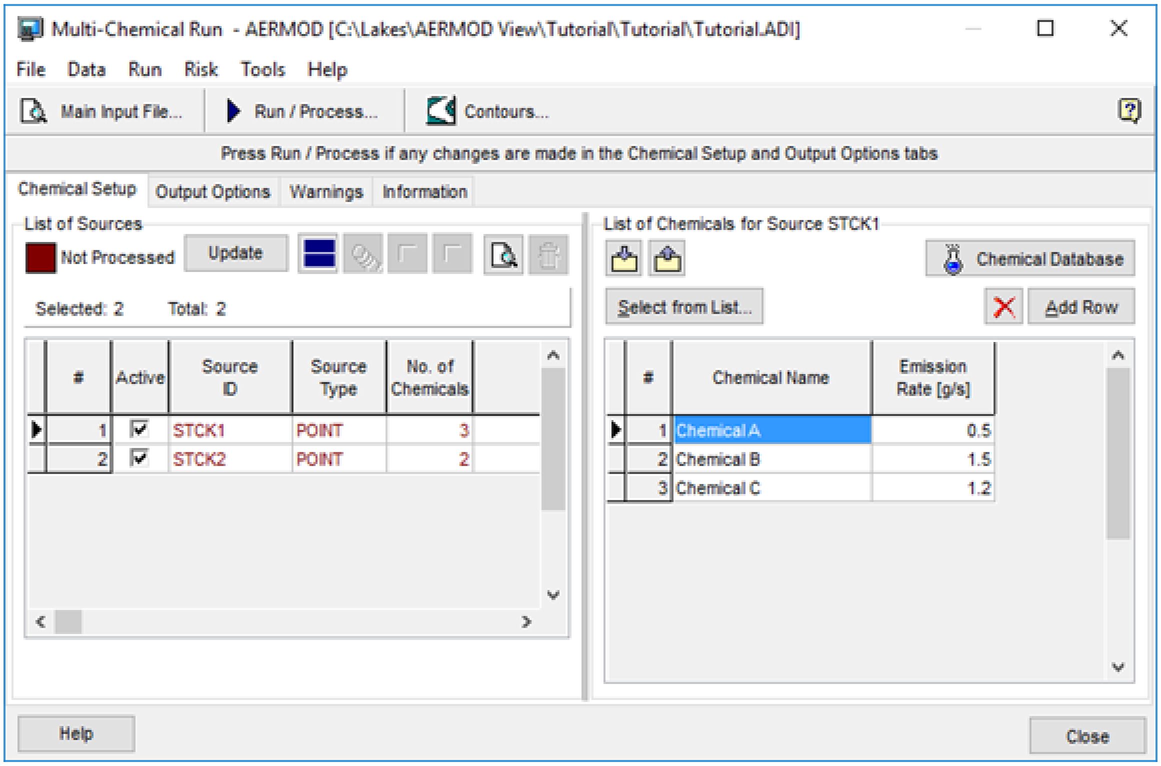
Features that are **NOT** supported include:

- Special pollutant options, such as SO2, NOx and PM10 – pos 97 NAAQS
- Optional control files, such as Re-Start files
- TOXICS/Gas Dry Deposition when the deposition velocity is calculated by the model
- TOXICS/SCIM Option
- Source groups other than Group ALL
- Discrete polar receptors and polar plant boundary receptors
- MAXTABLE (Maximum Values) and DAYTABLE (Daily Values) options
- Only chemical specific plotfiles are generated, any other type of output file is ignored, for example post-processing files
- The main output files that are generated are specific to only one source with unitized emission rate
- Chemical-specific output files are not being generated at this point

Since not all options are supported in a multi-chemical run it is important to inspect your input files. Any information in the input file not supported by the Multi-Chemical Run will be commented out (**). If this is information that is required, you will need to do a regular model run.

Multi-Chemical Utility - Interface Overview

You can load the Multi-Chemical Run utility by selecting Run - Multi-Chemical Run... The following dialog will load:



Multi-Chemical Run dialog

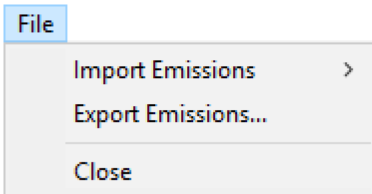
The main components of the **Multi-Chemical Run** utility are as follows:

- Menu Options
- Toolbar Buttons
- Chemical Setup tab
- Output Options tab
- Warnings tab
- Information tab

Multi-Chemical Utility - Menu Options

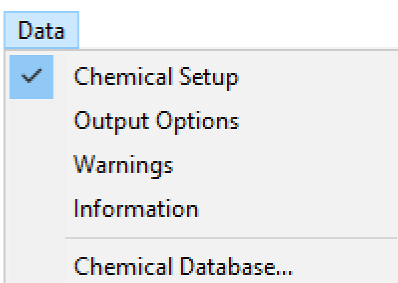
The following menu options are available for the **Multi-Chemical Run** utility:

File (Alt, F)



- **Import Emissions:** Displays the following submenu options -
  - **CSV File...:** Allows you to import emissions data for your sources from a *.csv file.
  - **Input File...:** The Import Emissions from ISC/AERMOD Input Files dialog is displayed which allows you to import emissions data for your sources from an ISC/AERMOD input file.
- **Export Emissions...:** Allows you to export the emissions data for your sources to a *.csv file.
- **Close:** Allows you to exit the **Multi-Chemical Run** utility.

Data (Alt, D)



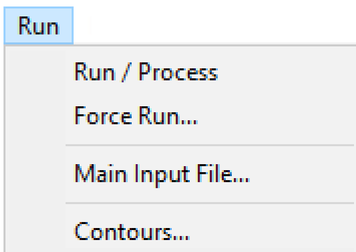
- **Chemical Setup:** Displays the Chemical Setup tab where you specify the multiple chemicals emitted from each source and their emission rates. A check indicates the tab is currently being displayed.
- **Output Options:** Displays the Output Options tab, which allows you to specify the output options. A check indicates the tab is currently being displayed.

- **Warnings:** Displays the Warnings tab which displays the information that was ignored from the input file by the Multi-Chemical Run utility. A check indicates the tab is currently being displayed.

- **Information:** Displays the Information tab, which tells you information about the current version of Multi-Chemicals Run. A check indicates the tab is currently being displayed.

- **Chemical Database...:** Opens the Chemical Database dialog where you must specify the chemicals being emitted by your sources.

Run (Alt, R)



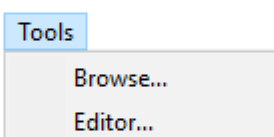
- **Run/Process:** Opens the AERMOD Batcher from where you can run the created input files for your sources.

- **Force Run...:** Forces the model executable to run the created input files for your sources.

- **Main Input File...:** Allows you to view the input file of the AERMOD View project that is currently open in WordPad.

- **Contours...:** Opens the POST View dialog where you can view the contour plots of your results from the multi-chemicals run.

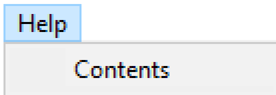
Tools (Alt, T)



- **Browse...:** Opens a Windows Explorer dialog to allow you to browse through files.

- **Editor...:** Displays Windows WordPad, allowing you easy access to a text editor.

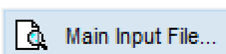
Help (Alt, H)



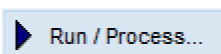
- **Contents:** Opens the Help Contents for the AERMOD View Help.

Multi-Chemical Utility - Toolbar Buttons

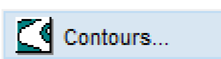
The toolbar buttons are shortcuts to some of the **Run** menu commands. The function of each one of these buttons is explained below:



Main Input File...: Allows you to view the input file of the AERMOD View project that is currently open.



Run/Process...: Opens the AERMOD Batcher where you can run the created input files for your sources.



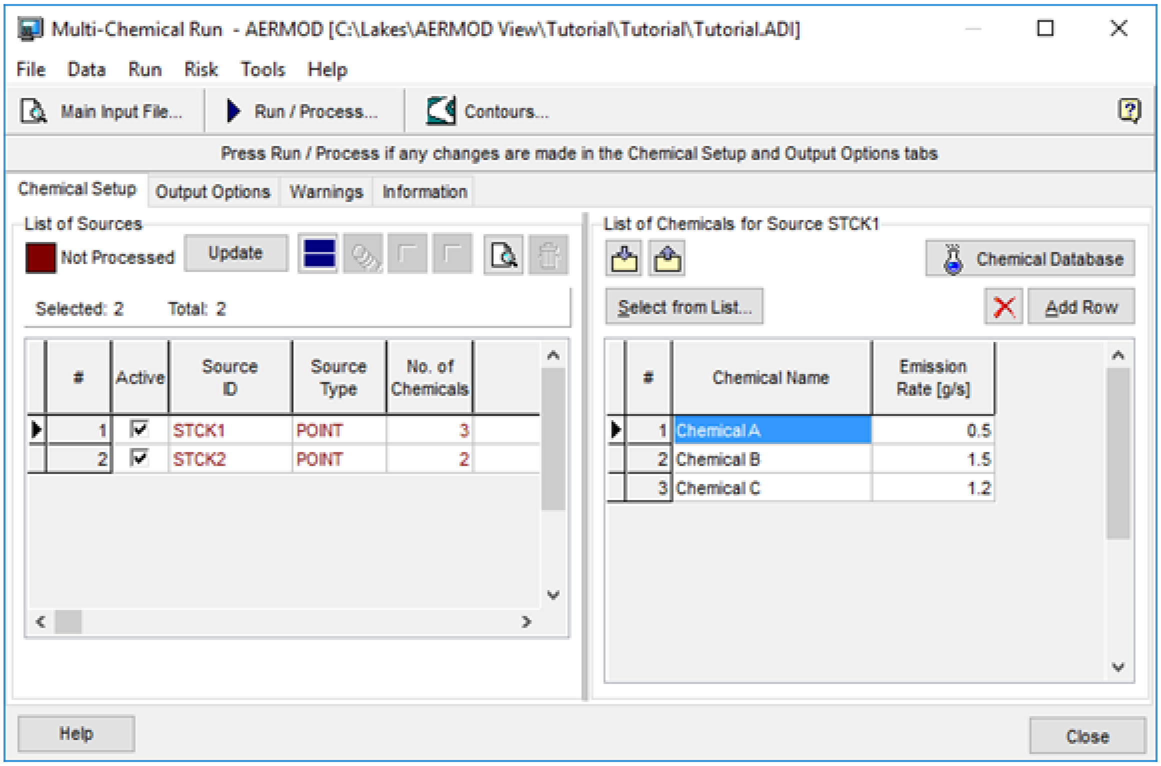
Contours...: Displays the main AERMOD View window where you can see contour plots of your results.

Set Up a Multi-Chemical Run

Warning: Advanced Users Only

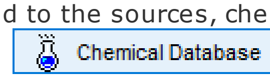
How to Set Up a Multi-Chemical Run:

1. To start the **Multi-Chemical Run** utility, select **Run | Multi-Chemical Run...** from the AERMOD View menu. The **Multi-Chemical Run** utility dialog will be displayed.



Multi-Chemical Run utility dialog

2. The sources that were specified in your project will be displayed in the Chemical Setup tab. Check the active box of all sources that are to be included in the multi-chemical run.

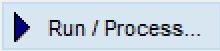
3. Before emissions can be assigned to the sources, chemicals will need to be added to the Chemical Database. Click on the 

button to open the **Chemical Database** where the names of the desired chemicals can be entered.

4. Once all of the chemicals have been entered, you may specify chemical-specific emission rates for each of the sources selected under the **List of Sources** table.

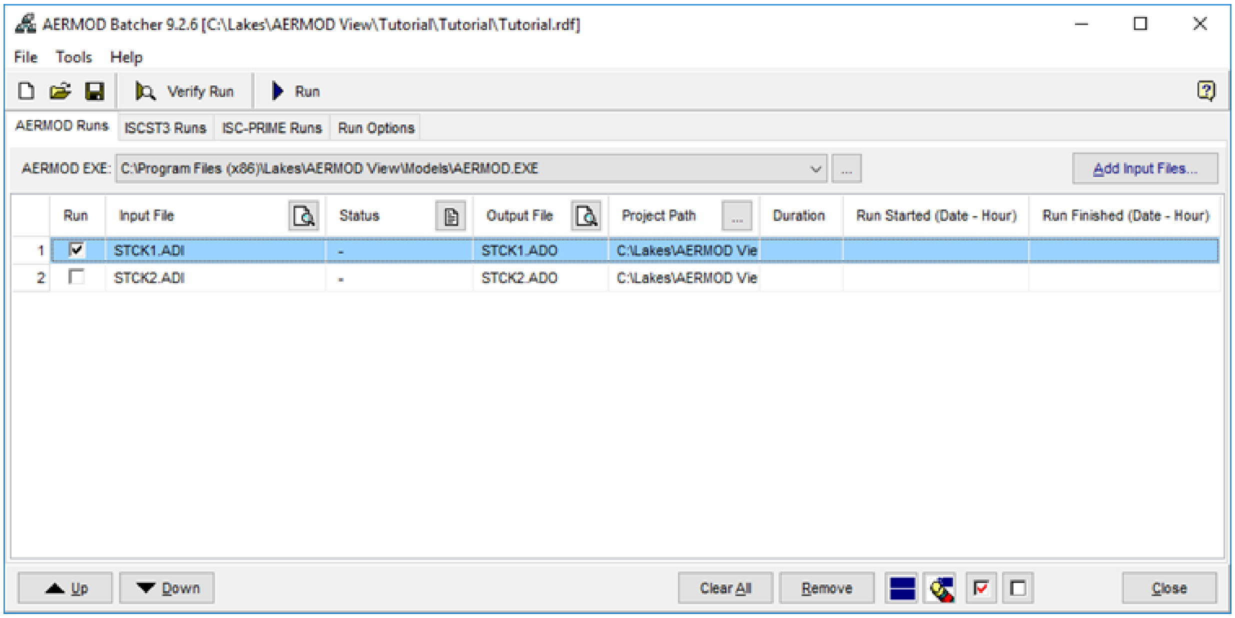
> It is important to inspect your input files as not all options are supported in a multi-chemical run. You can view the source input file by double clicking on a source in the **List of Sources** table or by selecting the source and clicking on the  button. Any information in the input file not supported by the Multi-Chemical Run will be commented out (**). If this information is required, you will need to do a regular model run. Always check the **Warning** and **Information** tabs for additional unsupported options.

5. Specify the output options for the run in the Output Options tab; these include **Averaging Times**, **High Values**, etc..

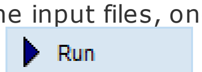
6. Once you have specified the chemicals and emission rates for each source, and your output options, press the 

button.

7. If you are running your model for the first time or if you have added new sources or receptors to the model the AERMOD Batcher will open.



AERMOD Batcher

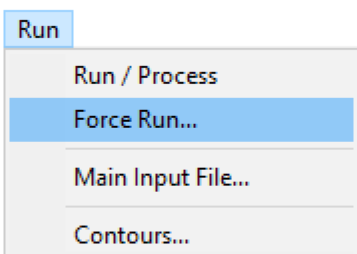
8. The **AERMOD Batcher** displays the input files, one for each source, as well as the model that will be used for the run. Click on the 

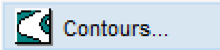
button to run the model.

9. Once the run is complete, the **AERMOD Batcher** will close.

10. If you have simply adjusted settings in the Output Options tab, the AERMOD Batcher will not run, but the data will be re-processed using the POST files generated during the run.

11. If you wish to run the model again, regardless of whether anything new has been added or removed, you can do so by selecting **Run | Force Run...** option from the main menu.

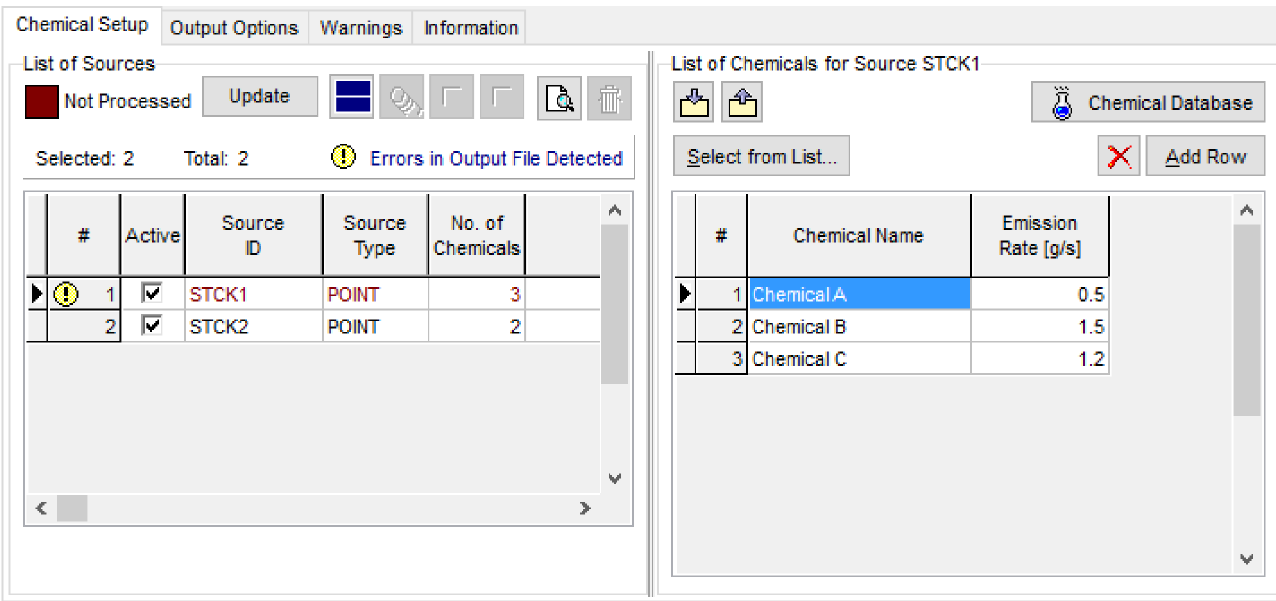


12. The **Multi-Chemical Run** utility will automatically generate the required plotfiles. When this process is complete, the **Multi-Chemical Run** utility can be closed, and the contour plots viewed in the AERMOD View interface. You may also view the contour plots by clicking on the 

button within the **Multi-Chemical Run** utility.

Chemical Setup tab

In the **Chemical Setup** tab you must specify all the chemicals and their emission rates for each source.



Multi-Chemical Run dialog - Chemical Setup tab

There are two main panels in the **Chemical Setup** tab -

List of Sources

This panel displays all of the sources defined in the current project. Sources displayed in red have not been run, while those in black have been processed. Sources that have been run, but unsuccessfully, will have the symbol displayed next to it indicating that errors in the output file were detected.

The following information is displayed for each of the sources -

- **Active:** Check this box if you would like your source to be active and included in the run.
- **Source ID:** Displays the Source ID.

- **Source Type:** Displays the Source type.

- **No. of Chemicals:** This field displays the number of chemicals the source emits. The names of these chemicals and their emission rates for the selected source are displayed in the List of Chemicals panel. If a source does not have any chemicals specified for it, then this source will not be processed and will be highlighted in red.

- **Description:** Displays a description of the source.

- **Post File:** If you have previously run the input file for the source, you may click on the ... button and specify the Post-Processing file (*.pos) for the source. This allows you to simply generate the plotfiles by clicking on the Run / Process... button.

The following buttons are available in the **List of Sources** panel -

Update Click this button to check if the POST files for the selected sources exist. If they do, these sources will be marked as processed.

Click on this button to remove the results for the selected source. This will permanently delete the Post-Processing file (*.pos) from the project.

Click on this button to view either the input or output file for the selected source. The output file will be available only if you have run the source.

Selects all sources.

Unselects all sources.

Checks the Active box for all selected sources.

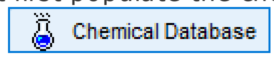
Unchecks the Active box for all selected sources.

List of Chemicals

This panel displays the list of chemicals to be modeled for the selected source and also allows you to specify the chemicals emitted by the selected source in the List of Sources panel as well as and their individual emission rates.

How to Specify the Chemicals for your Sources:

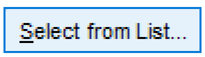
1. In the **List of Sources** panel, select the source for which you want to specify chemicals and emission rates.

2. Before you can specify the chemicals for the source, you must first populate the chemical database with all the chemicals for your project. Click on the 

button to open the Chemical Database dialog where you can set up all the chemicals you will be specifying for your sources.

3. Once you have defined all the chemicals in the **Chemical Database** you can assign one or more chemicals that are being emitted by your source using either of the following methods -

    - Click in the **Chemical Name** cell to display a drop-down list from where you can select a chemical.

    - Click on the 

button to display the List of Chemicals dialog from where you can select a chemical.

4. After selecting the chemical, you must then enter the emission rate in grams per second (g/s) in the Emission Rate field.

5. To add another chemical for the source, click on the button and then specify the chemical and emission rate.

The following buttons are also found in the **List of Chemicals** panel:

 Allows you to clear either the selected chemical or all the chemicals for the selected source.

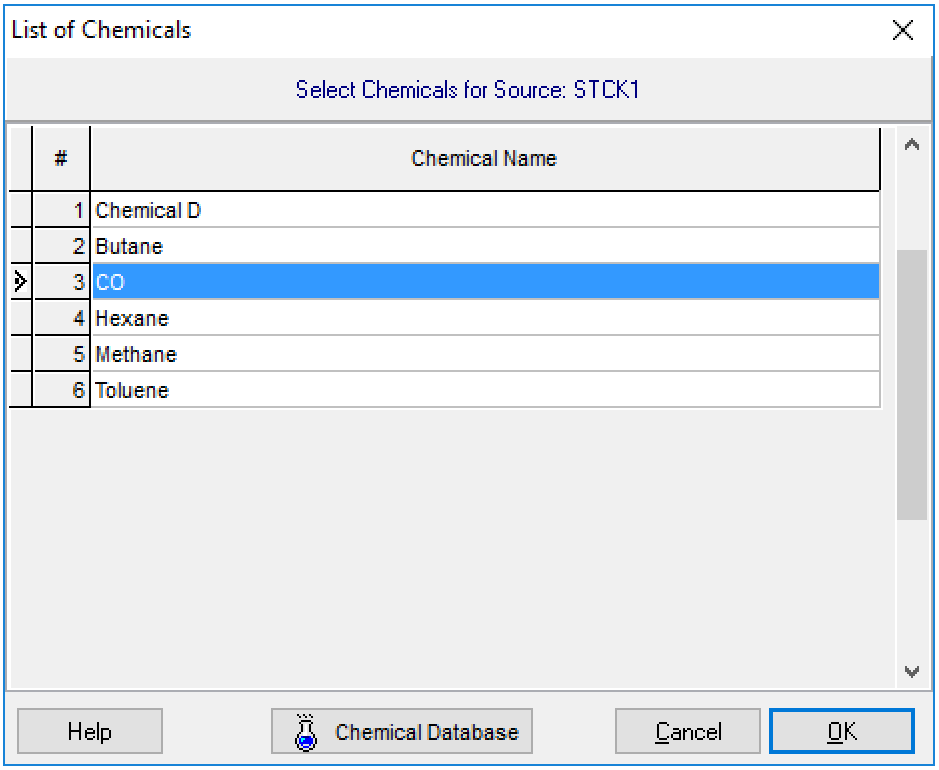
 Allows you to import chemicals and emission rates for your sources. You may import data either from a *.csv file or from an input file. If you select to import from an input file the Import Emissions from ISC/AERMOD Input Files dialog will be displayed.

> See Also Multi-Chemical File Import Format (*.csv).

 Allows you to export the chemicals and emission rates for your sources.

List of Chemicals

The **List of Chemicals** dialog displays a list of all the chemicals that you have defined in the Chemical Database dialog that have not yet been specified for the selected source. You have access to the List of Chemicals dialog by clicking on the Select from List... button in the Chemical Setup tab.



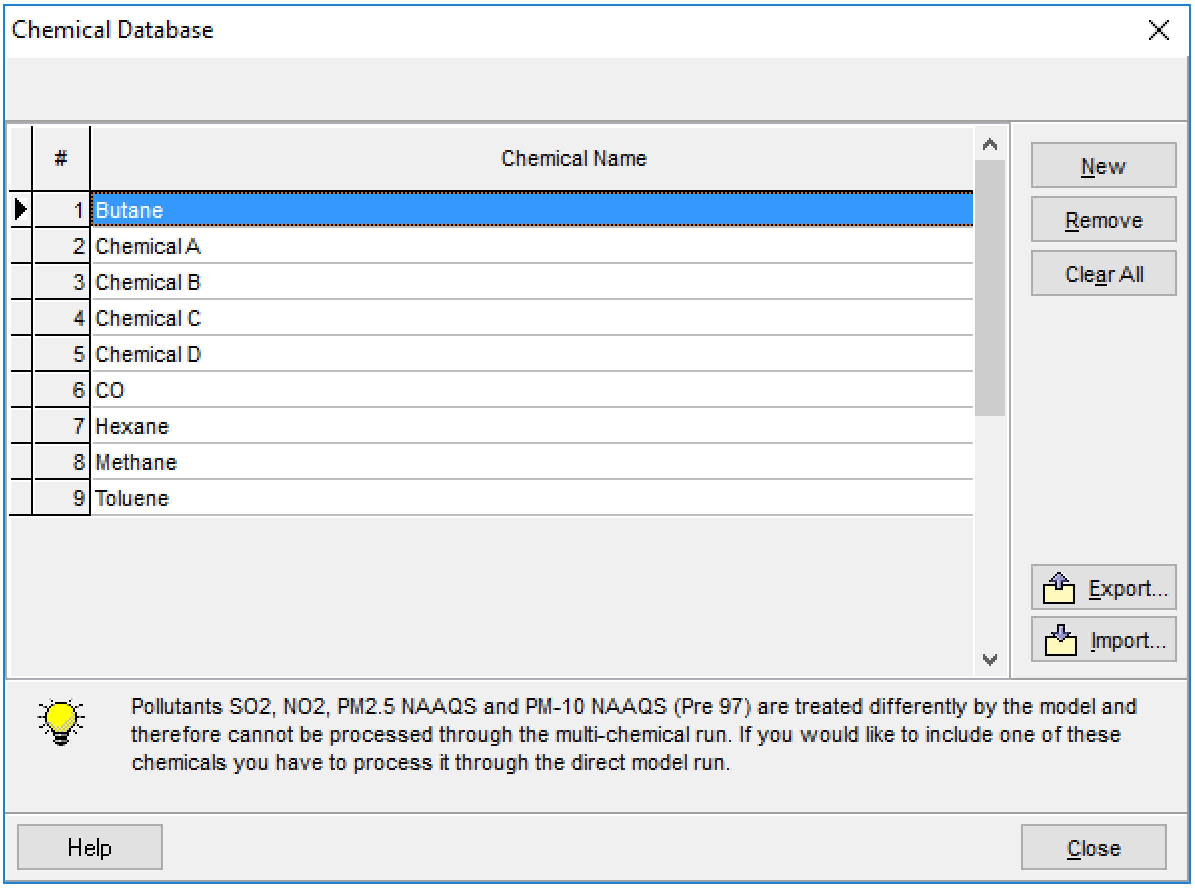
List of Chemicals dialog

From this list you simply select the chemical you wish to specify for your source. You can select multiple chemicals in sequence by pressing down the Shift key or you can make disjoint selections by pressing the Ctrl key.

Chemical Database

The **Chemical Database** allows you to specify all the chemicals for your project. Since sources are all treated as buoyant gases in AERMOD View, you do not need to specify any chemical properties, only the chemical name. The Multi-Chemical Run utility allows you to distinguish between the contributions from different components in a source by entering the emission rate of each component that makes up the source.

You have access to the **Chemical Database** dialog by clicking on the Chemical Database button in the Chemical Setup tab or by selecting **Data | Chemical Database...** from the menu.

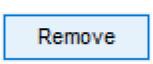


Chemical Database dialog

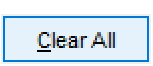
> Pollutants SO2, NOx, PM10-Pos 97 NAAQC are treated differently by the model and therefore cannot be processed through the Multi-Chemical Run utility. If you would like to include these chemicals, you will have to process it through the direct model run.

There are a number of buttons available, each of which are described below:

Allows you to specify a new chemical for the database. Press this button to add a new row to the table where you can then type the name of the chemical in the Chemical Name field.

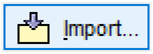


Removes the selected chemical from the list.



Removes all the chemicals from the database.

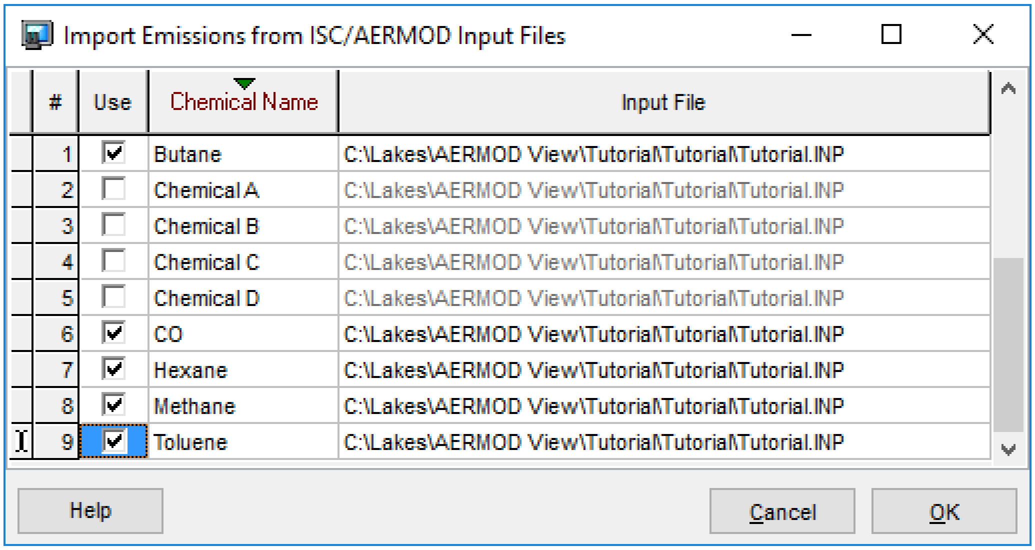
Allows you to export the current list of chemicals in text format (*.txt) for future use.



Allows you to import a list of chemicals in text format (*.txt).

Import Emissions from ISC/AERMOD Input Files

The **Import Emissions from ISC/AERMOD Input Files** dialog allows you to import emission rates for selected chemicals from an input file.



Import Emissions from ISC/AERMOD Input Files dialog

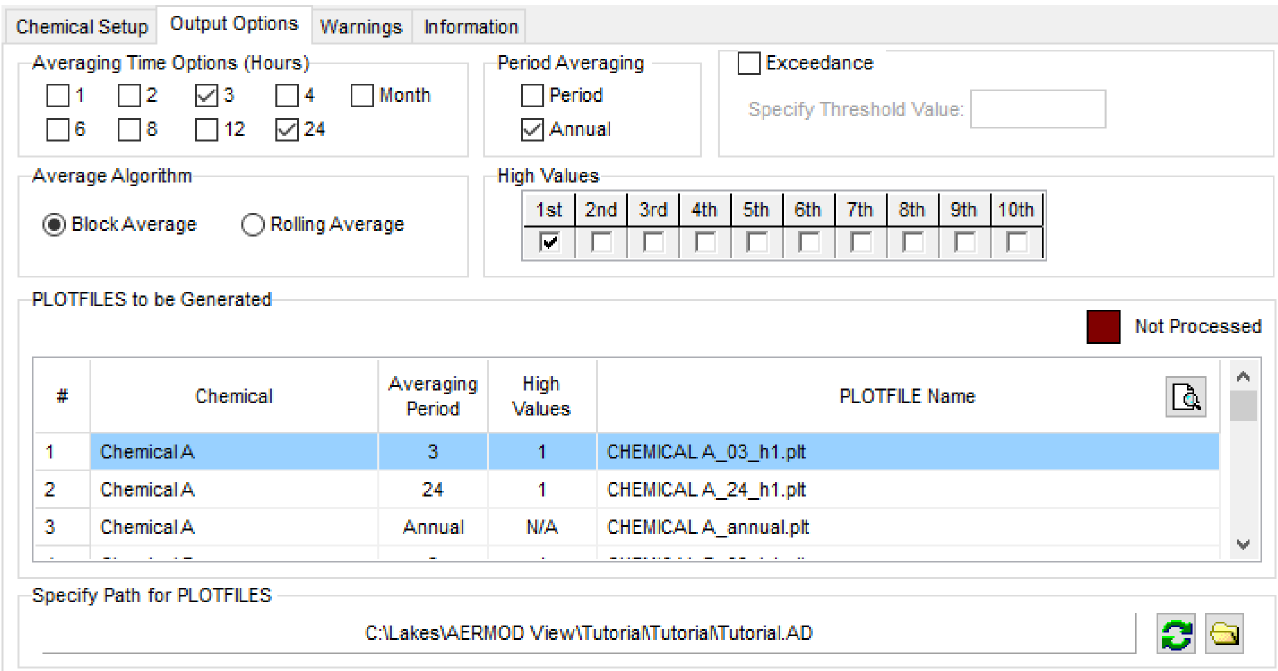
This dialog will list all the chemicals defined for the current project in the Chemical Database dialog. Select the chemical for which you wish to import emission rates by checking the **Use** box. In the **Input File** cell for the selected chemical click on the ... button to specify the name and location of the input file to use.

Emission rates for the selected chemical will be imported for any source IDs found in the specified input files that match the source IDs in your current project.

For example, if your project has the source ID STCK1 and you import emission rates from an input file which also has the source ID STCK1 then the selected chemical(s) will be added to the source STCK1 with the imported emission rate.

Output Options tab

The **Output Options** tab allows you to specify the output options for the **Multi-Chemical Run** utility. The output options that were selected in the AERMOD View project will automatically be selected in the **Multi-Chemical Run** utility. However, you can change and add to these options.



Output Options tab

The following output options are available in this tab -

- **Averaging Time Options (Hours):** Here you can specify the short term averaging times for which to calculate the output. Simply check the boxes of the averaging times you wish to use.

- **Period Averaging:** In this panel you may select the **Period** and **Annual** averaging options.

- **Exceedance:** Check this box if you wish to specify a threshold value.

- **Average Algorithm:** Here can select whether you wish to calculate a **Block Average** or a **Rolling Average**.

- **High Values:** Check the high values that are applicable to your modeling.

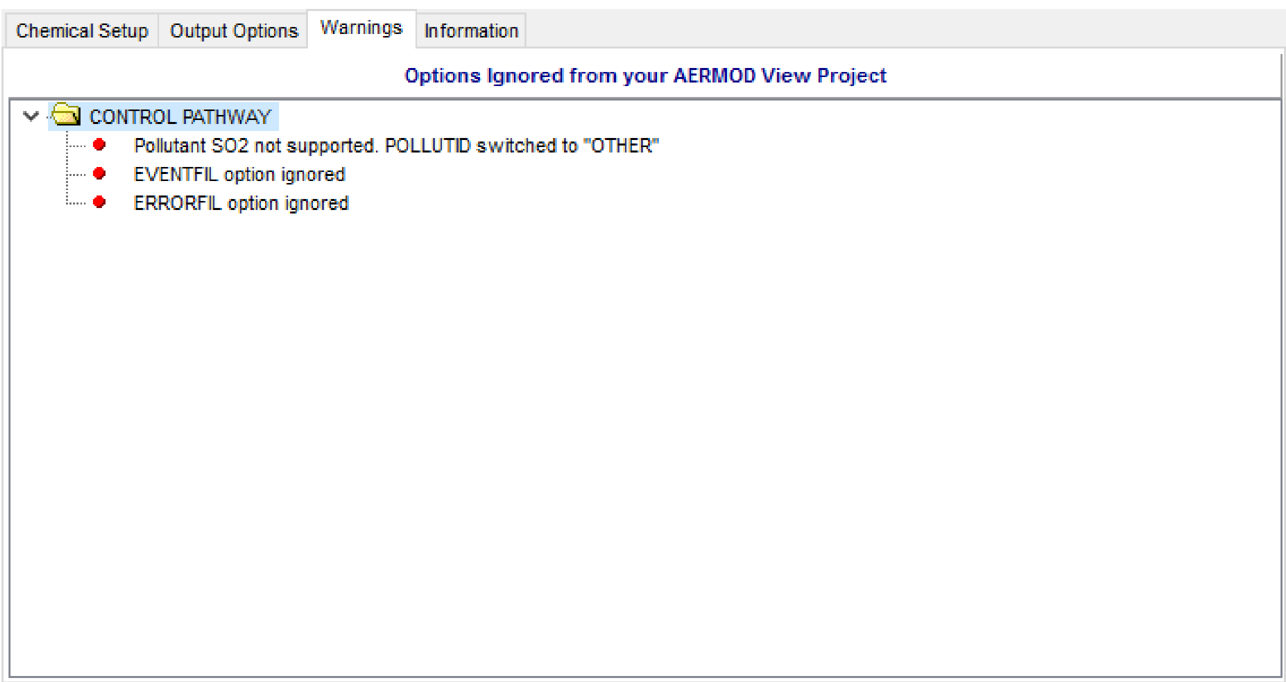
- **PLOTFILES to be Generated:** This list displays all of the possible plotfiles to be generated for every chemical, according to the selected averaging options and high values. Plotfiles displayed in red have not been processed while those in black have been processed. You can view a processed plotfile in WordPad by selecting it and clicking on the button.

- **Specify Path for Plotfiles:** By default, all plot files will be saved to preset locations. If you wish to specify an alternate location for the plotfiles, click on the button in the Specify Path for PLOTFILES panel and select the location to save the files. If you wish to return to the default location, click on the button.

> The default locations where the plotfiles are saved are determined according to the model that is run:
> **ISCST3 Runs:** project directory\projectname.IS\
> **ISC-PRIME Runs:** project directory\projectname.PR\
> **AERMOD Runs:** project directory\projectname.AD\

Warnings tab

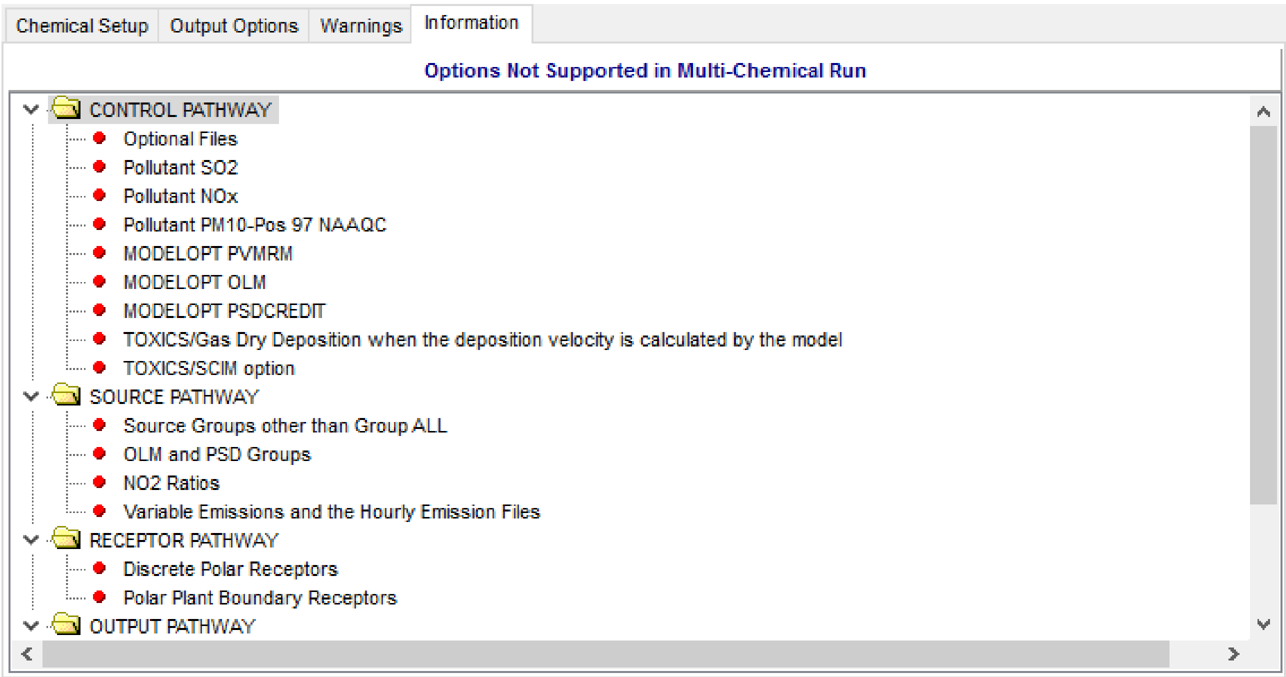
The **Warnings** tab displays the information that was ignored from the ISC/AERMOD input file by the **Multi-Chemical Run** utility. If there are any options listed here that you need to include in your run, you will have to carry out a regular model run.



Warnings tab

Information tab

The **Information** tab displays information regarding the options that are not supported in the current version of the Multi-Chemical Run utility.



Information tab

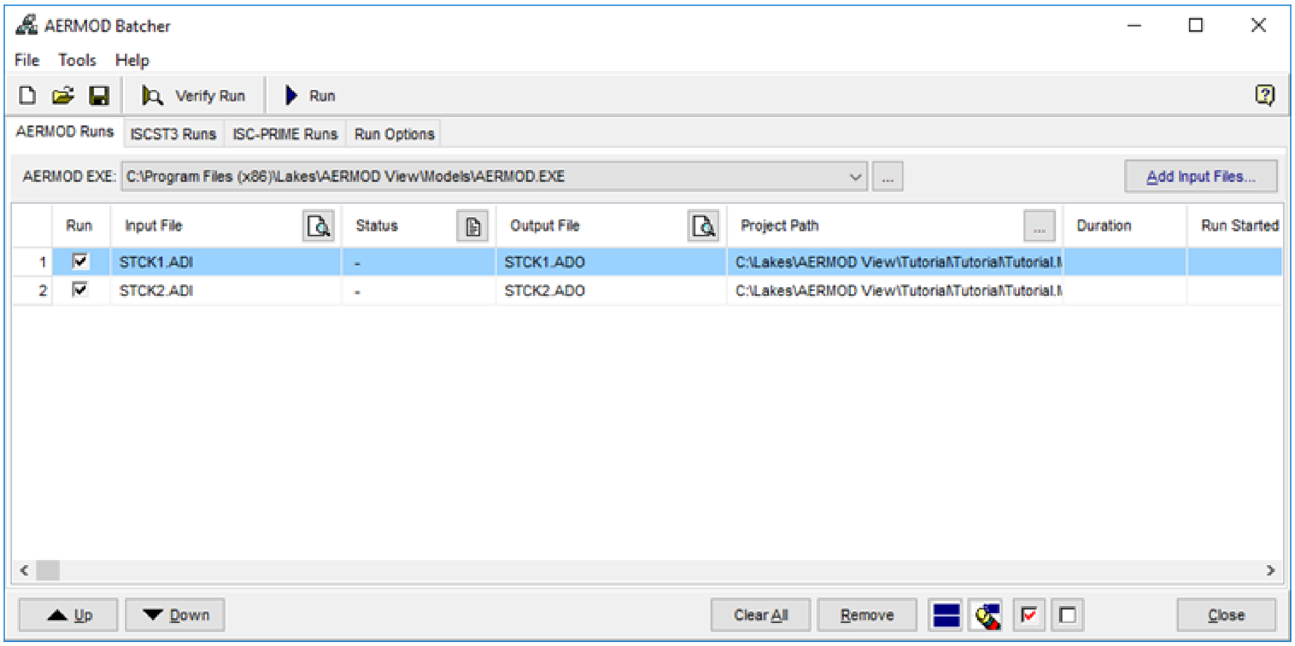
The following options are **NOT** supported:

- Special pollutant options, such as SO2, NOx and PM10 – pos 97 NAAQS
- Optional control files, such as Re-Start files
- TOXICS/Gas Dry Deposition when the deposition velocity is calculated by the model
- TOXICS/SCIM Option
- Source groups other than Group ALL
- Discrete polar receptors and polar plant boundary receptors
- MAXTABLE (Maximum Values) and DAYTABLE (Daily Values) options
- Only chemical specific plotfiles are generated, any other type of output file is ignored, for example post-processing files

- The main output files that are generated are specific to only one source with unitized emission rate
- Chemical-specific output files are not being generated at this point

AERMOD Batcher

AERMOD Batcher allows you to perform multiple model runs automatically. It is used primarily for the **Multi-Chemical Utility** and **RiskGen**.



AERMOD Batcher dialog

The Batcher contains options to conduct multiple AERMOD, ISCST3, and ISC-PRIME runs.

AERMOD/ISCST3/ISC-PRIME Runs

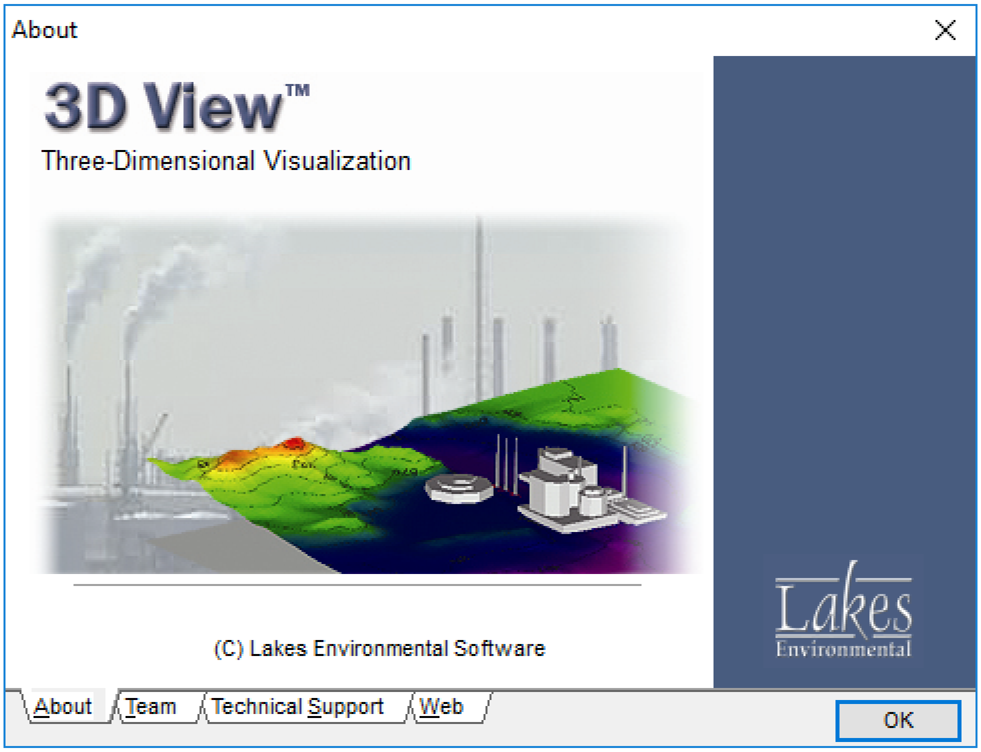
In these tabs you can select the input files for the batcher. If you are running the batcher as part of the **Multi-Chemical Utility**, the set-up will have been done in the utility dialog prior to calling the batcher.

Run Options

This tab allows you to run multiple models one after another by creating a list of input files. Batcher is also used in the AERMOD View Multi-Chemical utility to perform multi-chemical model runs.

3D View

3D View is a 3-dimensional visualization tool that enables you to quickly visualize digital terrain data in conjunction with your other project visualizations.



Giving you total control, you can zoom in and rotate your site and modeling results in true 3-Dimensional space. This allows you to view your model from any perspective and further investigate what is influencing your model results.

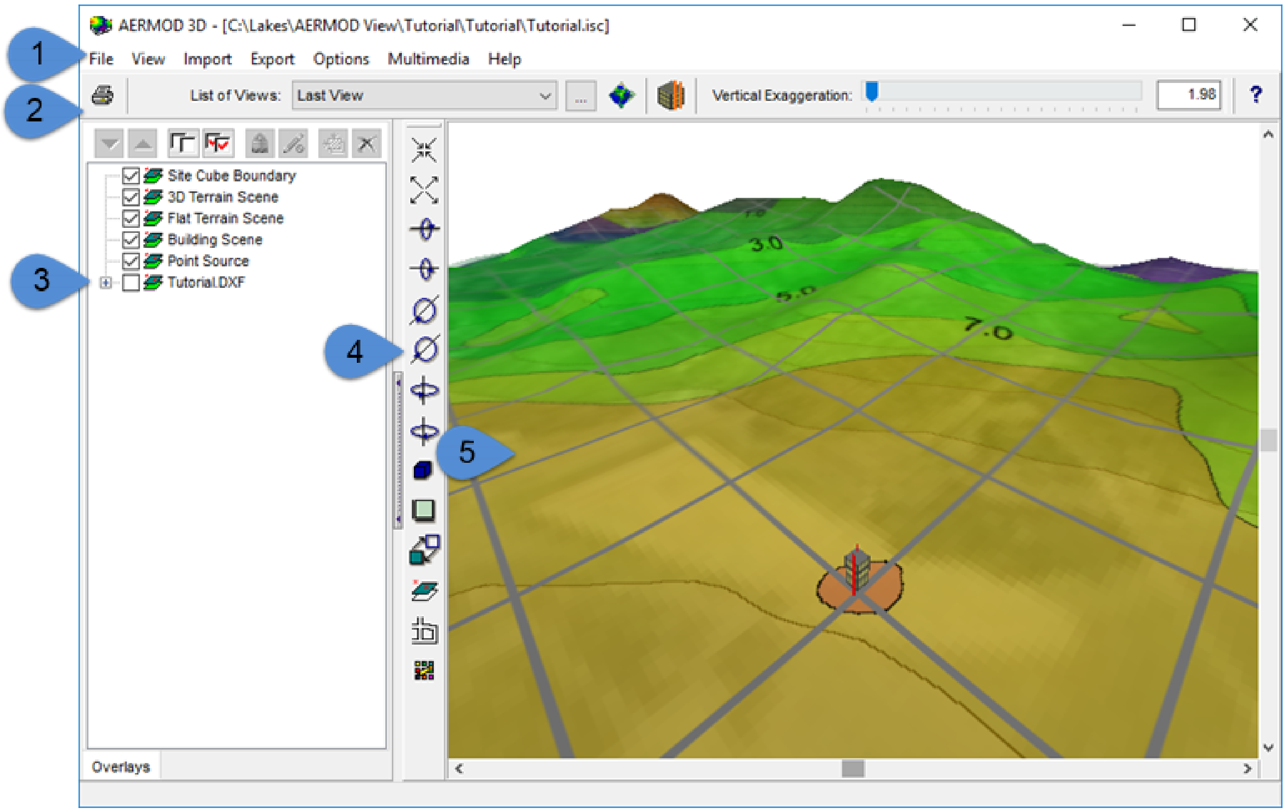
Some the features you will find in 3D View include the following:

- **Complete 3D Visualization** - View your site and surrounding terrain in true 3D. Site buildings and stacks (if applicable) appear in context with your modeling site terrain.
- **Custom Textures** - Apply custom textures to buildings to further increase building realism.
- **Multi-layer Visualization** - View multiple layers with ease using the powerful 3D engine. Display your 3D site in conjunction with your 2D data!
- **Realistic Views** - Clear, realistic communication is easy and your model will impress with real-time lighting effects, true-color shading and textures.
- **Concentration Contours** - Understand the effects of topography by displaying your modeled concentrations with your 3D model. Make your final report clear and concise by visualizing all your data.

3D View - Main Interface

3D View features a friendly, intuitive interface which provides easy access to all tools.

The components of the **3D View** window are briefly described below:



3d View Main Window

1. **Menu Bar:** Displays menu names. To open a menu, move the mouse over the menu name and then click the left mouse button. A drop-down menu appears displaying a list of related commands.

2. **Menu Toolbar and Options:** The Menu Toolbar provides quick access to some menu commands.

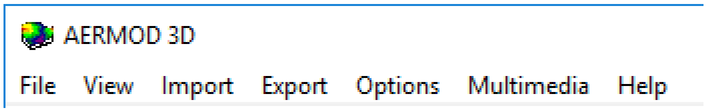
3. **Navigation Tree:** Allows you to turn the overlays on and off.

4. **3D Controls Toolbar:** The tools available in the 3D Controls Toolbar used to manipulate the three-dimensional image in the viewing window.

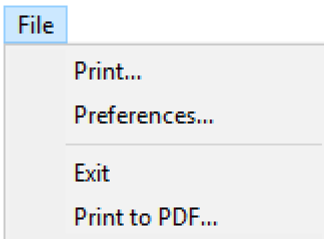
5. **Viewing Area:** Displays graphical representation of the sources, receptors, grids, buildings, etc. that are drawn in the exact locations you defined. The default color for the Viewing Area is white but this can be changed. Simply double click with the mouse anywhere in the Viewing Area. The Color dialog opens, allowing you to select the color of your choice.

3D View - Menu Options

The following menu options are available in **3D View**:

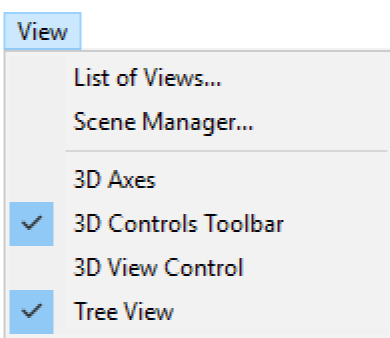


File



- **Print...:** Displays the Print Preview dialog where you can preview and print the contents of the drawing area.Print...: Displays the Print dialog where you can print the contents of the dialog.
- **Preferences...:** Displays the Preferences dialog, where you can specify preferences, such as company name, modeler & model default settings.
- **Exit:** Closes the program.
- **Print to PDF...:** Opens a Print Preview dialog that allows you to save your report as a PDF file.

View



- **List of Views...:** Displays the List of Views dialog, which lists views that have been previously saved.

- **Scene Manager...:** Displays the Scene Manager dialog, which lists all of the active and inactive scenes. In the Scene Manager, you can specify whether a scene is visible or not in the 3D View area.

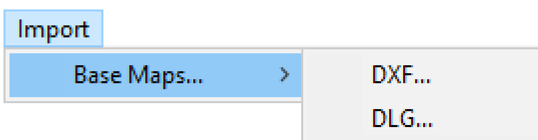
- **3D Axes:** Turns on and off the display of the 3D Axes display box, which shows you the current orientation of the 3D axes.

- **3D Controls Toolbar:** Turns on and off the display of the 3D Controls Toolbar. A check indicates that the toolbar is currently active.

- **3D View Control:** Turns on and off the display of the 3D View Control display box, from which you can graphically control the rotation of the 3D image using the sphere . A check indicates that the display box is currently active.

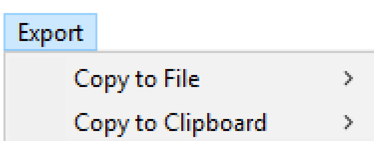
- **Tree View:** Turns on and off the display of the Tree View panel, in which you can select which overlays are displayed.

Import



- **Base Maps:** Displays the following submenu options -
  - **DLG...:** Displays the Importing DLG Base Map dialog.
  - **DXF...:** Displays the Importing DXF Base Map dialog.

Export

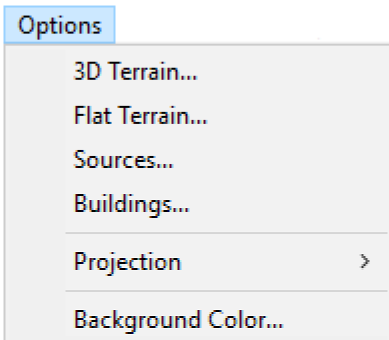


- **Copy to File:** Displays the following sub-menu options -
  - **Bitmap.....:** Displays the Export As Bitmap dialog, allowing you to save the contents of the drawing area as a Windows bitmap.
  - **Metafile...:** Displays the Save Enhanced Metafile dialog, allowing you to save the contents of the drawing area as a Windows metafile.
- **Copy to Clipboard:** Displays the following sub-menu options

- **Bitmap.....:** Copies the contents of the drawing area to the clipboard, allowing you to paste these contents into another windows application as a Windows bitmap.

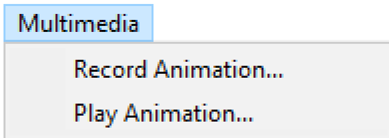
- **Metafile...:** Copies the contents of the drawing area to the clipboard, allowing you to paste these contents into another windows application as a Windows metafile.

Options



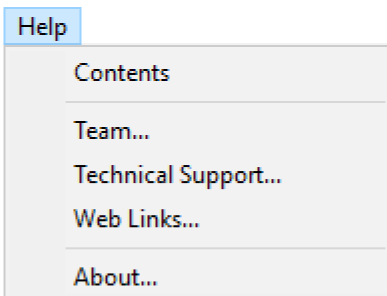
- **3D Terrain...:** This option displays the Terrain Options dialog, where you can control the appearance of your 3D terrain.
- **Flat Terrain...:** This option displays the Graphical Options dialog where you can customize the display of your 2D digital terrain data.
- **Sources...:** Displays the Graphical Options dialog, where you can customize the display of your sources.
- **Buildings...:** Displays the Graphical Options dialog, where you can customize the display of your buildings.
- **Projection:** Displays the following submenu options -
  - **Perspective:** This projection simulates the 3D view as it may appear to the human eye, providing an additional feeling of depth and perspective. A check indicates which projection is being displayed.
  - **Isometric:** This projection displays the 3D view without any distortion, that is, it displays the view with equivalent proportions for the objects. A check indicates which projection is being displayed.
- **Background Color...:** This option opens the Color dialog, allowing you to select an alternate color for the background of the 3D View area. The default color for the background is white.

Multimedia



- **Record Animation...:** Displays the Multimedia Options dialog from where you can record animations.
- **Play Animation...:** This option allows you to select the name and location of the animation file to play.

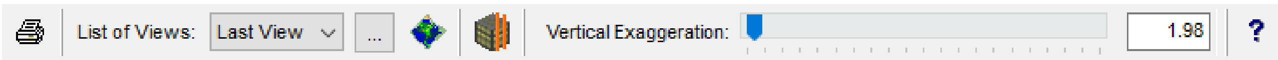
Help



- **Contents:** Displays Help Contents, from which you can select topics.
- **Team...:** Displays information on the program development team.
- **Technical Support...:** Displays a dialog containing available Technical Support options for the program.
- **Web Links...:** Displays a dialog containing links to sites useful to the user.
- **About...:** Displays the copyright notice and version number for the interface.

3D View - Menu Toolbar Buttons and Options

The **Menu Toolbar** provides shortcuts to some of the menu commands. The function of these buttons is explained below:

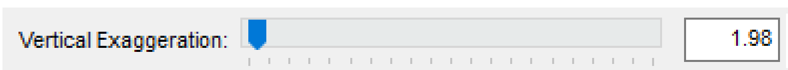


 Displays the Print Preview dialog where you can preview and print the contents of the drawing area.

 Displays the List of Views dialog where you can save the currently displayed view. From the **List of Views** drop down list you can select a previously saved view to be displayed in the viewing area.

 Returns the view to the default settings.

 Zooms in to the defined sources and buildings.



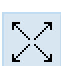
Use the slide marker to change vertical exaggeration, or enter the value in the text box.

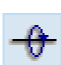
 Displays the **3D View Help Contents** from where you can select specific topics.

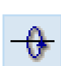
3D View - 3D Controls Toolbar

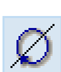
The tools available in the **3D Controls Toolbar** are used to manipulate the three-dimensional image in the viewing window. The function of each one of these tools is explained below:

 **Zoom In:** Zooms in on the image.

 **Zoom Out:** Zooms out on the image.

 **Pos X Rotate:** Rotates the image clockwise on the X-axis.

 **Neg X Rotate:** Rotates the image counter-clockwise on the X-axis.

 **Pos Y Rotate:** Rotates the image clockwise on the Y-axis.

Neg Y Rotate: Rotates the image counter-clockwise on the Y-axis.

Pos Z Rotate: Rotates the image counter-clockwise on the Z-axis.

Neg Z Rotate: Rotates the image clockwise on the Z-axis.

Wire Frame: Toggles the wire frame mode on and off.

Reset View: Resets the image back to the initial view.

Preview: Toggles the preview mode on and off. Preview mode greatly simplifies the image for faster manipulation of the image.

Scene Manager: Displays the Scene Manager dialog.

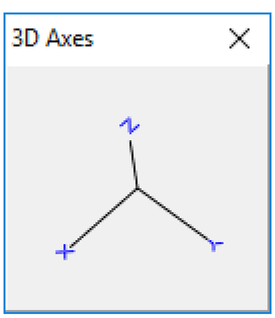
Map Import: Imports DXF or DLG base maps to be displayed in the drawing area.

Graphical Options: Displays the Graphical Options dialog.

3D Options

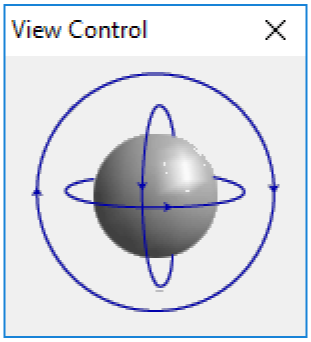
3D Axes and 3D View Control

The **3D Axes** display box shows you the current orientation of the three axes (x, y, and z) of your current 3D view. You can turn on and off the display of this control by selecting **View | 3D Axes** from the menu.



3D Axes

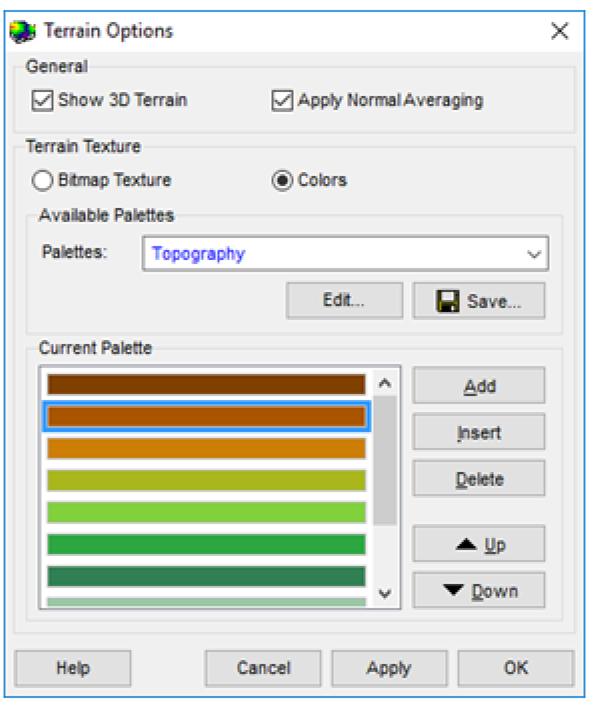
By default, the 3D **View Control** display is shown every time you start 3D View. You can turn on and off the display of this control by selecting **View | 3D View Control** from the menu. The 3D View Control allows you to graphically control the rotation of your 3D models. Holding down the mouse left button, click on the sphere and drag the mouse pointer in the direction you want the image to rotate.



3D View Control

3D View - Terrain Options

The **Terrain Options** dialog allows you to customize the display of your 3D digital terrain data. You can access this dialog by selecting **Options | 3D Terrain...** from the menu.



Terrain Options dialog

The following options are available:

General

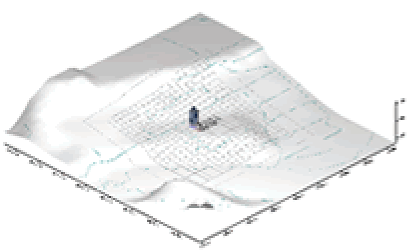
In this panel you can specify the appearance of your 3D model. The following options are available:

- **Show 3D Terrain:** Check this box to display the 3D Terrain in the viewing area.
- **Apply Normal Averaging:** Checking this box smoothes the appearance of the digital terrain by applying an averaging routine between the many mesh points that make up your 3D terrain model. This provides smoother visualization but does also consume additional computing resources.

Terrain Texture

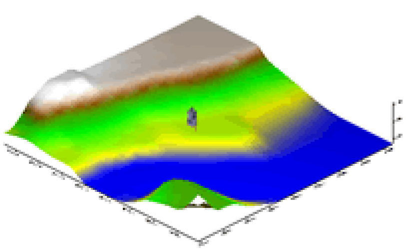
Here you can specify what type of fill you wish to use on your 3D model. The following options are available:

- **Bitmap Texture:** This overlays a texture that is identical to the current 2D visualization of your project in the program from which you launched your 3D view. For example, if you included maps or additional items in your 2D project, they will be overlain on the 3D terrain model.



Bitmap Texture

- **Colors:** Allows you to define and apply a range of colors to use as shaded relief to represent the elevations in your 3D terrain model.

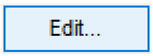


Color Fill

The **Colors** option provides you total control over the color palettes. The following two sections become available when the **Colors** option is selected:

Available Palettes

This panel allows you to select predefined palettes or save new custom palettes. From the **Palettes** drop-down list, select an existing color palette you would like to use. The colors of the selected palette will be displayed in the **Current Palette** panel.



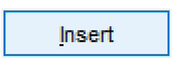
Opens the Palette Manager dialog where you can customize your own color palette.

Allows you to save the created or modified color palette displayed in the **Current Palette** section under a specified name. You can enter any text you wish to describe the color palette.

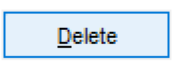
Current Palette

This panel displays the colors assigned to the current palette selected from the **Palettes** drop-down list. It also allows for complete customization of the colors assigned to the palette. You can use the following buttons to customize the displayed color palette:

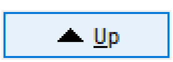
Opens the Color dialog allowing you to select a color to be added to the current palette. The new color will be added at the end of the table by default.



Opens the Color dialog allowing you to insert a new color to the current palette. Click on the position you wish to insert the color and press this button.



Deletes the selected color from the current color palette.

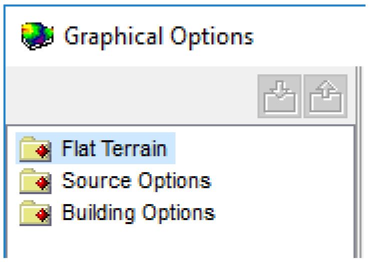


Moves the selected color up one level in the color order.

Moves the selected color down one level in the color order.

3D View - Graphical Options

The **Graphical Options** dialog allows you to control the display and specify properties for several objects in your project.

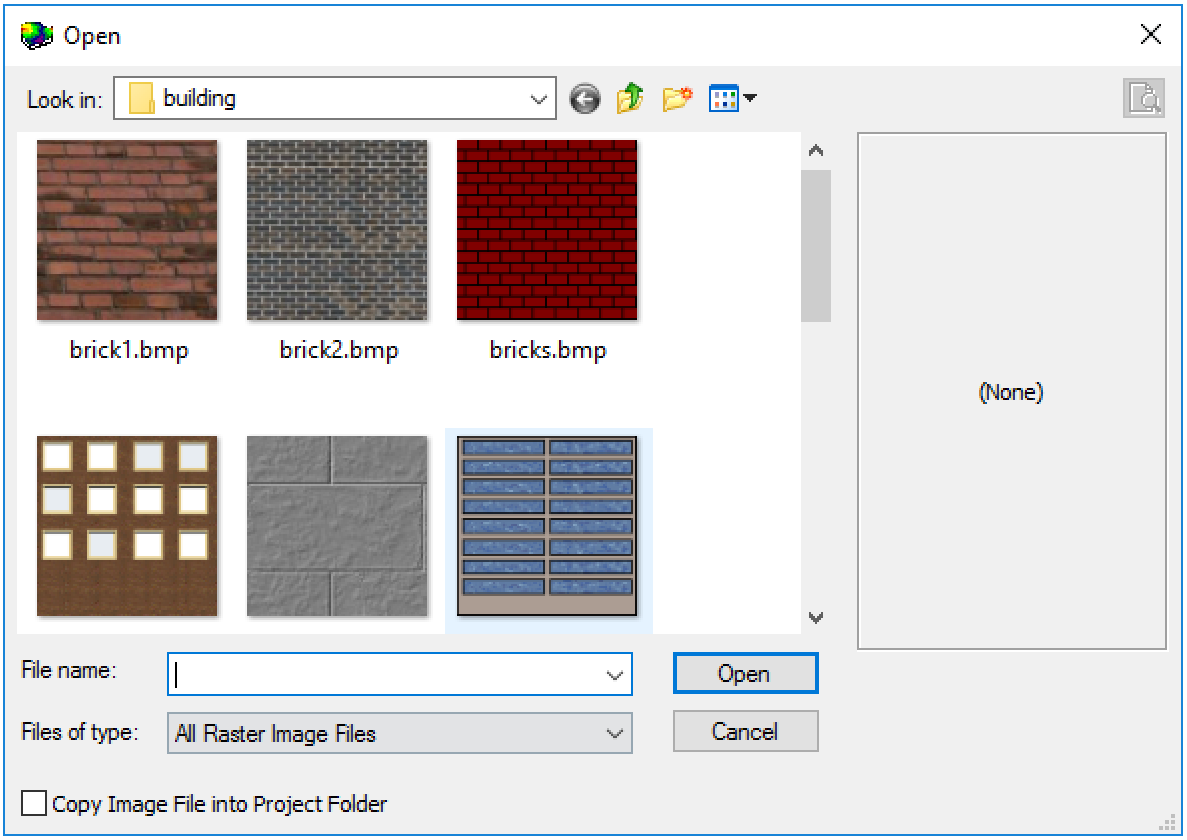


You have the following options in the **Graphical Options** dialog:

- Flat Terrain
- Source Options
- Building Options

3D View - Specify Bitmap

The **Specify Bitmap** dialog allows you to specify the name and location of the bitmap you wish to use as a texture to be applied to your building walls or flat terrain scene. You have access to this dialog by clicking on the Wall Texture... button located in the Building Options dialog.



Specify Bitmap dialog

The following options are available in this dialog:

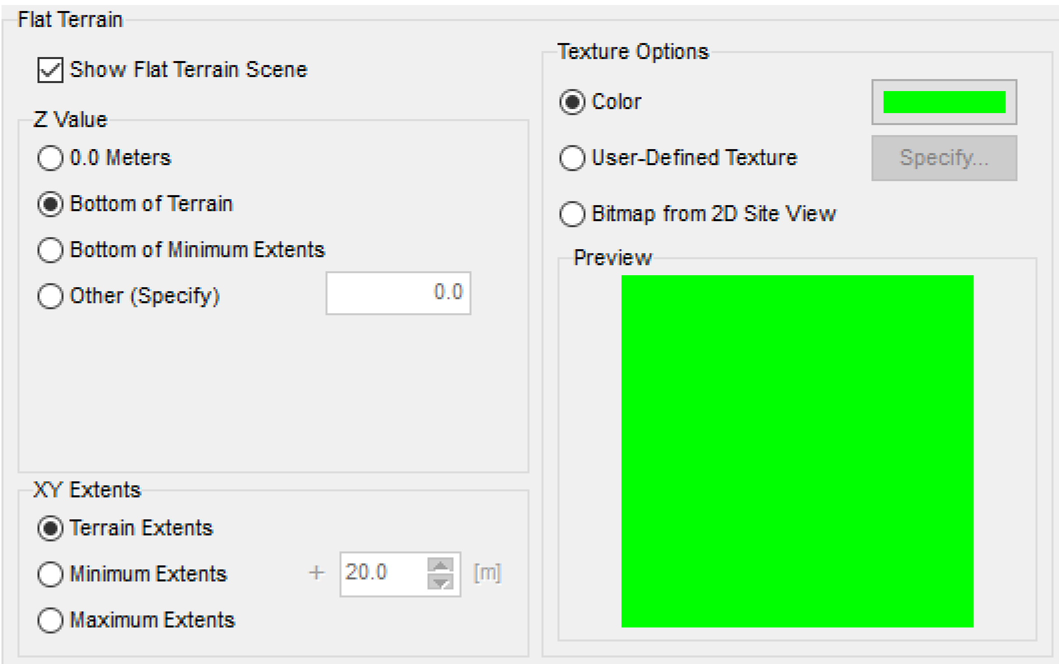
- **Preview:** Once you have located the folder that contains your texture bitmaps, you can easily preview them before making your selection. Simply select a bitmap and it will be displayed in the **Preview** panel. You can find bitmap samples that can be used for the terrain in the Texture folder located in the product installation folder.

- **Copy Bitmap into Project Folder:** This option copies the bitmap file you have specified to your project folder. This is useful if you need to backup or transfer your project to another machine. To select this option, check the **Copy Bitmap File into Project Folder** box. The default is unchecked. Once you selected your bitmap, click the **Open** button.

3D View - Flat Terrain

The **Flat terrain** dialog contains options for customizing the display of your 2D digital terrain data and enables you to specify properties for flat terrain. The flat terrain layer is represented by a flat plane in space. If your terrain is flat, this plane will represent the ground of your visualization area.

You have access to this dialog by selecting **Options | Flat Terrain...** from the menu.



Scene Options dialog - Flat Terrain

The following options are available:

- **Show Flat Terrain Scene:** Checking this box will display the flat terrain scene on the 3D view area.

Z Value

In this panel you can specify the Z value of the flat terrain scene. The following options are available:

- **0.0 Meters:** Select this option if you want the flat terrain to be positioned at elevation 0m.
- **Bottom of Terrain:** This option is only available if terrain data is available. In this case, you can choose to position the flat terrain scene at the lowest elevation point of the terrain data.

- **Bottom of Minimum Extents**: Select this option if you want the flat terrain to be positioned at the minimum extents.

- **Other (Specify):** In this option you can specify any elevation value for positioning the flat terrain scene.

XY Extents

This panel allows you to specify the XY extents of the flat terrain. The following options are available:

- **Terrain Extents:** This option will extend the dimensions of the flat terrain scene in the XY direction to the same extents of the terrain data, if available.

- **Minimum Extents:** This option will limit the dimensions of the flat terrain scene in the XY direction to the extents of the modeling objects, also called minimum extents.

- **Maximum Extents:** This option will extend the dimensions of the flat terrain scene in the XY direction to the size of the maximum extents. The maximum extents usually includes modeling objects, 3D terrain, and maps.

Texture Options

Here you can specify what type of fill you wish to use on your flat terrain model. The selected fill is displayed in the **Preview** panel. The following options are available:

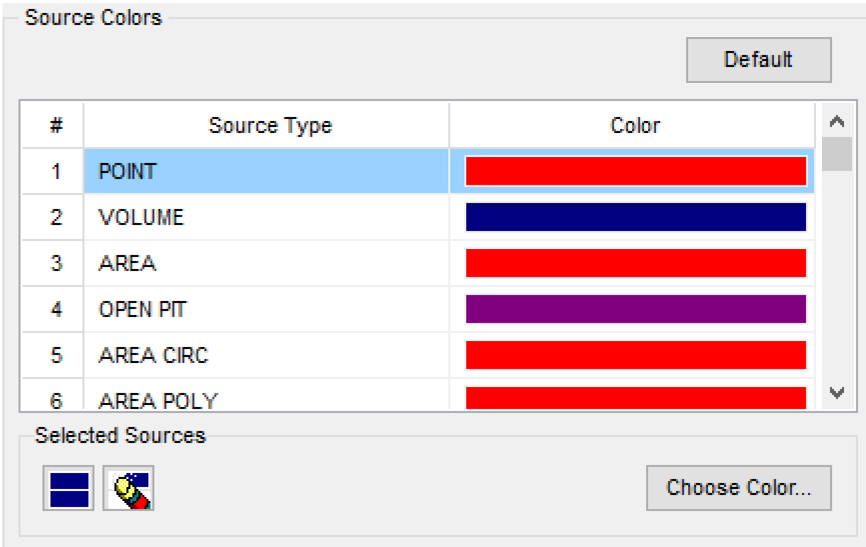
- **Color:** Allows you to define and apply a range of colors for your flat terrain model. Clicking on the **Color** button will open the Color dialog where you can select from a range of colors.

- **User-Defined Texture:** Allows you to select a texture for your flat terrain scene in bitmap format. Click on the Specify... button to open the Specify Bitmap dialog where you can specify the name and location of the bitmap file you want to use. A sample of bitmap textures are provided with your product installation.

- **Bitmap from 2D Site View:** This option allows you to use the automatically generated 2D site view image as a texture for the flat terrain. This image is georeferenced and defines xy extents for the flat terrain.

3D View - Source - Colors

The **Source Scene - Colors** page allows you to control the display of your sources in the 3D View area. You can access the Source Scene option by selecting **Options | Sources...** from the menu.



Scene Options dialog - Source Scene Colors

The following options are available under the **Source Scene - Colors** option:

- **Show Source Scene:** You can check or uncheck this box to display or hide your sources.
- **Default:** Click on this button to return the settings to the application default.
- **Source Type:** This column lists the various source types available in your project
- **Color:** This column displays the colors selected for your source types. To change the color, double click on the color bar or select the source type and click on the Choose Color... button to open the Color dialog. From here you can select an alternate color for your source.

Button Guide

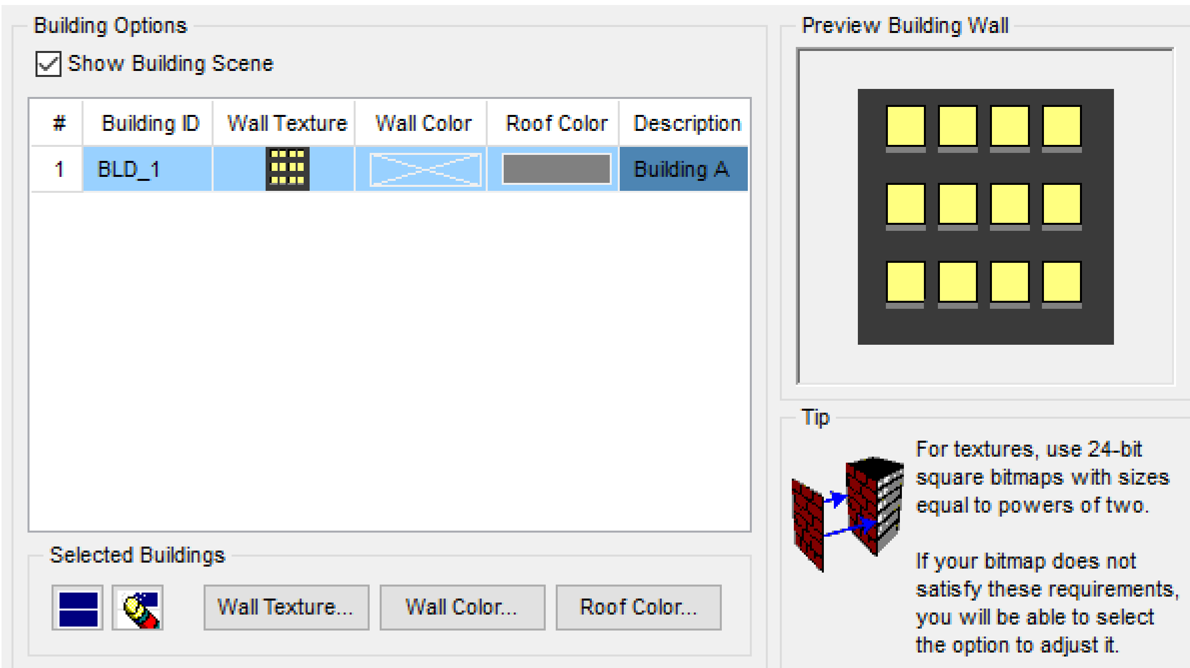
 Selects all sources, allowing you to press the Choose Color... button and to select a color to be applied to all sources.

 Unselects all sources.

3D View - Building Options

If your project has buildings defined it, then the buildings will appear in 3D with your terrain model.

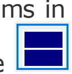
The **Building Options** dialog allows you to control the display and appearance of the buildings in your project. You have access to this dialog by selecting **Options | Buildings...** from the menu.



Building Options dialog

If you choose not to display your Building Scene, you can uncheck the **Show Building Scene** box and your buildings will not be displayed.

How to Apply Color or Texture to Your Buildings:

1. Select the building for which you wish to change the properties. You can select multiple items in sequence by pressing down the Shift key as you make selections. You can also click on the  to select all buildings, and the  button to unselect all buildings.

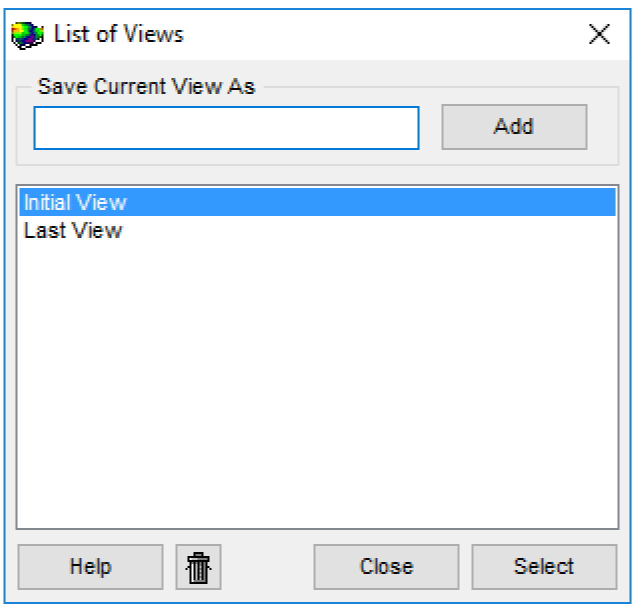
2. Clicking on either the Roof Color... or Wall Color... buttons will open the Color dialog, which allows you to select the color you wish to have applied to your building. The wall color is previewed in the **Preview Building Wall** panel.

3. Clicking on the Wall Texture... button will display the Specify Bitmap dialog, where you can specify a bitmap image to be applied as a wall texture on a building.

3D View - List of Views

3D View allows you to save views of your 3D images for future use. This can be done through the **List of Views** dialog which allows you to save the 3D view that is currently being displayed in the viewing area and displays a list of all the previously saved views.

You have access to this dialog by selecting **View | List of Views...** from the menu or by clicking on the ... button on the Menu Toolbar.



List of Views dialog

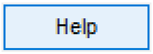
How to Use the List of Views

1. To view a saved view, simply click on the view you want and click the **Select** button.

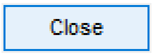
2. To save the 3D view that is currently being displayed in the viewing area, type a small description for the view you want to save in the **Save Current View** As box and click on the **Add** button. This view will then be listed in the **List of Views** dialog.

The following buttons are found in the **List of Views** dialog:

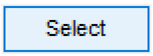
Adds the view you wish to save to the **List of Views**.



Displays the help for the **List of Views** dialog.



Closes the **List of Views** dialog.

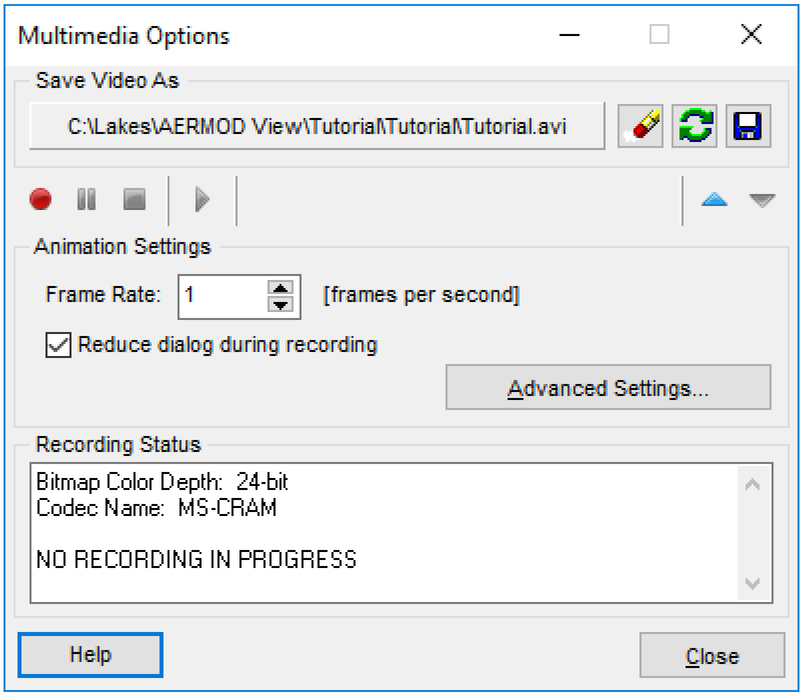


Displays the view you have selected.


Deletes the selected view.

3D View - Multimedia Options

You can access the **Multimedia Options** dialog by selecting **Multimedia | Record Animation...** from the menu.



Multimedia Options dialog

The following buttons are available from the **Multimedia Options** dialog:

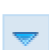

Start Recording: Begins recording the animation.


Pause Recording: Pauses the animation recording.


Stop Recording: Ends the recording of the animation.


Play Animation: Plays the recorded animation.


Reduce Dialog: Reduces the Multimedia Options dialog, allowing you access to only the dialog buttons.


Expand Dialog: Expands the Multimedia Options dialog, allowing you access to the entire dialog.

How to Record Animations:

1. Click on the **Save File As** () button to specify the name and location the created animation file (*.AVI) will be saved to. You can click on the **Clear File** () button to clear the specified file name and location if needed. Click on the **Get Default** () button to specify the default file name (i.e. <project>.AVI) and save the file to the project directory.

2. Select the Frame Rate by entering the number directly or by using the arrows. The **Frame Rate** refers to the number of frames per second you would like to record.

3. Check or uncheck the **Reduce dialog during recording** box. When you start recording, the **Multimedia Options** dialog will automatically reduce in size if this box is checked. This prevents the dialog from obstructing your animation.

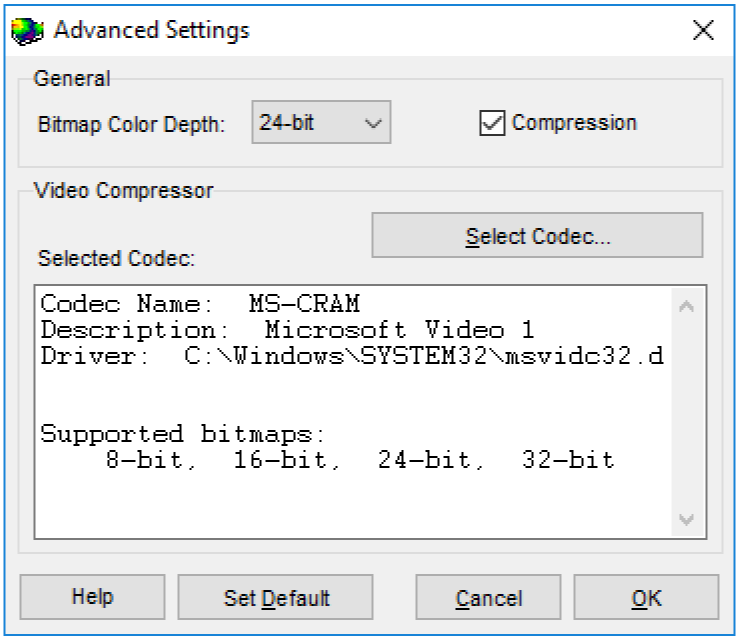
4. Click on the Advanced Settings... button to open the Advanced Settings dialog where you can define various advanced settings options.

5. Click on the button to start recording.

6. Carry out the desired functions you wish to record. Once you have finished, press the button to stop recording.

Advanced Settings

The **Advanced Settings** dialog provides you with additional settings for your multimedia recording options.



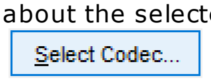
Advanced Settings dialog

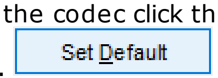
The following options are available:

- **Bitmap Color Depth:** Select a bitmap color depth from this drop-down list. The default is 24-bit.

- **Compression:** This box is checked by default. Compression refers to the amount your AVI file will be compressed.

> Creating AVI files can be demanding on a computer's resources, and usually creates quite large files. Increasing the amount of compression applied to animation recording can reduce the file size without seriously degrading the quality of the recording.

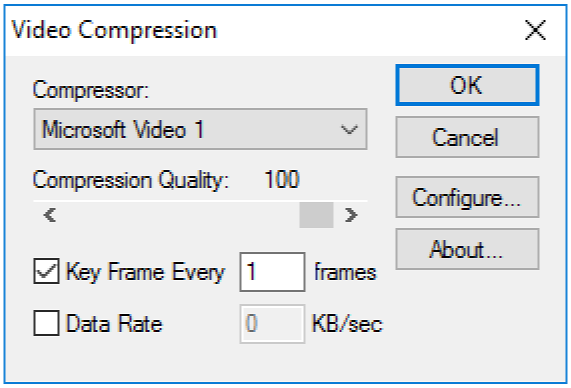
- **Video Compressor:** Some information about the selected codec is displayed in this panel. If you wish to change the codec click the 

button to open the Video Compression dialog. 

Click on this button to restore the codec defaults.

Video Compression

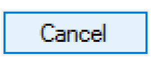
The following options are available in the **Video Compression** dialog:



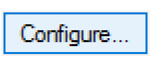
Video Compression dialog

- **Compressor:** Select a compressor from the drop-down list.
- **Compression Quality:** Here you can change the compression quality if you wish.
- **Key Frame Every:** You can check or uncheck this option and enter the number of frames.
- **Data Rate:** You can check or uncheck this option and enter the data rate.

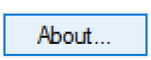
- Click to apply the codec currently selected in the **Compressor** field.



- Click to close the **Video Compression** dialog and keep the current codec.



- Click to load codec-specific dialog (if available) and configure related settings.

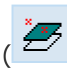


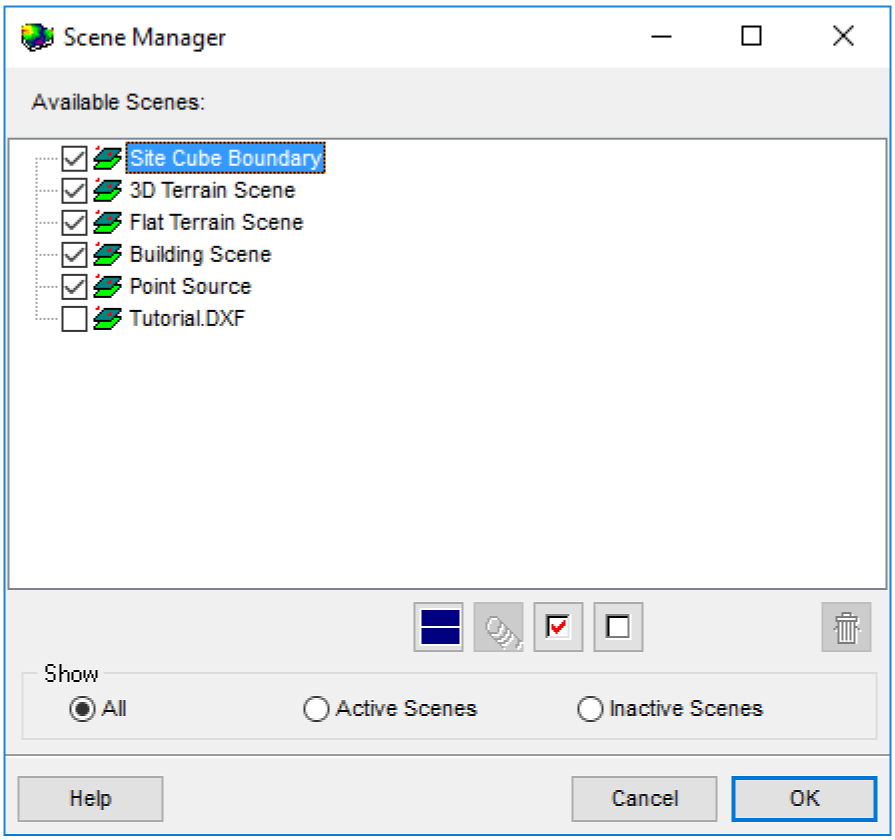
- Click to load and **About...** dialog for the currently selected codec.

3D View - Scene Manager

From time to time, during your project, you may wish to remove some scenes from the drawing area. For example, you can view or print only the desired objects in your drawing area. From the **Scene Manager** dialog, you can specify which objects should be displayed or not on the drawing area.

How to Turn On or Off the Display of Overlays:

1. Press the **Scene Manager** tool () located on the 3D Controls Toolbar or select **View | Scene Manager...** from the menu. The **Scene Manager** dialog is displayed.



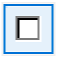
Scene Manager dialog

2. All the objects defined in your project are listed in this dialog. The checked/unchecked boxes indicates if the object is being displayed or not in the drawing area. You can easily turn on or off the display of these objects using the buttons located at the bottom of the dialog.

 **Select All:** Use this button to select all of the layers listed.

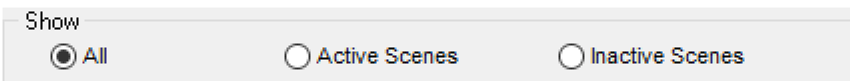
 **Unselect All:** Use this button to unselect all of the layers listed.

 **Check Selected:** Use this button to make all selected layers active.

 **Uncheck Selected:** Use this button to make all selected layers inactive.

 **Delete Selected:** Use this button to delete the selected layers. You can only delete the layers containing base maps.

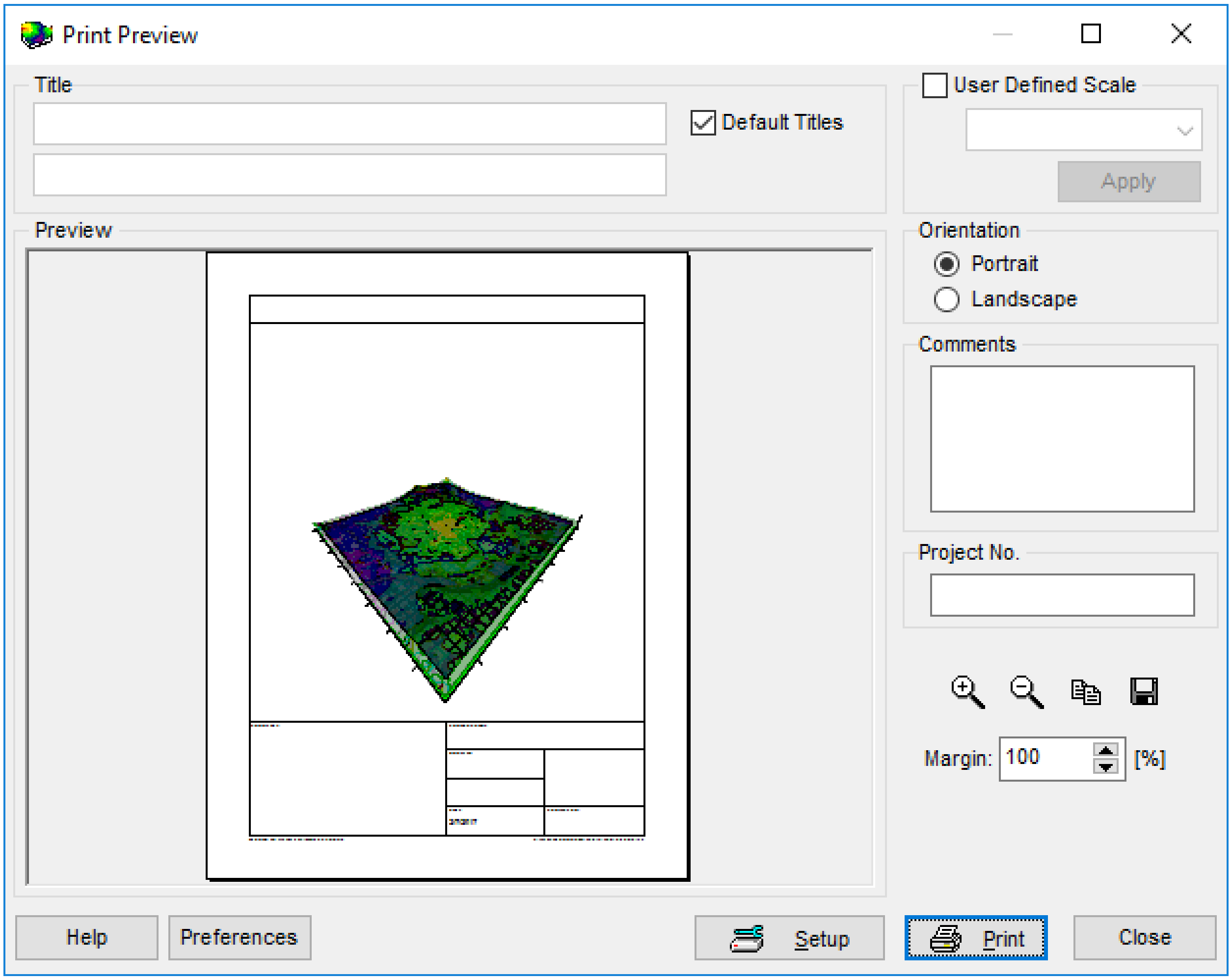
You can display the list of layers in different ways. You can show only the active scenes by selecting **Active Scenes**, or show only the inactive scenes by selecting **Inactive Scenes**.



3D View - Print Preview

The **Print Preview** dialog is displayed by clicking on the  button located on the **Menu Toolbar** or when you select **File | Print...** from the menu.

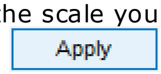
Before you can print the contents of the drawing area, AERMOD View displays the **Print Preview** dialog. Here you can select printing options and preview how your printouts will look. AERMOD View prints the contents of the drawing area into templates. These templates were designed so you can have important information about your project automatically added to your printouts.



3D View Print Preview dialog

The **Print Preview** dialog contains the following options:

- **Title:** AERMOD View places the name of your project as the default title. However, you have the option of specifying different titles. To specify different titles, uncheck the **Default Titles** box and type in the titles.

- **User Defined Scale:** Here you can the scale for your printout. The default scale used is 1:14848 but if you wish to change the scale, check the **User Defined Scale** box which then allows you to select the scale you wish to use from the drop-down list. You can also type in a scale and click on the 

button to apply it.

- **Orientation:** This is the orientation for your printout. Note that the preview area shows a preview of your results in both orientations- portrait and landscape.

- **Comments:** In this field you can type any comments or notes you want to be printed along with your printouts.
- **Project No.:** In this field you can specify your project number.
- **Margin:** Enter a percentage for the margin. The default is 100%. Specifying a percentage <100% will produce a narrower margin than the default. Specifying a percentage >100% will produce a wider margin than the default.

Button Guide

At the lower right corner of the dialog a series of buttons are available. See the function of these buttons below:

Zoom In: If you select this tool, your mouse pointer changes to a magnifying glass. On the Preview area, click on the location you want to zoom in. Click as many times as necessary, until you have the right magnification.

Zoom Out: Select this tool to go back to the original image size.

Copy to Metafile: Select this tool if you want to copy the image to the clipboard as a Microsoft® Windows® Metafile. You can then paste into any Windows application that supports pasting of a Windows Metafile from the clipboard.

Save to File: Select this option if you want to save the print preview image to a file. You can save the printout as an Enhanced Windows Metafile (*.emf) or a bitmap (*.bmp).

Displays the **Help** contents for the options contained in the **Print Preview** dialog.

Displays the Preferences dialog where you can specify printing and labeling options for your printouts.

Displays the **Print Setup** dialog where you can specify printing options such as paper size and orientation.

Displays the **Print** dialog where you can specify printing options such as printer and number of copies.

Closes the **Print Preview** dialog.

Tutorials

This section contains four tutorials that will help you get familiar with various aspects of AERMOD View and its accessory packages.

For best results complete the tutorials in order:

- AERMET View Tutorial
- RAMMET View Tutorial
- AERMOD View Tutorial
- Multi-Chemical Tutorial

AERMET View Tutorial

In order to conduct a refined air dispersion modeling project using the US EPA short-term air quality dispersion model - AERMOD, you need to process the meteorological data representative of the general area being modeled. The collected meteorological data needs to be preprocessed first using the U.S. EPA AERMET program.

This tutorial guides you through the basic steps towards preprocessing meteorological data files for use with the AERMOD model. Meteorological data files will be processed using AERMET View, which is a user-friendly interface for the AMS/US EPA AERMET program.

The **AERMET View Tutorial** has the following steps, which should be followed in the order provided below.

Contents:

AERMET Overview

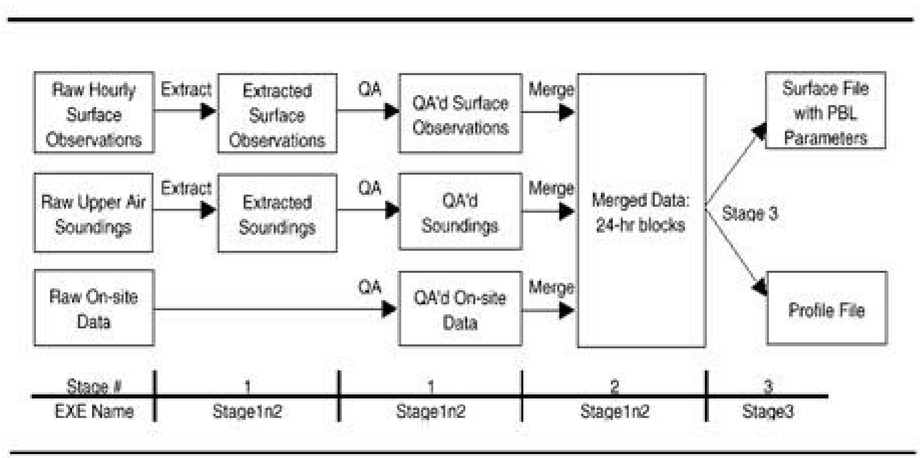
The US EPA AERMET program is a meteorological preprocessor which prepares hourly surface data and upper air data for use in the US EPA AERMOD short-term air quality dispersion model. AERMET was designed to allow for future enhancements to process other types of data and to compute boundary layer parameters with different algorithms.

AERMET pre-processes meteorological data in three stages and from this process two files are generated for use with the AERMOD model:

1. **Surface File (*.SFC):** contains hourly boundary layer parameters estimates;

2. **Profile File (*.PFL):** contains multiple-level observations of wind speed, wind direction, temperature, and standard deviation of the fluctuating wind components.

A flowchart depicting the AERMET processing stages is shown below:

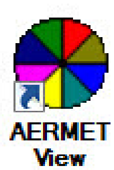


AERMET Processing Stages (Source: U.S. EPA User's Guide for AERMET – DRAFT)

Creating an AERMET View Project

Follow the steps below to create your AERMET View tutorial project:

Step 1: From the Windows **Start** menu choose **All Programs > Lakes Environmental > AERMOD View > AERMET View** or double-click on the **AERMET View** icon from the **Lakes Environmental** folder located on your desktop.



Step 2: The **About** dialog is displayed. Click the **OK** button.

Step 3: Select **File | New Project...** menu option and specify the location and file name for your project according to table below. Click **Save**.

Parameter	Value
Project Location	C:\Lakes\AERMOD View\My Tutorial\AERMET
Project Name	Tutorial.amf

If you only want to browse this tutorial, we have included the complete tutorial project, **Tutorial.amf**, under **C:\Lakes\AERMOD View\Tutorial\AERMET** folder.

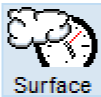
Minimum Met Data Requirements

When preprocessing meteorological data for use with the AERMOD model, the following are the minimum data input requirements for AERMET View:

Minimum Met Data Requirements	
Surface met data	Hourly surface observations for: • Wind speed • Wind direction • Dry bulb temperature • Cloud cover (tenths) You must have the all the above parameters in order for the AERMOD model to accept your preprocessed meteorological data. Although some met stations do not measure cloud cover, this is a required parameter for AERMOD and must be present in your met data.
Upper air met data	The upper air data are generally collected twice daily, at 0000 Greenwich Mean Time (GMT) and 1200 GMT (these times are also referred to as 00Z and 12Z, respectively).

Hourly Surface Data

The first window that opens when you create a new project is the **Hourly Surface Data** window. You can also go to this window any time by clicking the **Surface** toolbar button.

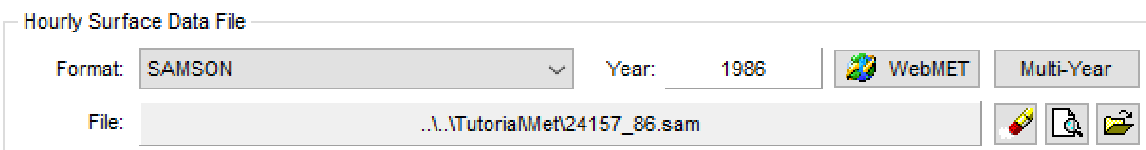


This window contains the following tabs:

- Hourly Surface Data
- ASOS 1-Minute
- QA Surface Variables
- Surface Variables Ranges

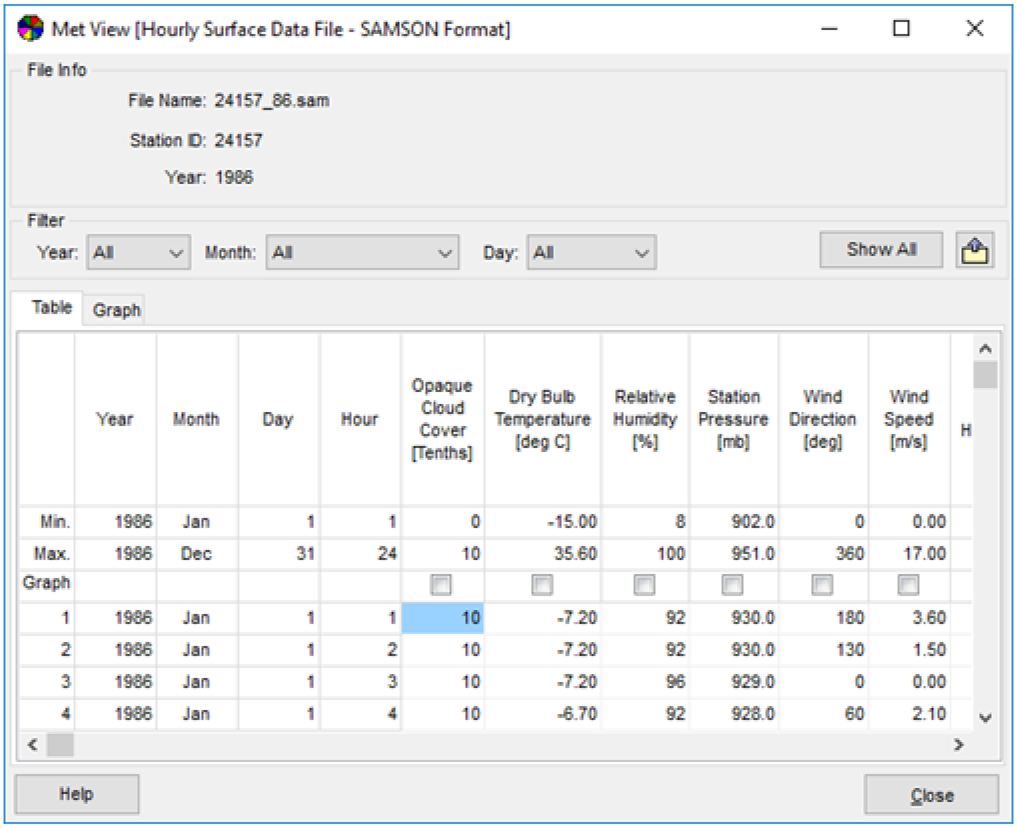
Follow the steps described below to build your tutorial project.

Step 1: In the **Hourly Surface Data File Name** panel you must specify the name and path for the hourly surface data file. Click the  button and specify the met file indicated in table below:



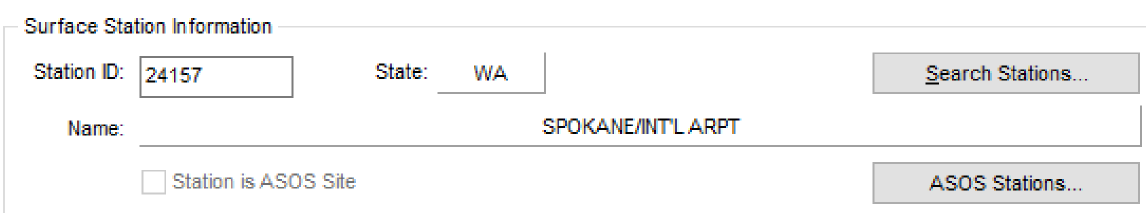
Parameter	Value
Folder Location	C:\Lakes\AERMOD View\Tutorial\Met
File Name	24157_86.sam
Format	SAMSON The format of the met file you specified is automatically detected by the AERMET View interface.

Step 2: Click the **Preview** button ( ) to view the surface met data file in either **Text** or **Grid** format. When finished reviewing data, click **Close**.

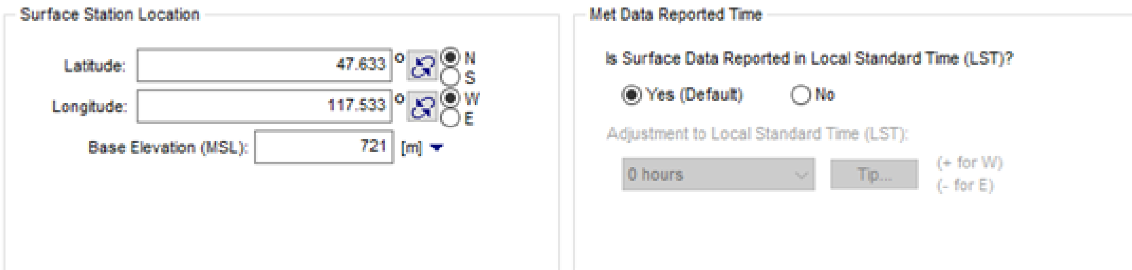


Surface Met Data Viewed in Grid Format

Step 3: The **Surface Station Information** panel displays the information for the specified hourly surface met data file. The surface station information should appear automatically for US NWS (National Weather Service) surface stations.



Step 4: The **Surface Station Location** panel displays the information for the surface station location from the hourly surface met data file. The latitude and longitude information for the station should appear automatically for US NWS surface stations. The **Met Data Reported Time** panel displays the time the met data was reported in.



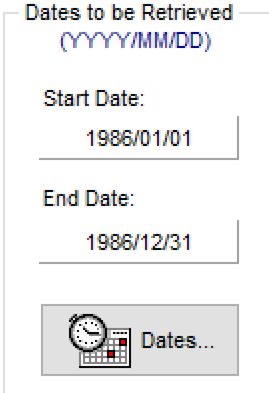
Parameter	Value
Latitude	47.633 N
Longitude	117.533 W
Station Base Elevation (MSL)	721 m MSL = Mean Sea Level
Is Surface Data Reported in Local Standard Time (LST)?	Yes
Adjustment to Local Standard Time (LST)?	No adjustment is necessary. **Note:** If you are using the TD3505 met data format, then you will need to specify the adjustment since this data format is reported in GMT time. In such cases, we recommend you to use

Warning: It is extremely important that the **Adjustment for Local Time** is correctly specified. Most surface met data are collected in local time in which case the 0hr (No Conversion) option must be specified.

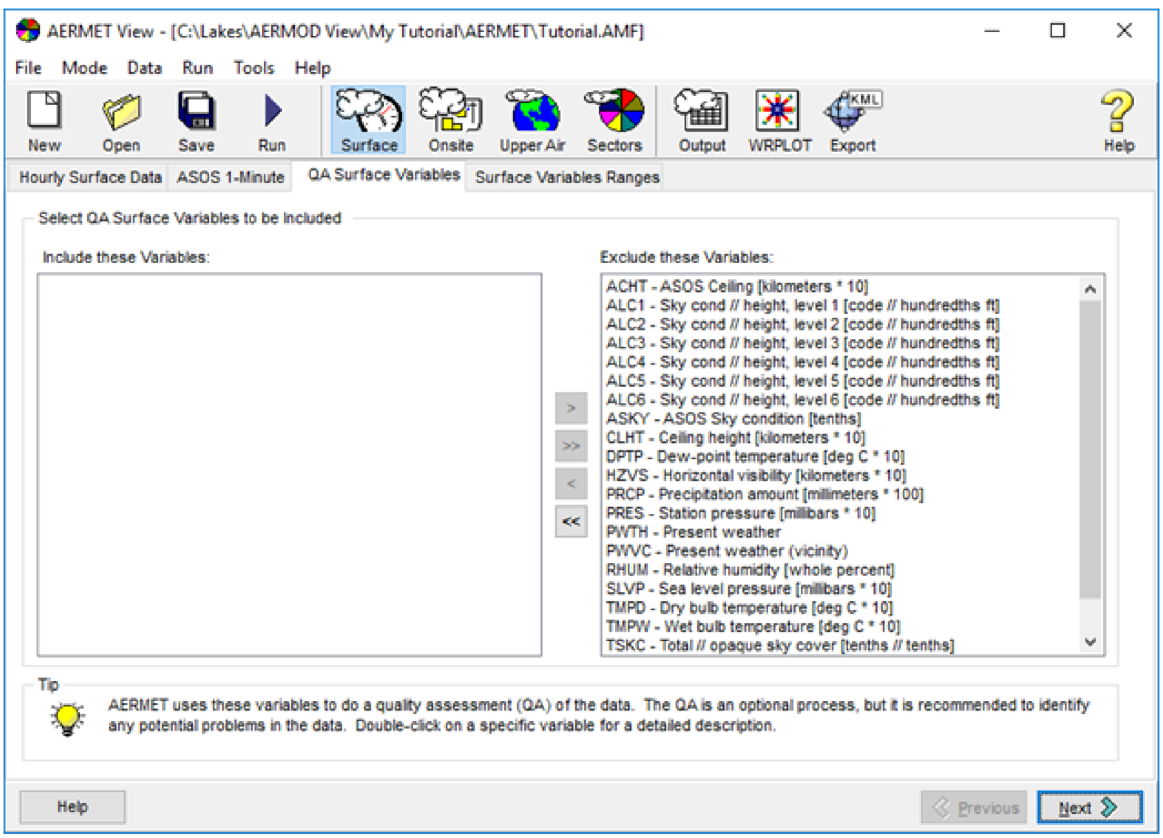
Step 5: AERMET View reads the start and end dates from the specified met data file and automatically sets the dates under the **Dates to be Retrieved** panel. Please make sure the dates are set to the following:

Parameter	Value
Start Date	1986/01/01 (January 01, 1986)

End Date	1986/12/31 (December 31, 1986)



Step 6: In the **QA Surface Variables** tab, you may select one or more variables to be audited. During the quality assessment (QA) process, audited variables are checked as being missing or outside the range of acceptable values. By double-clicking on a specific variable, you will be able to check the default ranges, description, units, and missing indicators for all variables.



QA Surface Variables tab

Step 7: Double-click on any QA surface variable to check its default values. For this tutorial, we are not carrying out any QA on the surface variables.

Upper Air Data

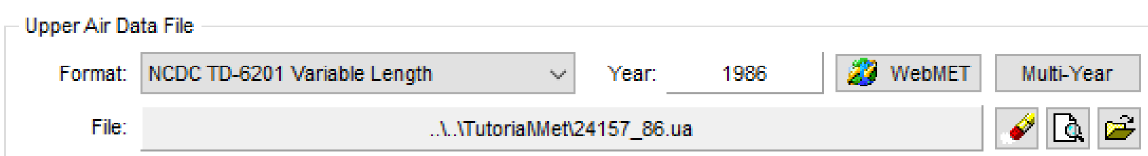
Click the **Upper Air** button from the menu toolbar. The **Upper Air Data** window is displayed.

The **Upper Air Data** window contains the following tabs:

- Upper Air Data
- QA Upper Air Variables
- Upper Air Variables Ranges

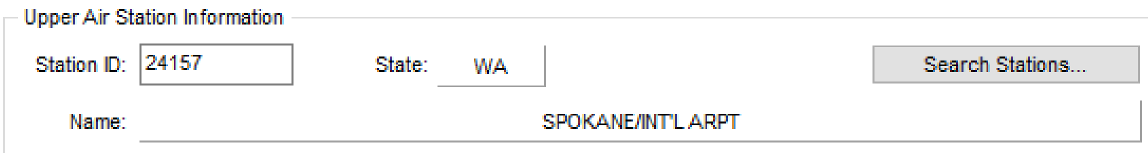
Follow the steps described below to build your tutorial project:

Step 1: In the **Upper Air Data File Name** panel you must specify the name and path for the upper air data file. Click the  button and specify the met file as provided in table below:

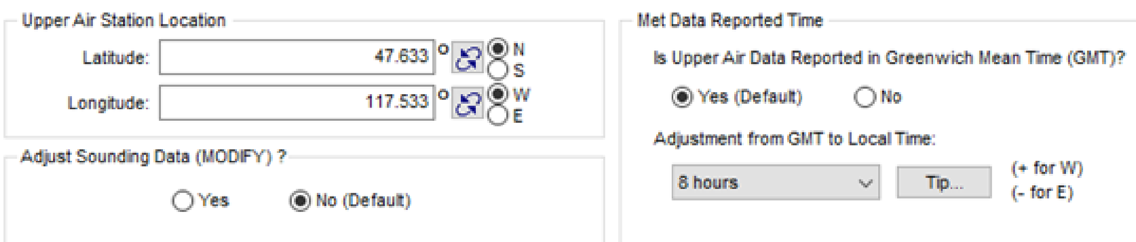


Parameter	Value
Folder Location	C:\Lakes\AERMOD View\Tutorial\Met
File Name	24157_86.ua
Format	NCDC TD-6201 Variable Length **Note:** The format of the met file you specified is automatically detected by the AERMET View interface.

Step 2: The **Upper Air Station** Information panel displays the information for the upper air station for the specified met data file. The station information (**Station ID**, **State**, **Name**) should appear automatically for NWS stations in the U.S.



Step 3: The **Upper Air Station Location** panel displays the information for the upper air station location. The latitude and longitude information for the station should appear automatically for US NWS stations. The **Met Data Reported Time** panel displays the time the met data was reported in.



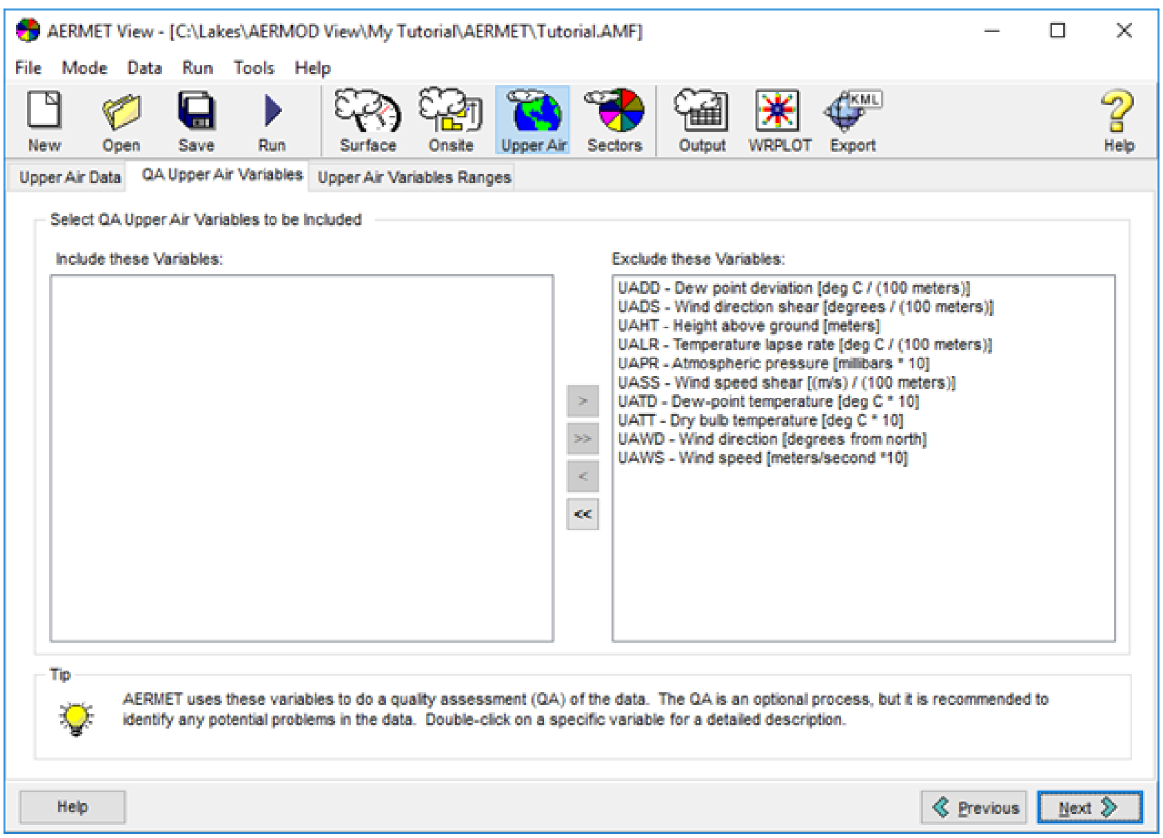
Parameter	Value
Latitude	47.633 N
Longitude	117.533 W
Adjust Sounding Data?	No
Is Surface Data Reported in Greenwich Mean Time (GMT)?	Yes
Adjustment from GMT to Local Time	8 hours **Note:** This is the number of hours required to convert the time of each met data record from GMT to local standard time (LST). Since the upper air file we are using is reported in Greenwich Mean Time (GMT), and the modeling site is at time zone -8 (Pacific Time), the adjustment should be set to 8 hours.

Warning: It is extremely important that the **Adjustment for Local Time** is correctly specified. Most upper air stations collect data in GMT, in which case the adjustment value must NOT be 0.

Step 4: AERMET View reads the start and end dates from the specified met data file and automatically sets the dates under the **Dates to be Retrieved** panel. Please make sure the dates are set to the following:

Parameter	Value
Start Date	1986/01/01 (January 01, 1986)
End Date	1986/12/31 (December 31, 1986)

Step 5: In the **QA Upper Air Variables** tab you may select one or more variables to be audited. During the quality assessment (QA) process, audited variables are checked for being missing or outside the range of acceptable values. By double-clicking on a specific variable, you will be able to check the default ranges, description, units, and missing indicators for all variables.

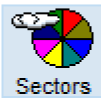


QA Upper Air Variables tab

Step 6: Double-click on any QA upper air variable to check its default values. For this tutorial, we are not auditing any upper air variables.

Processing Options

The **Sectors** window is displayed when you click the **Sectors** menu toolbar button.

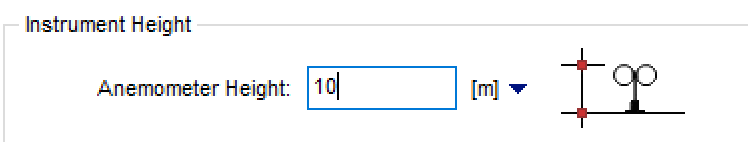


This window contains the following tabs:

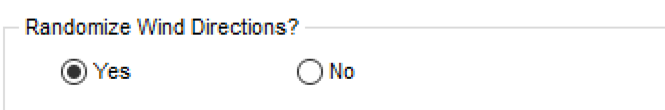
- Processing Options
- Sectors (Surface)
- Output Files

Under the **Processing Options** tab, specify the parameters following the steps below:

Step 1: In the **Instrument Height** panel enter **10 meters** and the **Anemometer Height**.



Step 2: In the **Randomize Wind Directions** panel select **Yes**.



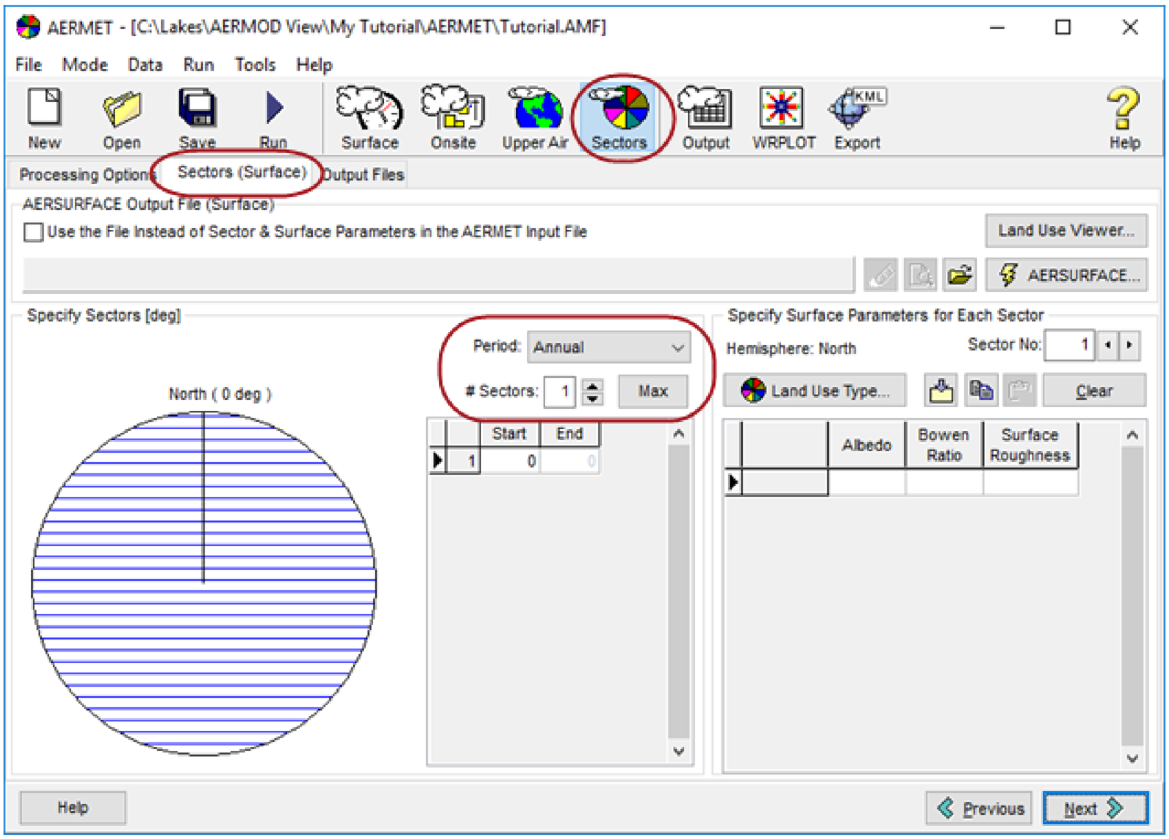
Sectors (Surface)

Surface characteristics at the measurement site influence boundary layer parameter estimates. These influences are quantified through the albedo, Bowen ratio, and surface roughness length. In order to better quantify these characteristics, you need to specify the frequency that these characteristics change (annual, seasonal, or monthly) and the number of different sectors.

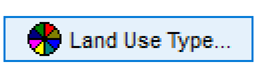
Under the **Sectors (Surface)** tab, specify the parameters following the steps below:

Step 1: A minimum of 1 and maximum of 12 sectors can be specified. Sectors are defined clockwise and they must cover the full circle. The end of one sector must be the beginning of another. For this tutorial, specify only **1 sector** and **Annual** frequency.

> The method used in this tutorial was simplified. We are going to specify only 1 section and annual values and choose a uniform land use (Grassland). For your future projects, you should define additional sectors to identify changes in the land use around the meteorological station (usually in an area within a 3km radius from the station).

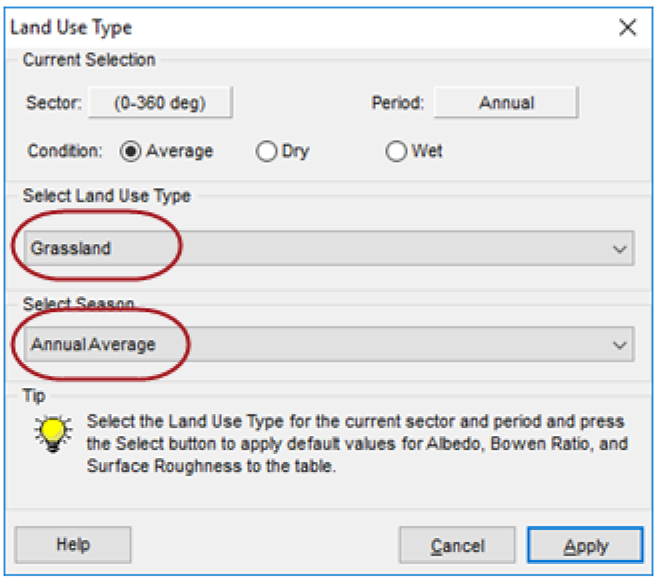


> In this tutorial, we are going to demonstrate how to manually select the surface parameters. For projects located in the **United States**, modelers should use the **AERSURFACE** program, which automatically calculates all the surface parameters, for each sector, based on the provided digital land use data file (USGS NLCD92 format).

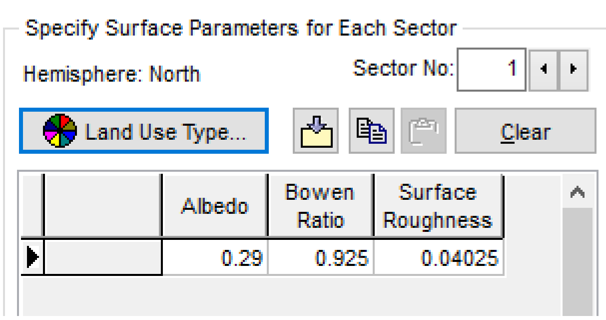
Step 2: Press the 

button and select the options below. Press the **Apply** button when done.

Parameter	Value
Condition	Average
Land Use Type	Grassland
Season	Annual Average **Note:** In the AERMET View tables, Winter is considered with snow cover. Annual Average values include Winter values.



Step 3: Values for the 3 surface parameters are automatically applied to the table.



> Typical values for albedo, Bowen ratio and surface roughness parameters are also available by clicking the ... button located on the right hand side of each cell. This button is only visible when you click on the cell.

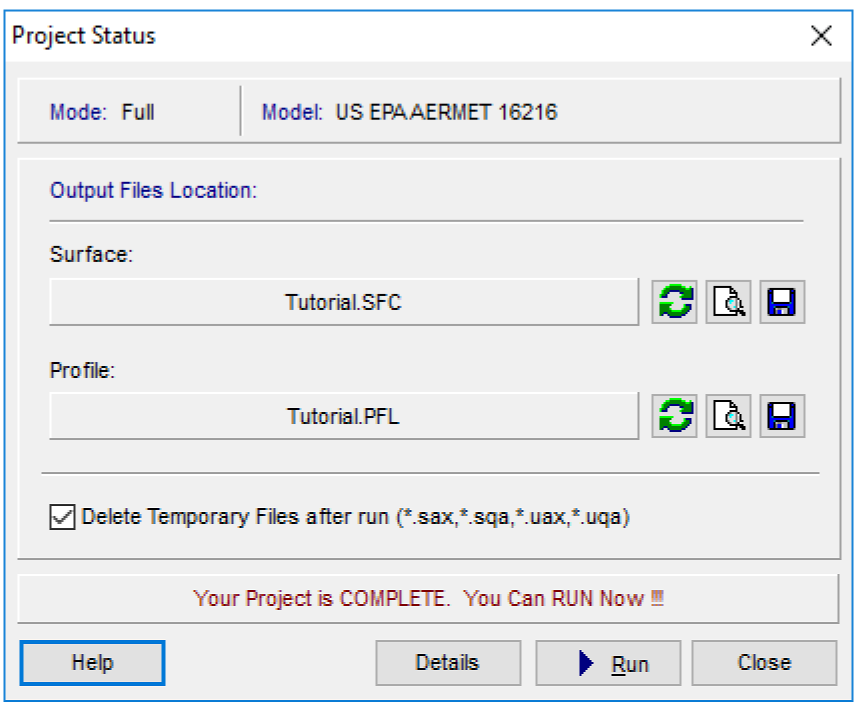
Running AERMET

Now that we have specified all the data for our tutorial project, it is time to run the U.S. EPA AERMET model. Before running your project, we suggest that you follow these steps:

Step 1: Click the **Run** menu toolbar button or select **Run | Run AERMET...** from the menu.

Step 2: The **Project Status** dialog is displayed informing you the name for the two output files that will be generated after AERMET runs. If your project is incomplete, you can click the **Details** button to get a brief description of the missing data.

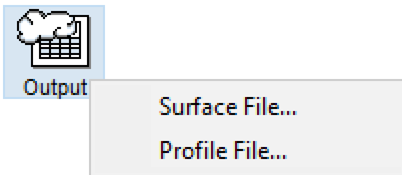
Step 3: Make sure the **Delete Temporary Files after run** box is checked. This will remove any temporary files that are created and placed by default in your project folder during the AERMET model run.



Project Status dialog

Step 4: When all the necessary information is supplied, you can click on the **Run** button located in the **Project Status** dialog.

Step 5: Once the run is finished and successful, you can preview the output files generated by AERMET by clicking the **Output** toolbar button and then selecting one of the menu options.



Step 6: You can now close your project and exit AERMET View. Your project will be automatically saved.

RAMMET View Tutorial

In order to conduct a refined air dispersion modeling using ISCST3 and ISC-PRIME short-term air quality dispersion models, you need to process the meteorological data representative of the general area being modeled. The collected meteorological data is not always in the format supported by these two models, therefore, the meteorological data needs to be preprocessed using the U.S. EPA PCRAMMET program.

This tutorial guides you through the basic steps towards preprocessing meteorological data files for use with the ISCST3 and ISC-PRIME models. Meteorological data files will be processed using RAMMET View, which is a user-friendly interface for the U.S. EPA PCRAMMET program.

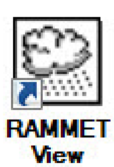
The **RAMMET View Tutorial** has the following steps, which should be followed in the order provided below.

Contents:

Creating a RAMMET View Project

Follow the steps below to create your RAMMET View tutorial project:

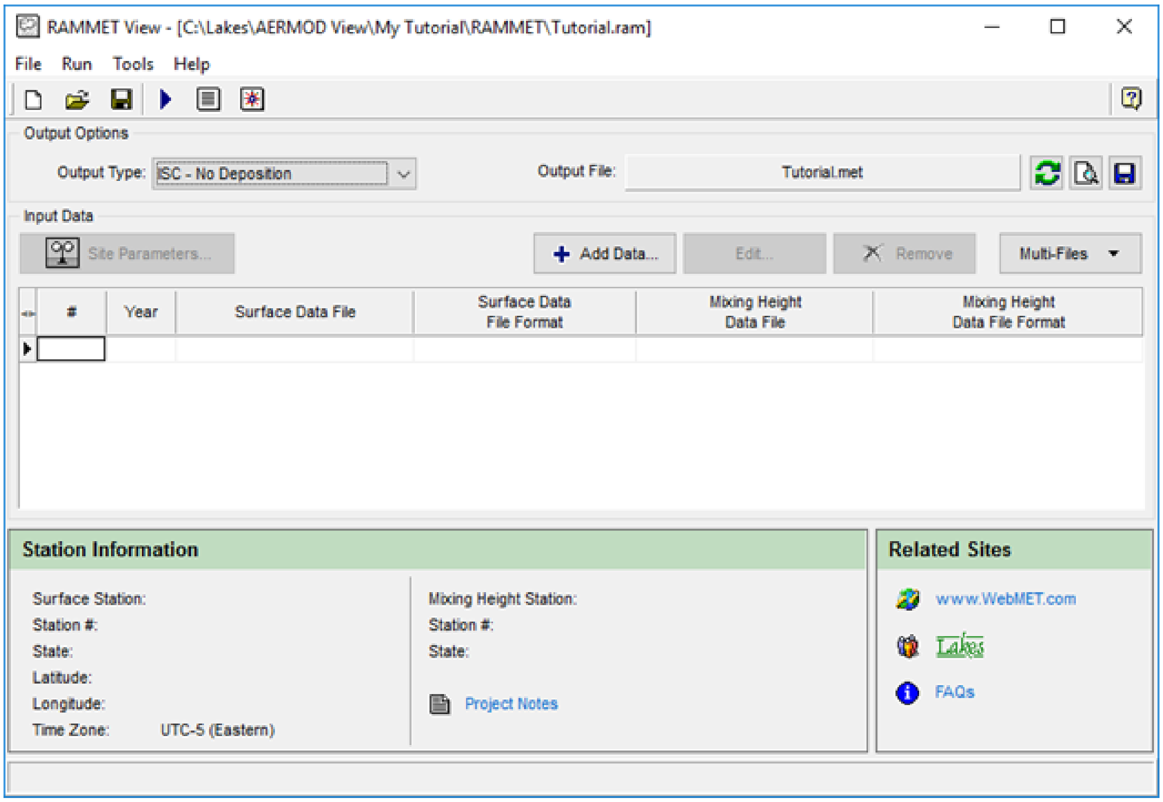
Step 1: From the **Windows Start** menu, choose **Programs > Lakes Environmental > AERMOD View > RAMMET View** or double-click on the **RAMMET View** icon, from the **Lakes Environmental** folder located on your desktop.



Step 2: The **About** dialog appears on the screen. Click the **OK** button.

Step 3: Select **File | New Project...** menu option and specify the location and file name for your project according to table below. Click the **Save** button when done.

Parameter	**Value**
Project Location	C:\Lakes\AERMOD View\My Tutorial\RAMMET
Project Name	Tutorial.ram



If you only want to browse this tutorial, we have included the tutorial project file, **Tutorial.ram**, in the Tutorial folder located by default in **C:\Lakes\AERMOD View \Tutorial\RAMMET** folder.

Minimum Input Data Requirements

The input data requirements for PCRAMMET depend on the dispersion model and the model options for which the data is being prepared. The minimum input data requirements for PCRAMMET are:

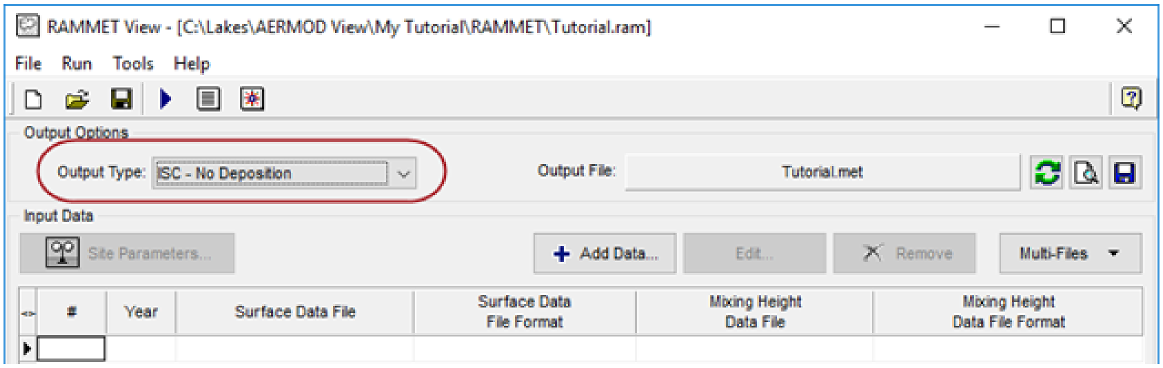
Minimum Met Data Requirements	
Mixing Height Data File	• AM Mixing Value • PM Mixing Value
Hourly Surface Data File	• Opaque Cloud Cover • Dry Bulb Temperature • Wind Direction • Wind Speed • Ceiling Height • Present Weather (For wet deposition only) • Hourly Precipitation (For wet deposition only) • Station Pressure (For dry deposition only)

When preprocessing meteorological data for use with the ISCST3 and ISC-PRIME models, the following are the minimum data input requirements for RAMMET View:

Parameter	**Calculation Type**		
	Concentration	**Dry Deposition**	**Wet Deposition**
Hourly Surface Data	x	x	x
Mixing Height Data	x	x	x
Additional Parameters		x	x
Precipitation Data			x

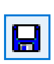
Output Options Panel

Step 1: For the RAMMET View tutorial, select **ISC - No Deposition** as the **Output Type**. Six options are available:



- **ISC - No Deposition:** This option should be selected if you are going to run ISCST3 and/or ISC-PRIME for concentration calculations only.
- **ISC - Dry Deposition:** This option must be selected if dry deposition calculations are going to be included in the ISCST3 and/or ISC-PRIME runs.
- **ISC - Wet Deposition:** This option must be selected if wet deposition calculations are going to be included in the ISCST3 and/or ISC-PRIME runs.
- **CALPUFF No Precipitation:** Select this option if the meteorological data is to be used to estimate dry deposition estimates in the CALPUFF Screening Mode.
- **CALPUFF With Precipitation:** Select this option if the meteorological data is to be used to estimate wet deposition estimates in the CALPUFF Screening Mode.
- **CALRoads View:** Select this option if the meteorological data is to be used for the CAL3QHCR model in CALRoads View.

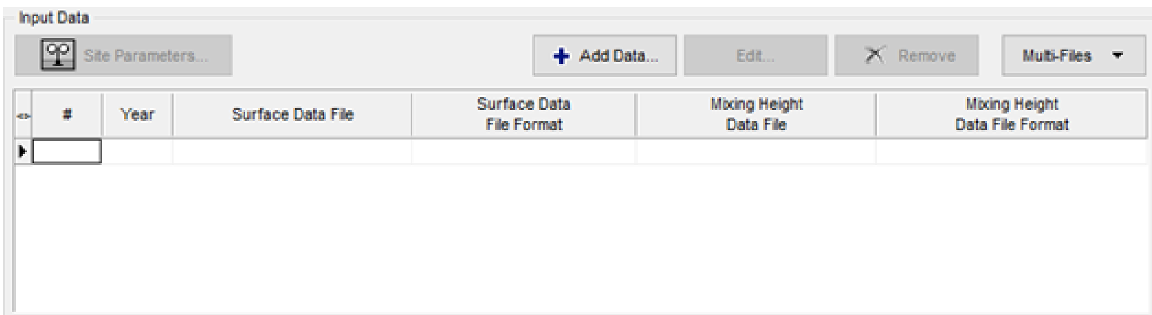
Step 2: The **Output File** panel is where you specify the name and the location of the preprocessed output file that is created after running PCRAMMET. By default, RAMMET View specifies the output file with the same name as the RAMMET View project but with extension **.met** (e.g., Tutorial.met). The location of this file is by default, in the project folder.

Click the  button if you want to specify a file name and/or a location different from the default. To get the default back, click on the  button.

Parameter	Value
Output Type	ISC - No Deposition
Output File	Tutorial.met

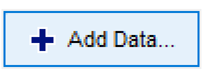
Input Data Panel

In the RAMMET View Tutorial we will be creating a preprocessed met file that spans five years. We will specify the five years of data that is required in the **Input Data** panel.

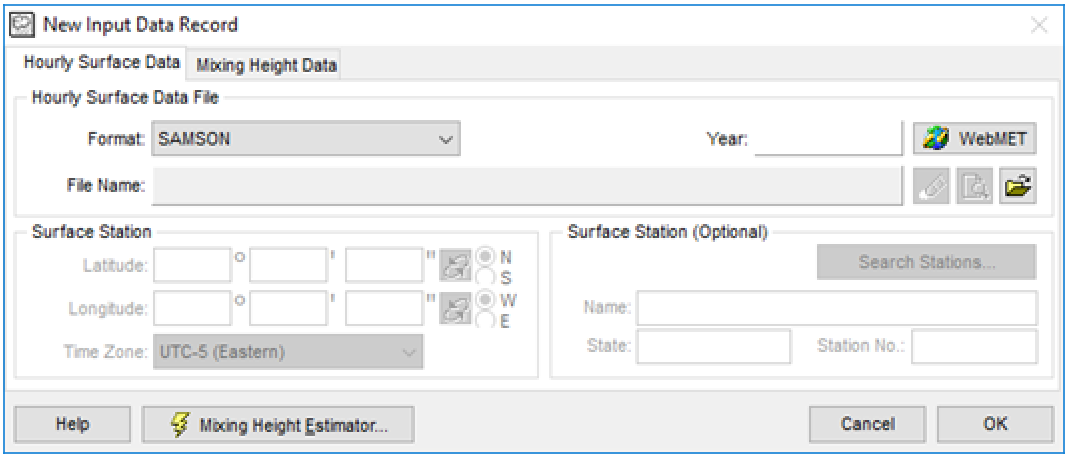


Input Data panel

Step 1: To add the met data files to your project, click the **Add Data...** button.

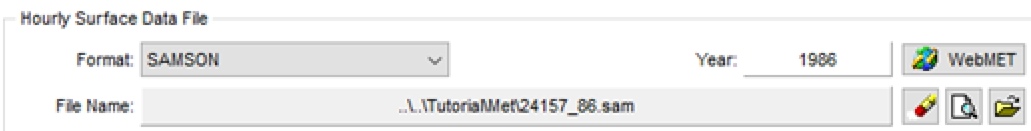


Step 2: The **New Input Data Record** dialog is displayed. This dialog contains two tabs, the **Hourly Surface Data** tab and the **Mixing Height Data** tab.



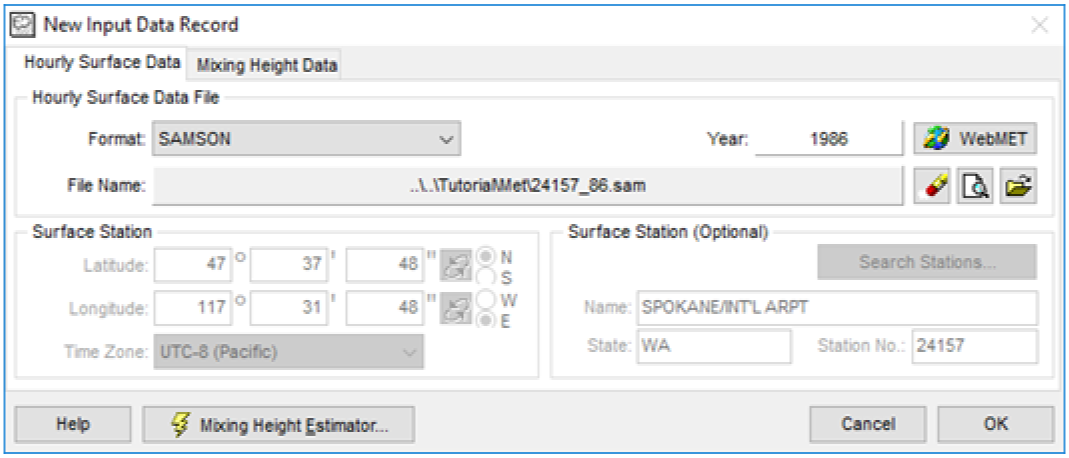
New Input Data Record dialog

Step 3: In the **Hourly Surface Data File Name** tab you must specify the hourly surface met data file. Click the  button and specify the met file indicated in the table below:

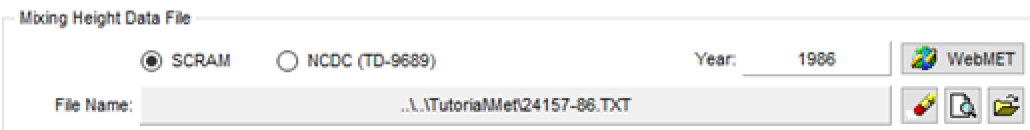


Parameter	**Value**
Folder Location	C:\Lakes\AERMOD View\Tutorial\Met
File Name	24157_86.sam
Format	SAMSON **Note:** The format of the met file you specified is automatically detected by RAMMET View.
Year	1986

Step 4: The **Surface Station** panel displays the information for the surface station for the specified hourly surface met data file. The latitude and longitude information for the station should appear automatically for NWS surface stations in the U.S. as well as the surface station information (**Station Name**, **State**, **Number**).



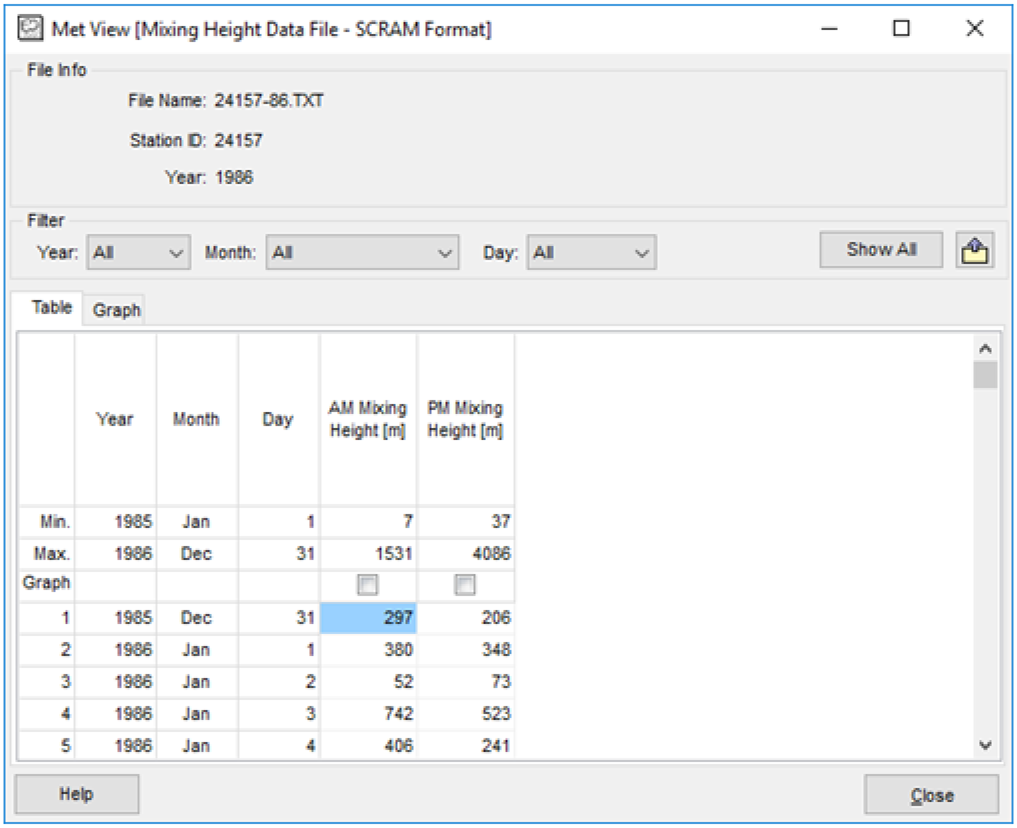
Step 5: In the **Mixing Height Data File** tab you must specify the mixing height data file. Click the  button and specify the file indicated in the table below:



Parameter	**Value**
Folder Location	C:\Lakes\AERMOD View\Tutorial\Met
File Name	24157-86.TXT
Format	SCRAM **Note:** The format of the met file you specified is automatically detected by RAMMET View.
Year	1986

It is extremely important that the **Hourly Surface Data File** and the **Mixing Height Data File** cover the same date period.

Step 6: Click the **Preview** button () to view the mixing height data file in either a **Text** or a **Grid** format.



Mixing Height Data Viewed in Grid Format

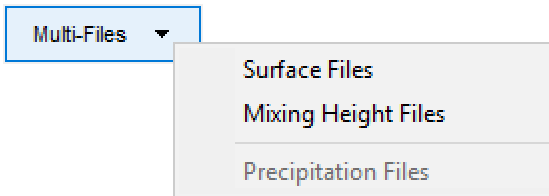
Step 7: The **Mixing Height Station** panel displays the information for the mixing height station for the specified mixing height met data file. The mixing height station information (**Station Name**, **State**, **Number**) should appear automatically for NWS surface stations in the U.S.

Step 8: Click on the **OK** button to exit the **New Input Data Record** dialog once you have finished specifying the met data files.

Multi-Files Tool

The **Multi-Files** tool can also be used to specify input data, and can select files for several years at once. We will enter the remaining four years of data using this tool.

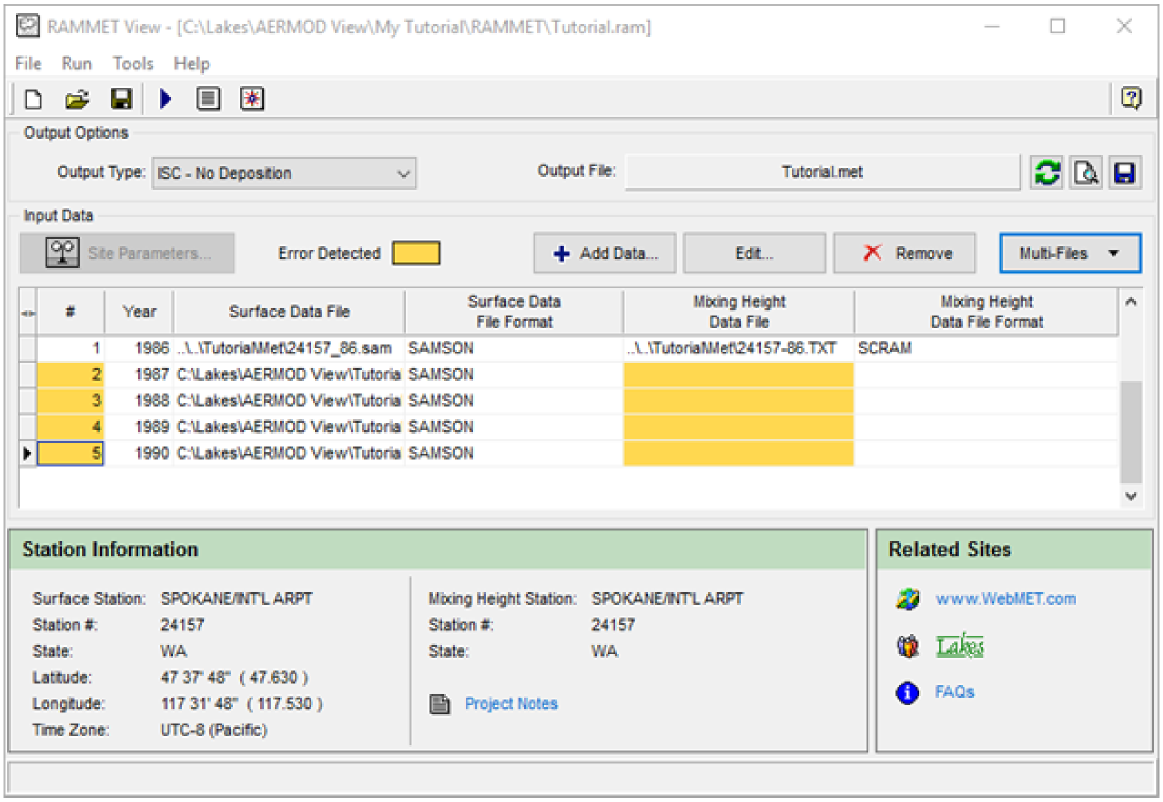
Step 1: Click the **Multi-Files** button. This will open the following drop down menu.



Step 2: Select the **Surface Files** option. This will display the **Specify Hourly Surface Data File** dialog from where you can specify multiple surface files. Select the following hourly surface data files:

Parameter	Value
Folder Location	C:\Lakes\AERMOD View\Tutorial\Met
File Names	24157_87.SAM 24157_88.SAM 24157_89.SAM 24157_90.SAM
Format	SAMSON **Note:** The format of the met file you specified is automatically detected by RAMMET View.
Years	1987 1988 1989 1990

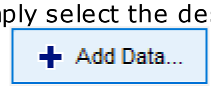
Step 3: Five years of surface met data should now be displayed in the main RAMMET View window as shown in image below. Any missing data will be highlighted in yellow. Four of the five years are highlighted in yellow as no Mixing Height Data File has been specified for those years.



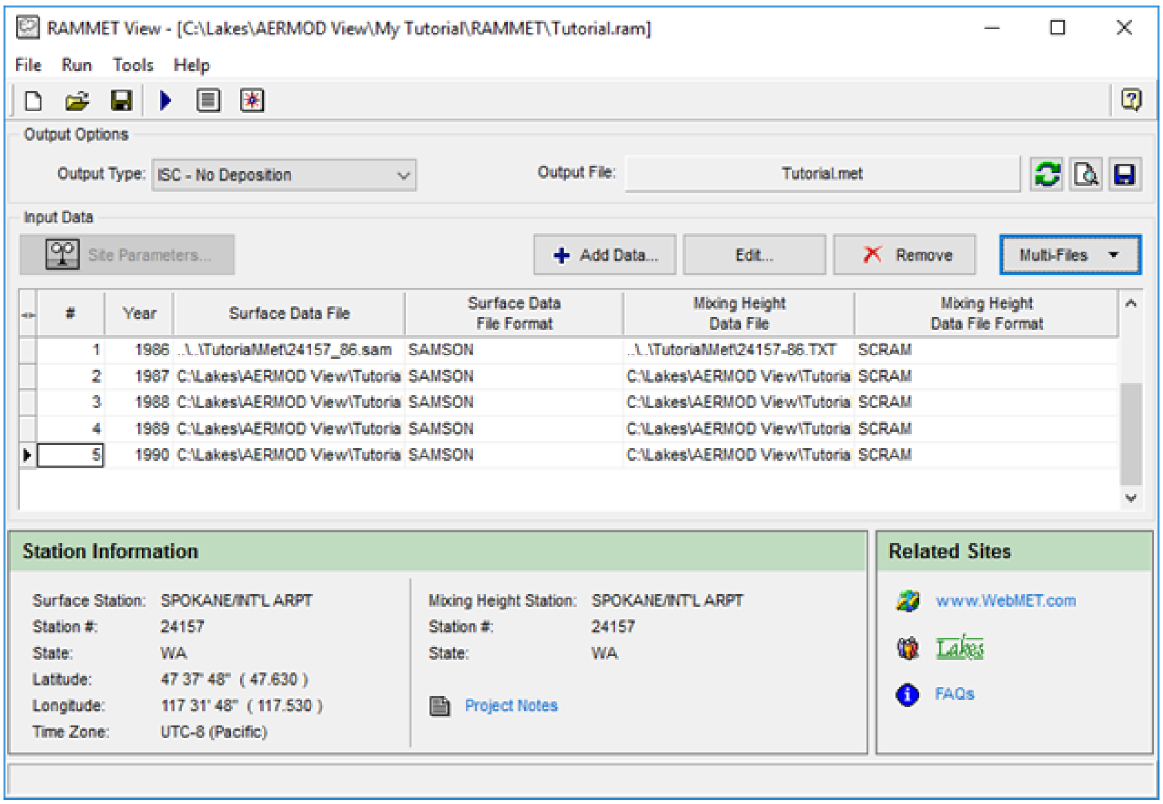
Step 4: Select the **Multi-Files** button and choose the **Mixing Height Files** option; this will display the **Specify Mixing Height Data File** dialog from where you can specify multiple surface files. Select the following mixing height data files:

Parameter	**Value**
Folder Location	C:\Lakes\AERMOD View\Tutorial\Met
File Names	24157_87.TXT 24157_88.TXT 24157_89.TXT 24157_90.TXT
Format	SCRAM **Note:** The format of the met file you specified is automatically detected by RAMMET View.
Years	1987 1988 1989

Parameter	Value
	1990

Step 5: The main RAMMET View window will now show complete input data for five years. If you wish to edit any of these entries, simply select the desired year and click the **Edit** button. This will open the same dialog as the 

button, but for the existing entry.

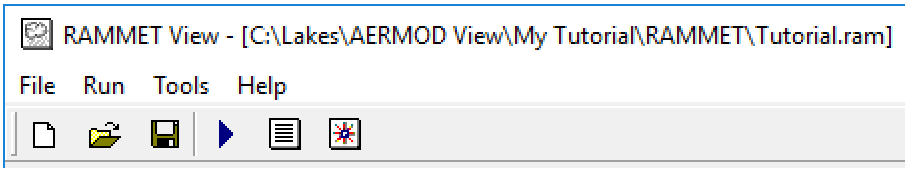


Five Complete Years of Met Data

The station information for the specified met files is displayed under the **Station Information** panel at the bottom of the RAMMET View main window.

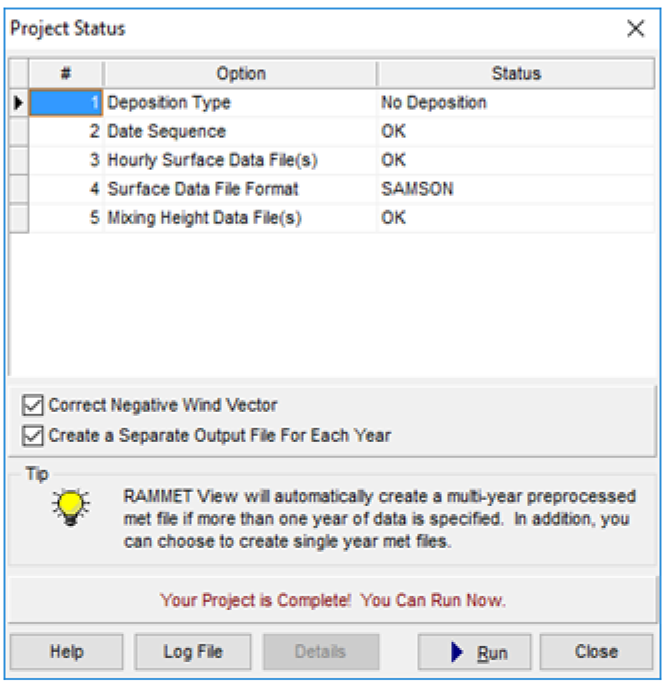
Running PCRAMMET

Your project should be complete by now. Before running your project, we suggest that you follow the steps below:



Step 1: Check the project status to make sure your options are correct and if any information is missing. The **Project Status** dialog is displayed every time you click the button or select the **Run | Run PCRAMMET...** menu option.

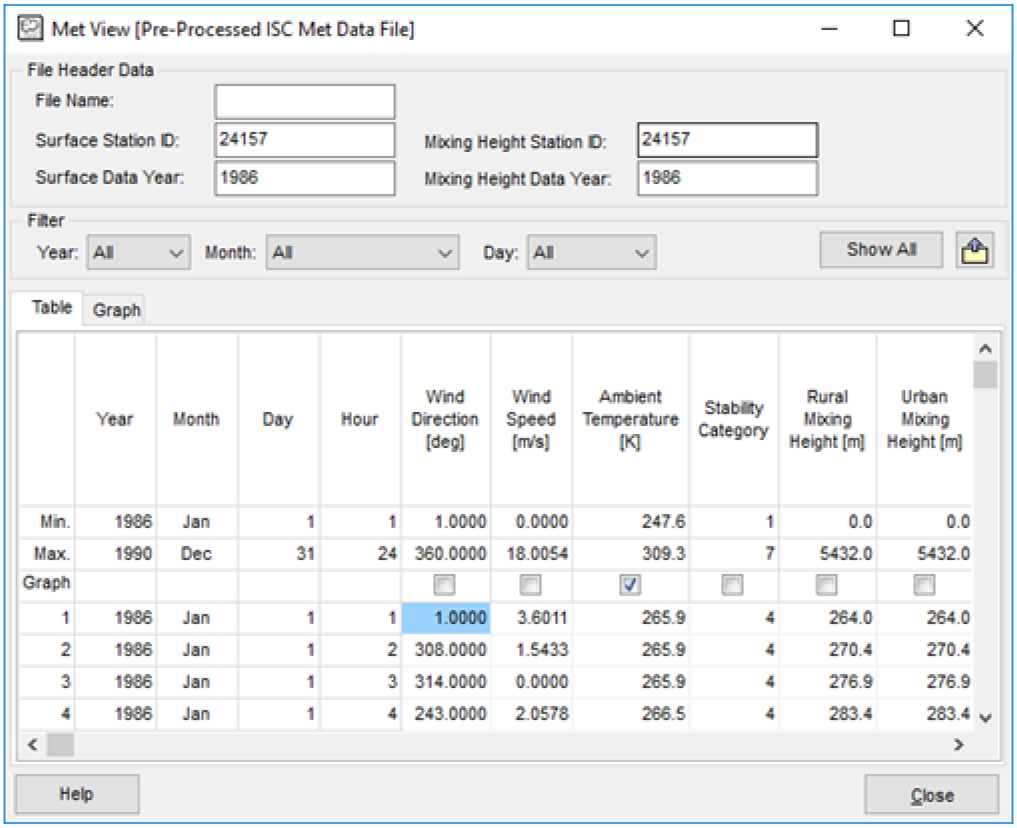
Step 2: Check the **Correct Negative Wind Vector** option. Make sure the **Create a Separate Output File For Each Year** option is also checked. This option creates preprocessed met files for each of the specified years in addition to the multi-year file that is automatically created. The single year files are named as ***year_stationnumber.met***. For example, the 1986 file will be named "86_24157.met".



Project Status dialog

Step 3: When all the necessary information is supplied, you can click on the **Run** button located at the bottom of the **Project Status** dialog.

Step 4: Once the run is finished and successful, you have the option of viewing the output file. If you choose not to view the output file, you can later view it in either text or grid format by clicking the button, or selecting **Run | Output File** from the menu. See below the contents of the output file viewed in grid format.



Met View grid

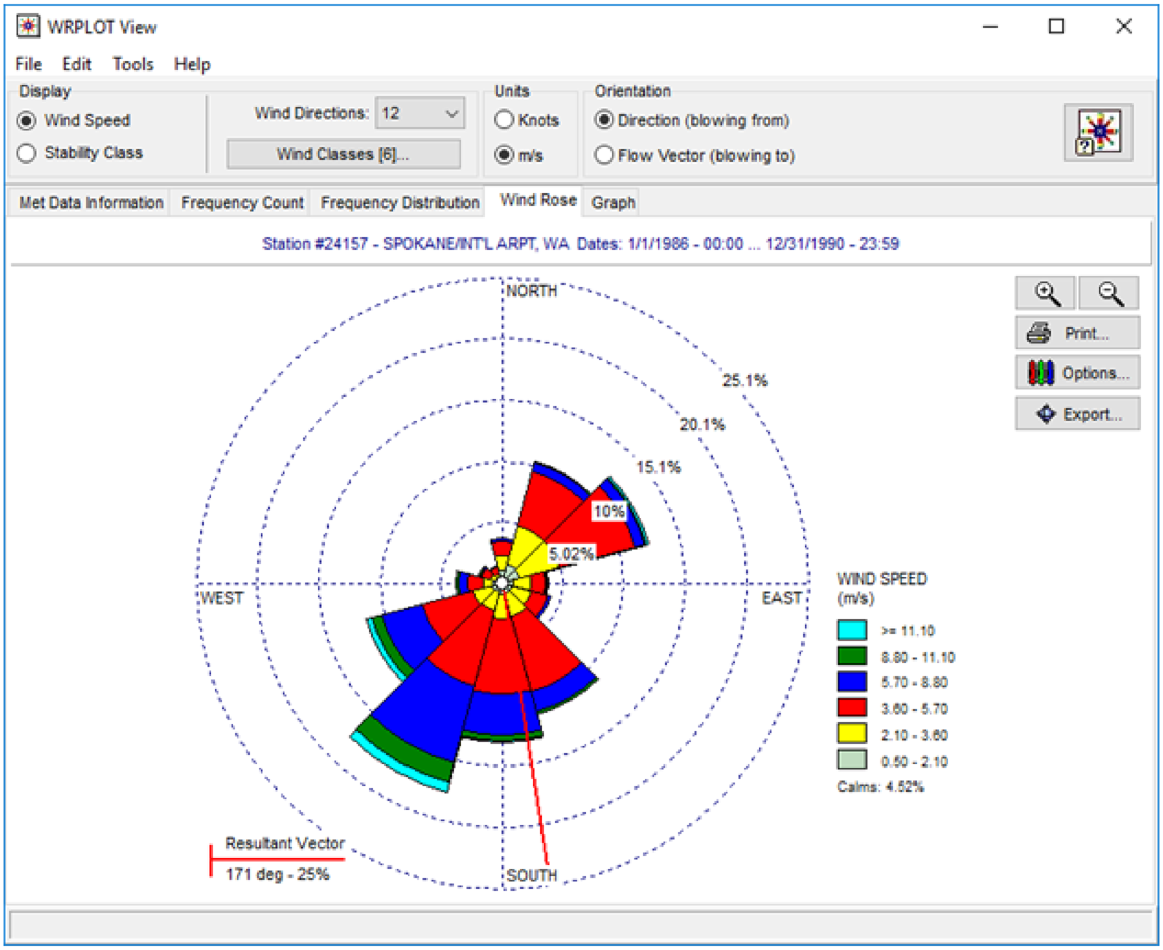
WRPLOT View Options

WRPLOT View is a program that generates wind rose statistics, frequency tables, and graphs for a wide variety of met data file formats (SCRAM, CD144, HUSWO, SAMSON, etc.), and for ISC preprocessed met data files.

From **RAMMET View**, you have access to **WRPLOT View** by clicking on the button.

Step 1: If you have already run your RAMMET View project, then once you click the **WRPLOT** toolbar button a pop-up menu is displayed with two file options: **Preprocessed Output File** and the **Hourly Surface Data File**. Select the **Preprocessed Output File** option.

Step 2: **WRPLOT View** is opened with the selected file already loaded. In the top section of the WRPLOT View main window, you can select several options related to the display of the visualization of the met data such as number of wind directions, units, orientation, among others. Change some of the options and see how the information on the various tabs changes.



WRPLOT View Main Window

WRPLOT View main window also contains the following tabs:

1. The **Met Data Information** tab is where you can add your met data files, view the information for each met file and specify date and time ranges for the met data.

2. The **Frequency Distribution** tab displays the normalized frequency of occurrences of winds for each direction sector and for each wind class for the selected date and time period. The normalized frequency multiplied by 100 gives you the percent frequency.

3. The **Frequency Count** tab displays, in tabular form, the number of occurrences of winds for each direction sector and for each wind class for the selected date and time period.

4. The **Wind Rose** tab displays in graphics the frequency distribution of occurrences of winds for each direction sector and for each wind class for the selected Date and Time period.

5. The **Graph** tab displays two bar charts showing the wind class frequency distribution and the stability class frequency distribution for the selected date and time period. The stability class frequency distribution chart is only available if the specified met data contains stability class information.

Step 3: Close **WRPLOT View** and **RAMMET View**. Your RAMMET View Tutorial project is complete.

AERMOD View Tutorial

This tutorial will guide you through the basic steps to develop an ISCST3, ISC-PRIME and AERMOD project using AERMOD View. This tutorial assumes that you have some familiarity with air dispersion modeling.

The AERMOD View Tutorial has the following steps, which should be followed in the order provided below.

Contents:

The Situation

XYZ Company wants to obtain a permit to operate a chemical plant in a rural area. The effluent from the facility, SO_2, is released to the atmosphere through two stacks. A building is located close to the stacks.

An air dispersion modeling study using the U.S. EPA AERMOD model needs to be conducted to find out the impact from the chemical plant emissions to the atmosphere, the significance of these impacts, and the area being impacted.

This tutorial requires files generated under the **AERMET View** and **RAMMET View** tutorials. It is advisable that you complete these tutorials before starting the **AERMOD View** tutorial. See below the required files:

Meteorological Files Preprocessed with AERMET View	
Surface Met File	Tutorial.SFC
Profile Met File	Tutorial.PFL
Folder Location	C:\Lakes\AERMOD View\My Tutorial\AERMET

Meteorological Files Preprocessed with RAMMET View	
Met File	Tutorial.met
Folder Location	C:\Lakes\AERMOD View\My Tutorial\RAMMET

The New Project Wizard

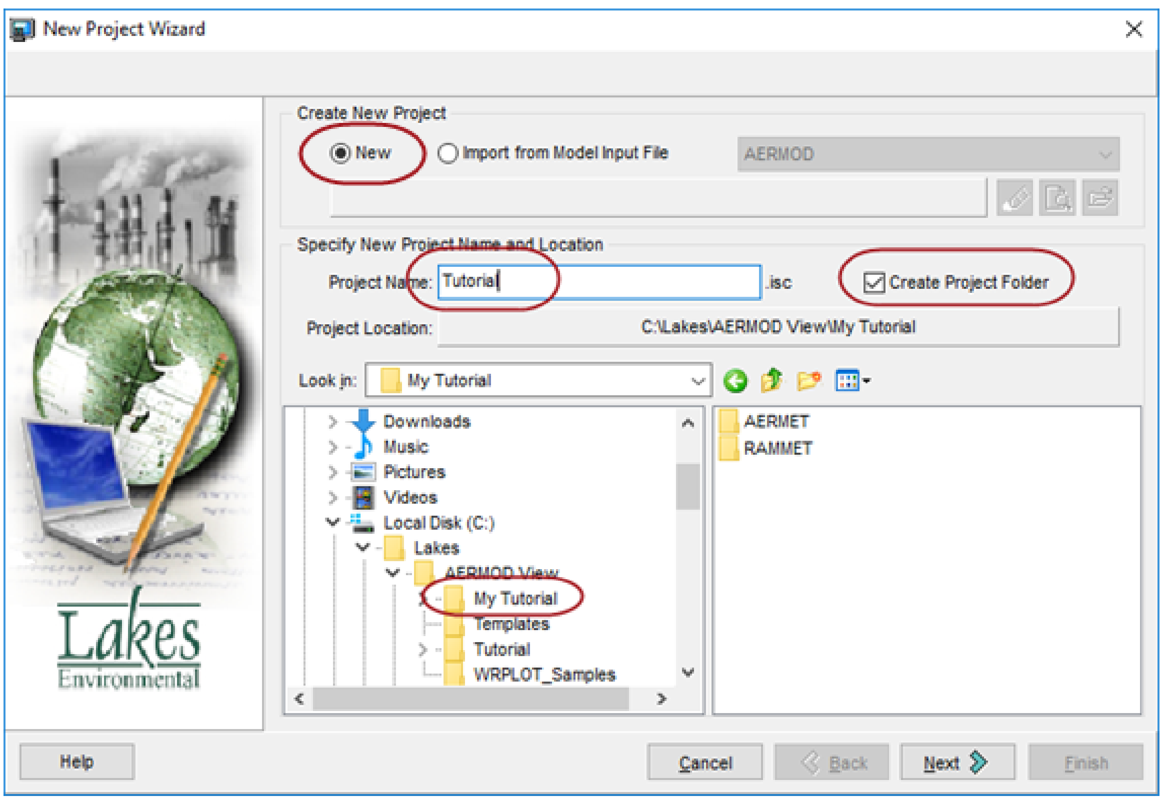
Follow the steps below to create your AERMOD View Tutorial project:

Step 1: From the **Windows Start** menu, choose **Programs > Lakes Environmental > AERMOD View** or double-click on the **AERMOD View** icon from the **Lakes Environmental** folder located on your desktop.

Step 2: The **About** dialog appears on the screen. Click the **OK** button and the AERMOD View initial window is displayed.

Step 3: Select **File | New Project...** from the menu. The **New Project Wizard** is displayed. In the first panel, specify the following options:

Parameter	Value
Create New Project	New
Project Name	Tutorial
Create Project Folder	☑ **Note:** It is recommended that you create each AERMOD View project in a separate folder.
Project Folder	C:\Lakes\AERMOD View\My Tutorial\

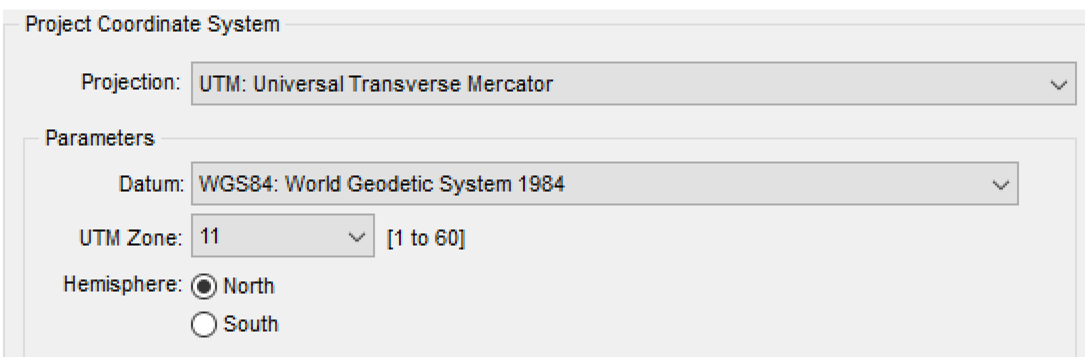


If you only want to browse this tutorial, we have included the tutorial project, **Tutorial.isc**, located under **C:\Lakes\AERMOD View\Tutorial** folder.

Step 4: Click the 

button to display the next panel. Under this panel, specify the following parameters:

Parameter	Value
Projection	UTM: Universal Transverse Mercator
Datum	NAD83: North American Datum 1983
UTM Zone	11 North

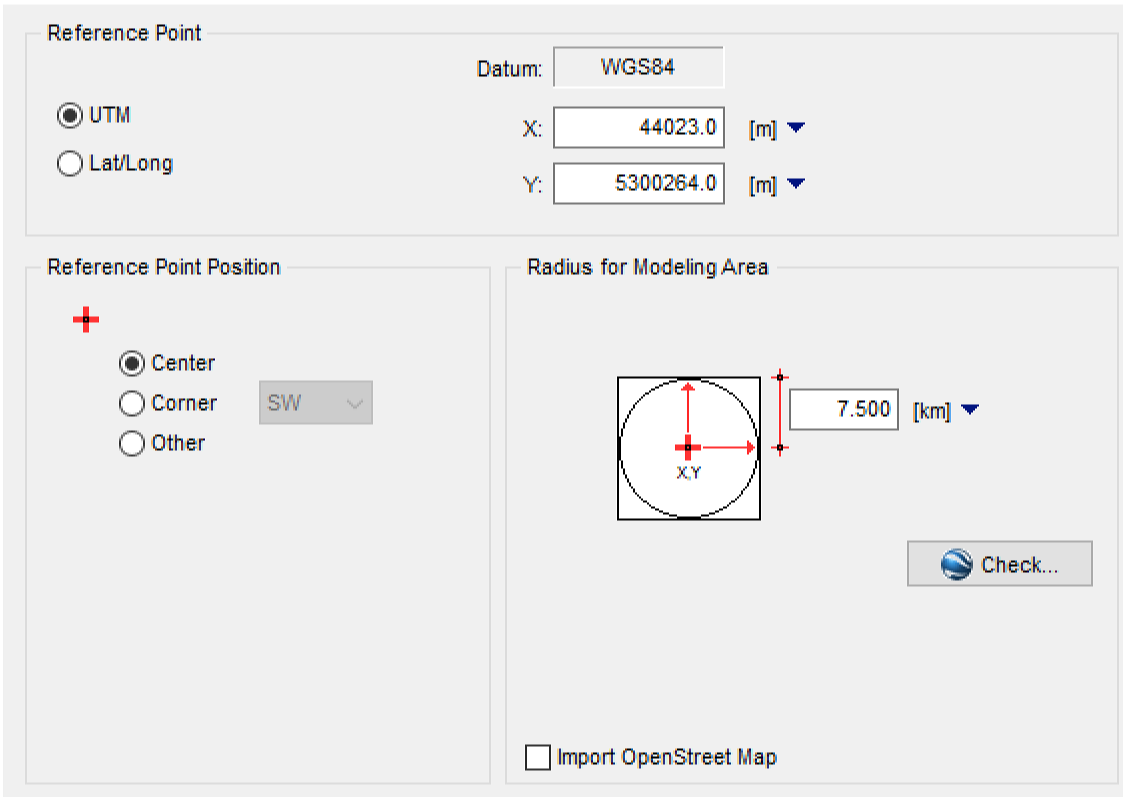


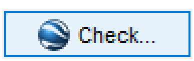
Step 5: Click the 

button to display the next panel. In this panel you will specify a reference point within your modeling area and the dimensions of your modeling area.

Parameter	Value
Reference Point	UTM
UTM X	442023.00
UTM Y	5300264.00
Reference Point Position	Center

Parameter	Value
Radius	7.5 km



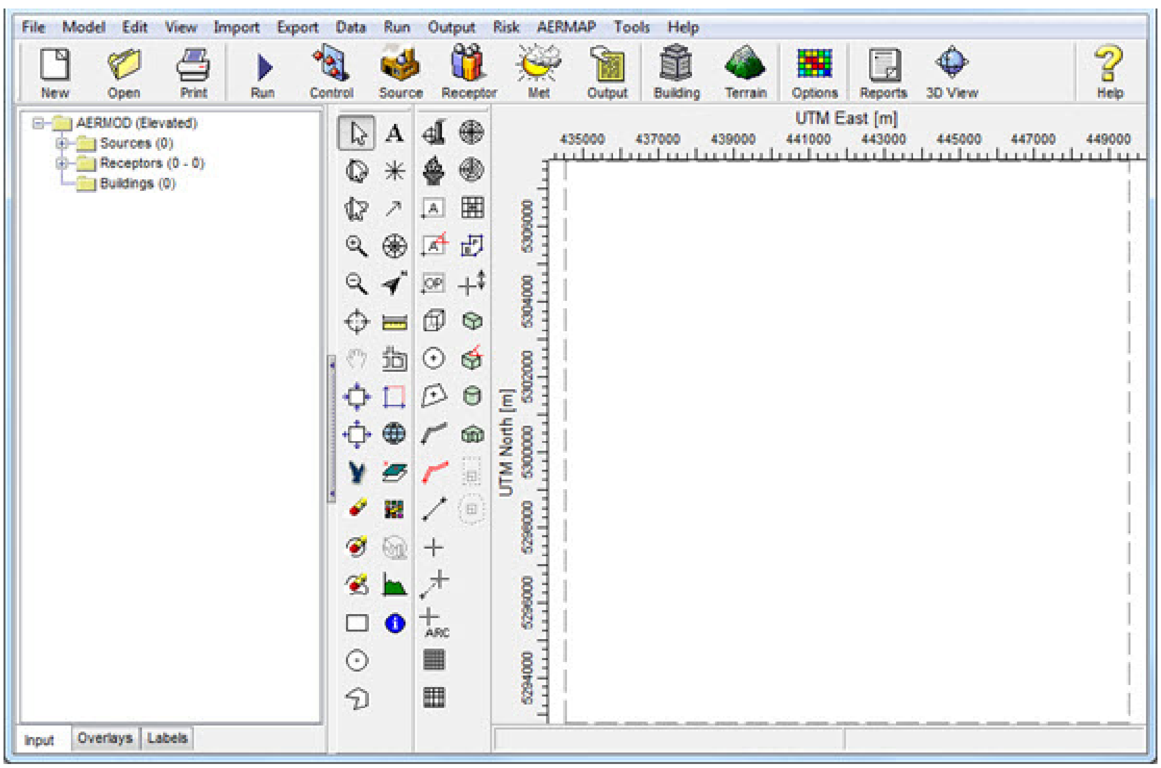
Step 6: Click the 

button to check the position of your modeling site in Google Earth™.

Step 7: In Google Earth, you should see the location of the Reference Point and the specified modeling domain as seen in image below.

Modeling domain

Step 8: Return to AERMOD View **New Project Wizard** and click the Finish button. The AERMOD View main window is displayed.



Importing Base Maps

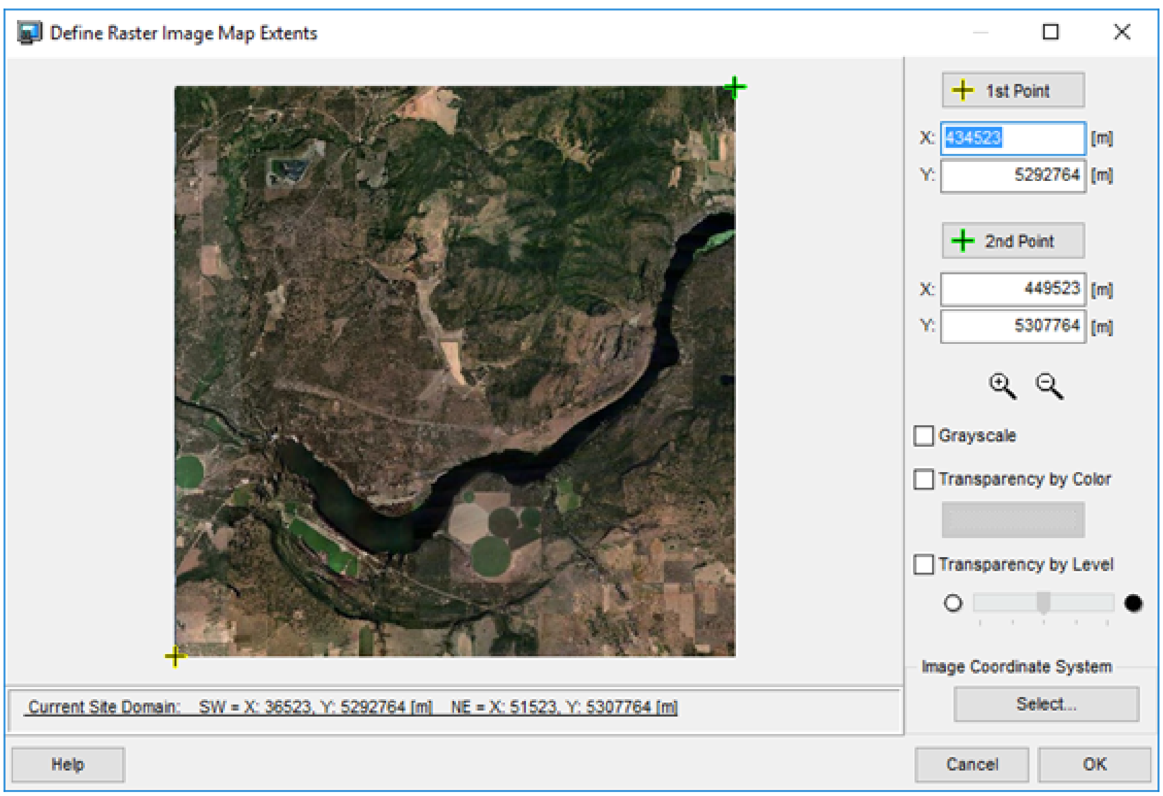
Base maps allow the modeler to quickly identify the location of the site, sources, buildings and other important characteristics of the surrounding area. For this project, we are going to import JPEG raster maps created in Google Earth and a DXF showing the location of the stacks and onsite building.

Follow the steps below on how to import base maps into your project:

Step 1: Select the menu option **Import | Base Maps | Raster Images** and select the following base map:

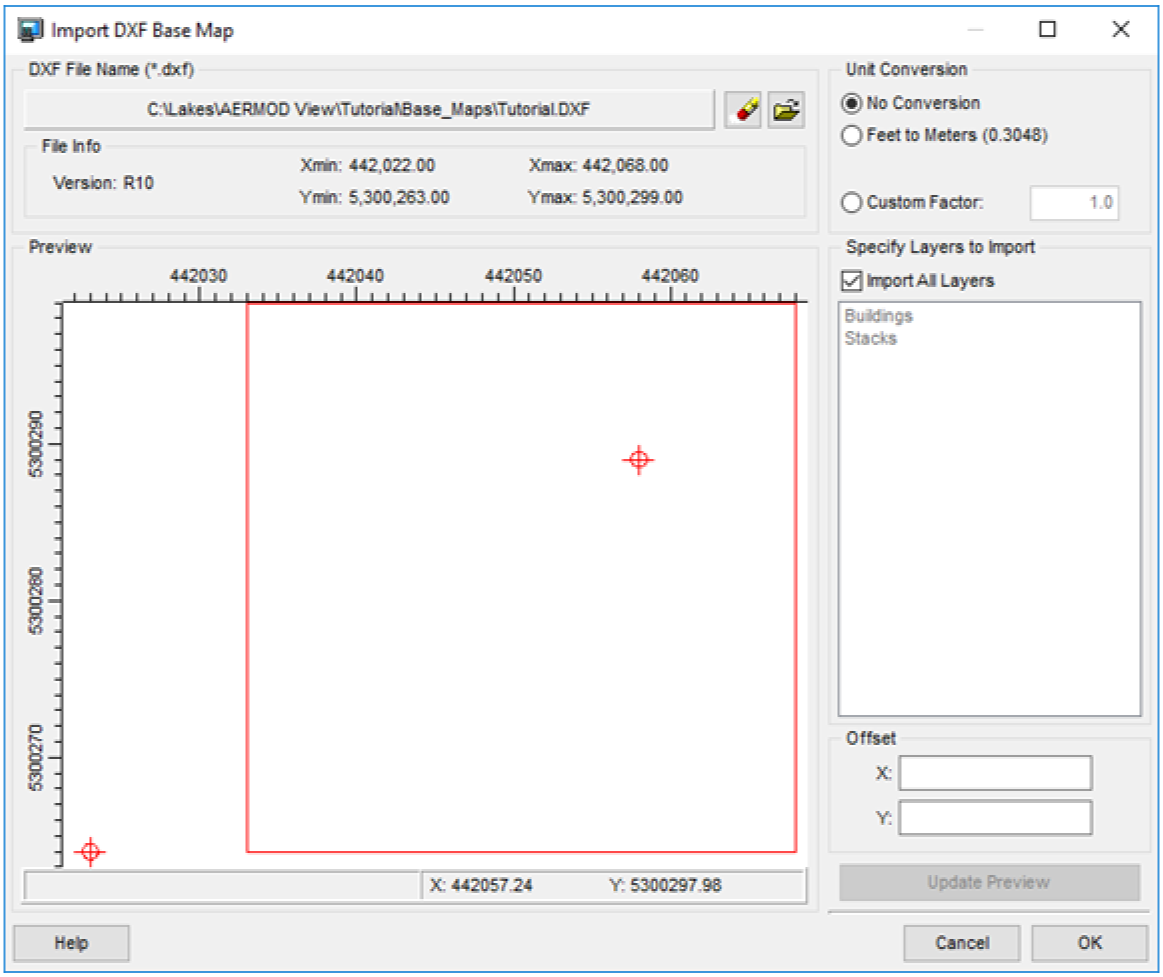
Parameter	Value
Folder Location	C:\Lakes\AERMOD View\Tutorial\Base_Maps
File Name	Tutorial_Map_15km.jpg

Step 2: The **Define Raster Image Map Extents** dialog is displayed. Since this raster file was already geo-referenced for you, just click the **OK** button.

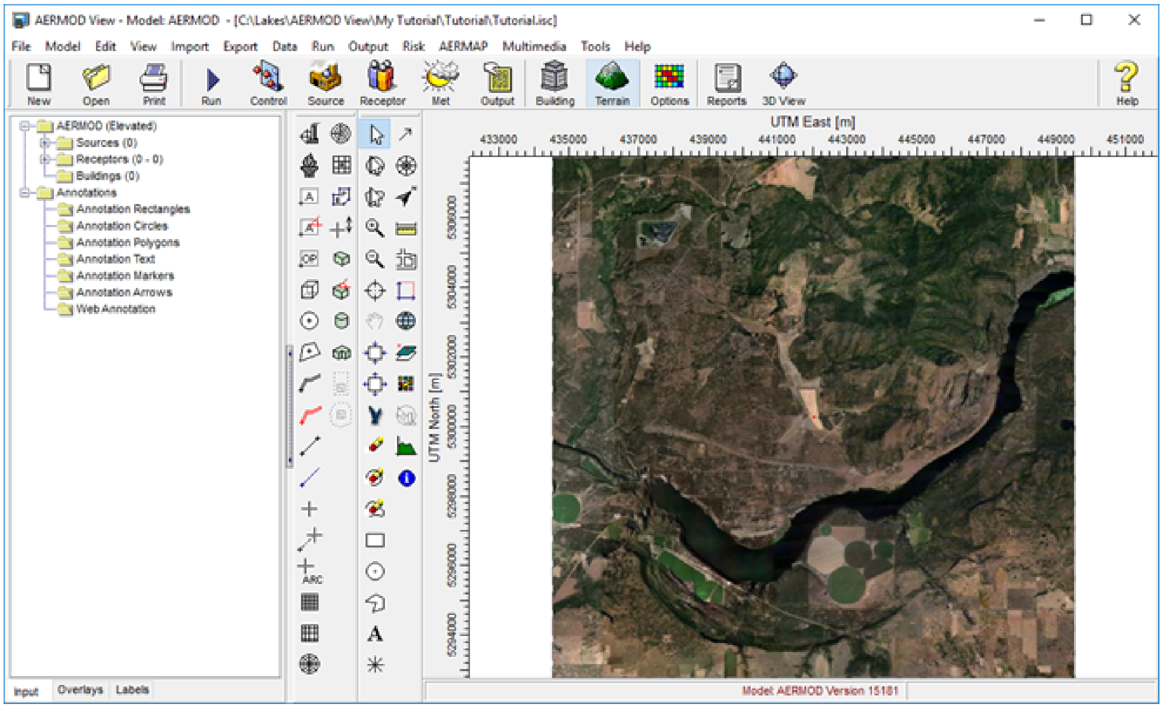


Step 3: Now you need to import the DXF file. Select the menu option **Import | Base Maps | DXF**. In the **Import DXF Base Map** dialog, click the  button and select the file specified in table below.

Parameter	Value
Folder Location	C:\Lakes\AERMOD View\Tutorial\Base_Maps
File Name	Tutorial.DXF



Step 4: Click the **OK** button to close the **Import DXF Base Map** dialog. You will see that the DXF base map is displayed in red at the center of the modeling domain.



Defining the Building

You now need to define the building in your project. There are three ways to define the location and dimensions of your buildings:

- Graphical Mode
- Text Mode
- Import from an AutoCAD DXF File

For this tutorial, we will define the building in graphical mode. As a guideline, we will use the DXF site map you imported, containing the location and exact dimensions of the building.

Step 1: Zoom in to the area where the building is located using the **Zoom In** tool () from the Annotation Toolbar.

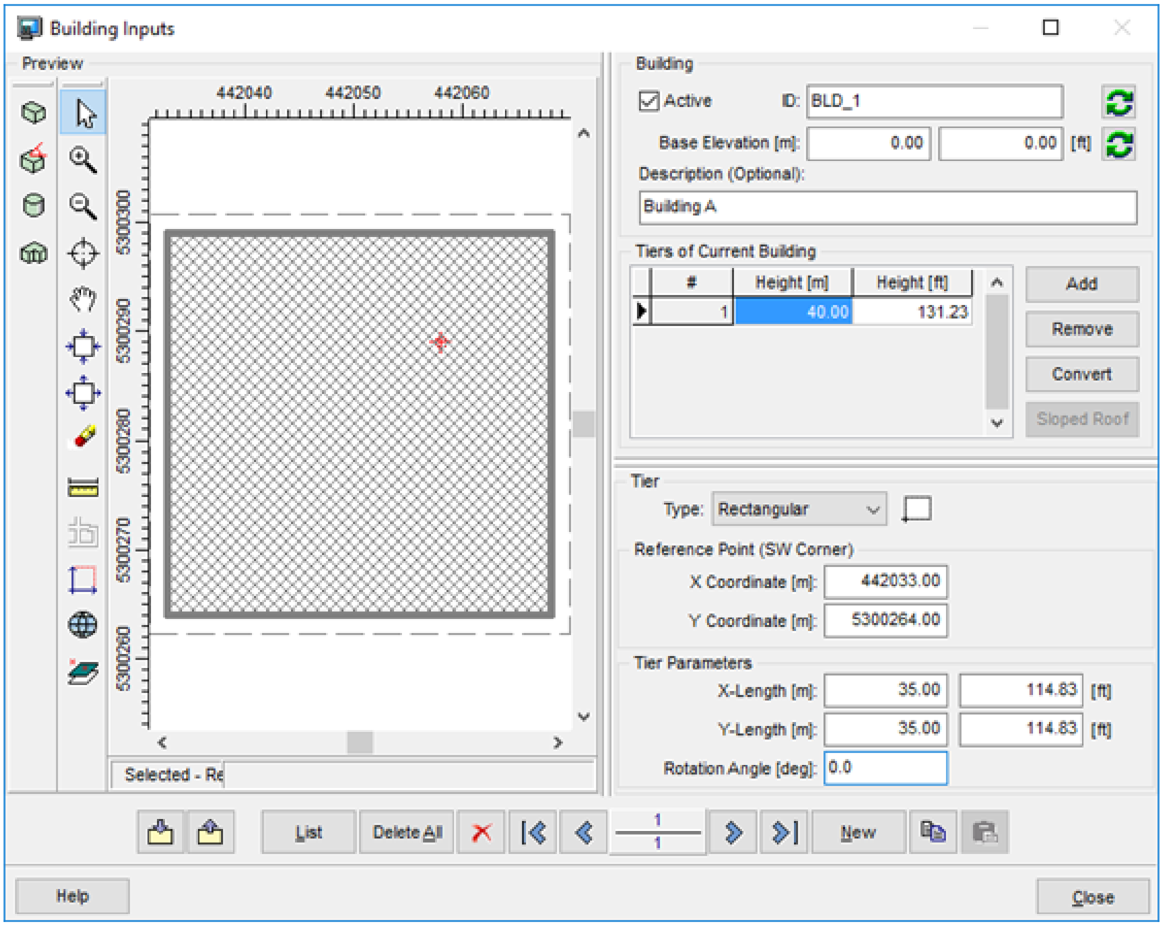
> When fully zoomed out, the area of interest is the red dot located at the center of the map - that is the .DXF map you imported and it delineates the sources and buildings that need to be defined.

> The raster image map will disappear at a certain zoom level. The DXF map will not.

Step 2: Select the **Rectangular Building** tool ( ) from the Application Toolbar to digitize the building using the red square of the DXF map as a guide. To digitize a building, click with the mouse pointer at one of the building corners and while holding down the left mouse button drag the cursor until you reach the opposite corner. Release the mouse button.

Step 3: The **Building Inputs** dialog is displayed. Here you can adjust the building parameters according to the table below. Click the Close button when done.

Parameter	Value
ID	BLD_1
Description	Building A
Base Elevation	0.00
Height	40 m
Type	Rectangular
X Coordinate (SW Corner)	442033
Y Coordinate (SW Corner)	5300264
X-Length	35 m
X-Length	35 m
Rotation Angle	0 deg



Building Inputs dialog

It is also possible to import the buildings from an AutoCAD DXF file instead of digitizing each building as you did in the above steps. In this case, use the menu option **Import | Buildings | AutoCAD DXF**. This approach is faster and will undoubtedly save you time on future projects. For this tutorial, you do not need to do this, since you already specified the building for this project. Please see the AERMOD View Help file for more information on Importing Buildings from a DXF file.

Defining the Stacks

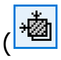
You now need to define the stacks, or point sources, in your project. There are two ways to define the location of the sources:

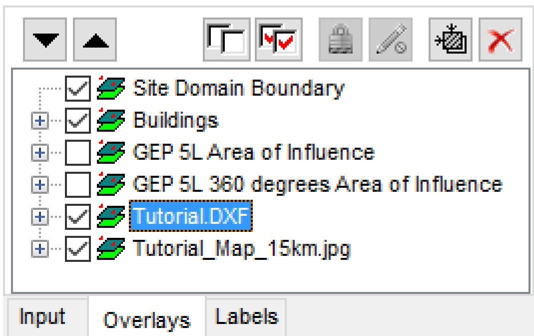
- Graphical Mode
- Text Mode

- Import Mode

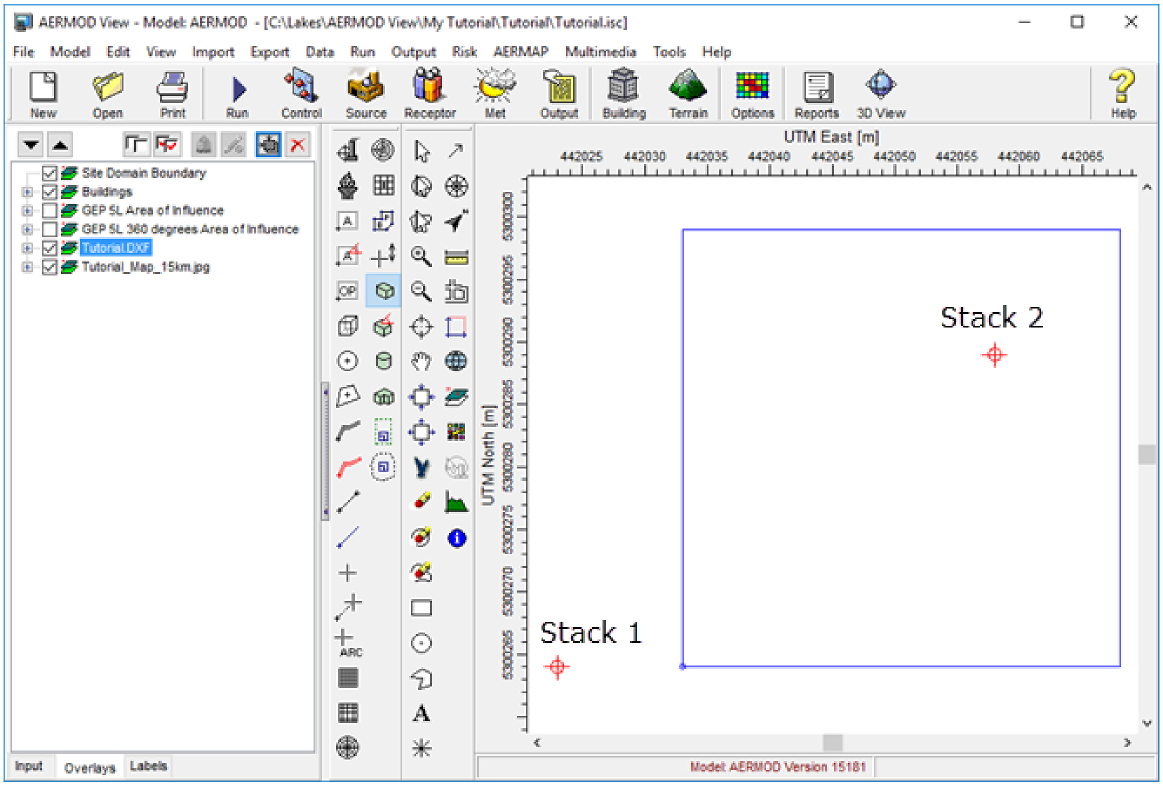
For this project, we will define the stacks in graphical mode using as a guide the DXF base map that we imported in the previous section. If you have a base map of your facility, then the graphical mode method might be more efficient and faster than the text mode.

Follow the steps below to specify the stacks in graphical mode:

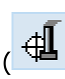
Step 1: Before you start specifying the stacks, go to the **Overlays** tab, select the **Tutorial.DXF** layer and click on the **Zoom to Overlay** button ().

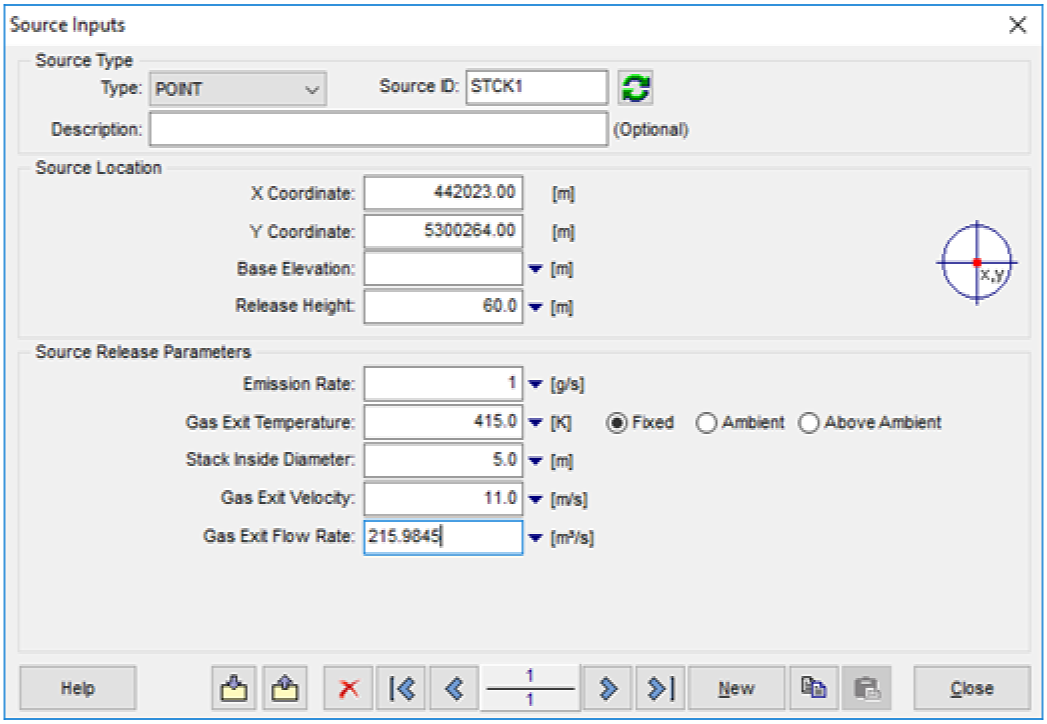


Step 2: The drawing area shows a zoom view of the extents of the DXF map showing the location of the building and the 2 stacks.



> The DXF file imported for this project was imported as a base map, and therefore it does not contain any reference of the stack or building location/dimensions. What you see on the drawing area is just an image showing the location and shape of these objects.

Step 3: Click the **Point Source** tool (), located on the Application Toolbar, and then click with the left-mouse button on the location of **Stack 1**. The **Source Inputs** dialog is displayed.



Source Inputs dialog

Step 4: Under the **Source Inputs** dialog, adjust the X and Y coordinates and enter the additional parameters for **Stack 1** according to table below. Click the **Close** button when done. Repeat the same steps provided for **Stack 1** to specify **Stack 2**. Use information provided in table below.

Parameter	STACK 1	STACK 2
Type	POINT	POINT
ID	STCK1	STCK2
X Coordinate	442023.00	442058.00
Y Coordinate	5300264.00	5300289.00
Base Elevation	0.00	0.00
Release Height	60 m	50 m
Emission Rate	1 g/s	1 g/s
Gas Exit Temperature	415 K (Fixed)	350 K (Fixed)

Parameter	STACK 1	STACK 2
Stack Inside Diameter	5 m	1 m
Gas Exit Velocity	11 m/s	10 m/s

Step 5: The location of the 2 stacks will now be displayed in the drawing area along with the building.

Visualization with 3D View

Now that you have specified your building and stacks, you can look at their three-dimensional visualization. This is a good time to check if anything is wrong with the dimensions and shape of the objects.

AERMOD View offers you two ways to visualize your project in 3D:

- **Visualization with 3D View** - Lakes Environmental 3D visualization application
- **Visualization with Google Earth** - this option requires that you have Google Earth installed in your machine

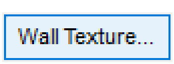
Follow the steps below on how to visualize your project using **3D View**:

Step 1: Select **View | 3D View...** from the menu or click on the 

button and select **3D View**... The **3D View** window is displayed. In this window you have complete control of rotation of the image in all directions, the ability to change color or add textures to your buildings, save views, and much more.

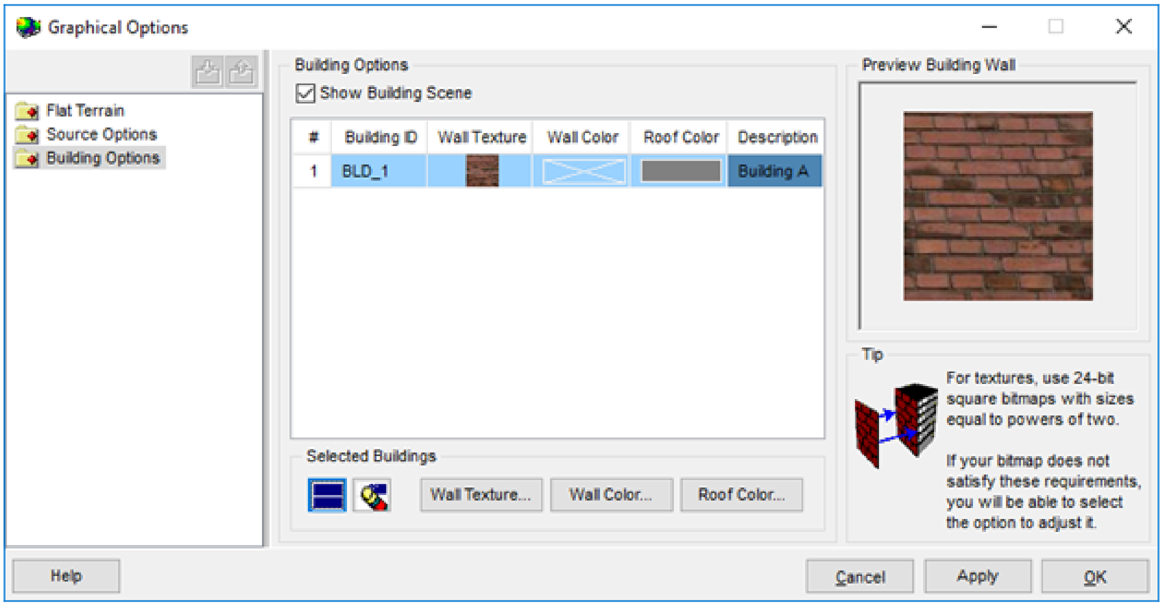
Step 2: Browse the options available in the **3D View** window. Use the mouse wheel or **Zoom In** tool () to get a closer look at your building and stacks.

Step 3: To apply texture or color to your building walls, select **Options | Buildings...** from the menu to open the **Graphical Options** dialog.

Step 4: Click the 

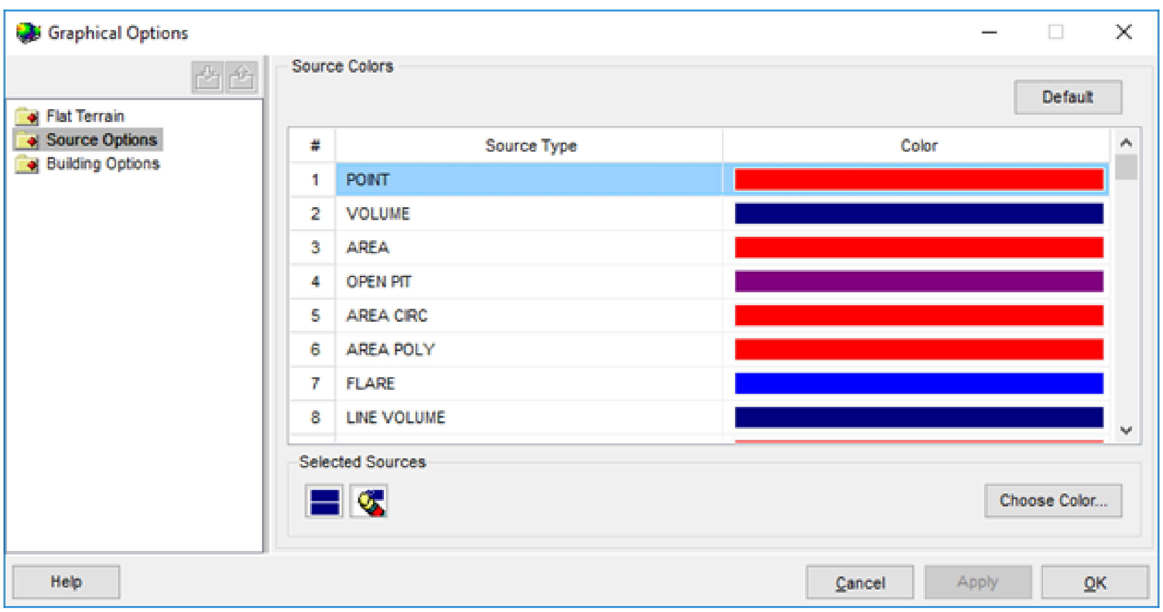
button to load the choices for wall texture.

Step 5: Double-click on the table in the appropriate column to change the **Wall Texture** for all the buildings. You can also change the **Roof Color** and **Wall Color**.



Step 6: Click the **OK** button to close the dialog. Note that the selected Wall Bitmap or Wall Color has been applied to your buildings.

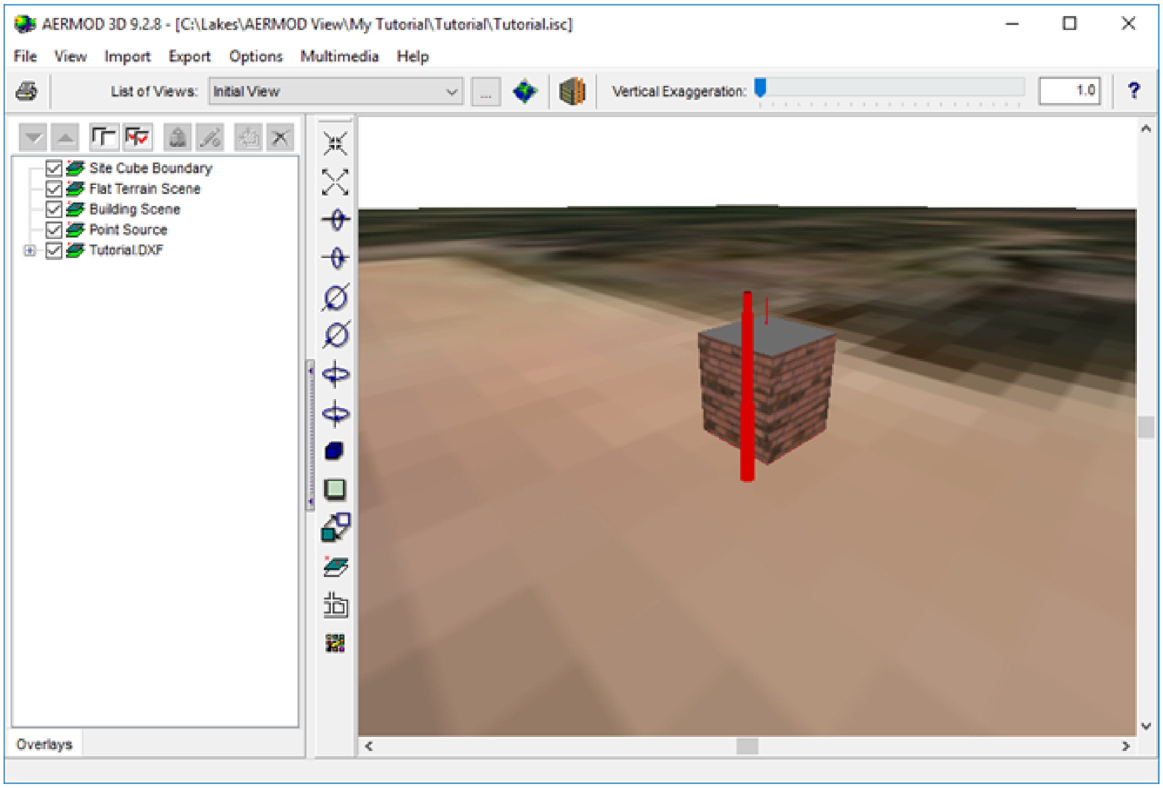
Step 4: You can also change the appearance of your sources. Select **Options | Sources...** from the menu to open the **Graphical Options** dialog.



Step 5: Double-click on the **Point** color bar to change the color of your stacks.

Step 6: Click the **OK** button to close the dialog. Note that the selected color has been applied to your stacks.

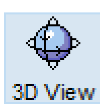
Step 5: When you have finished browsing the 3D View options, please close the **3D View** main window.

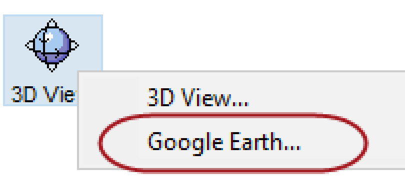


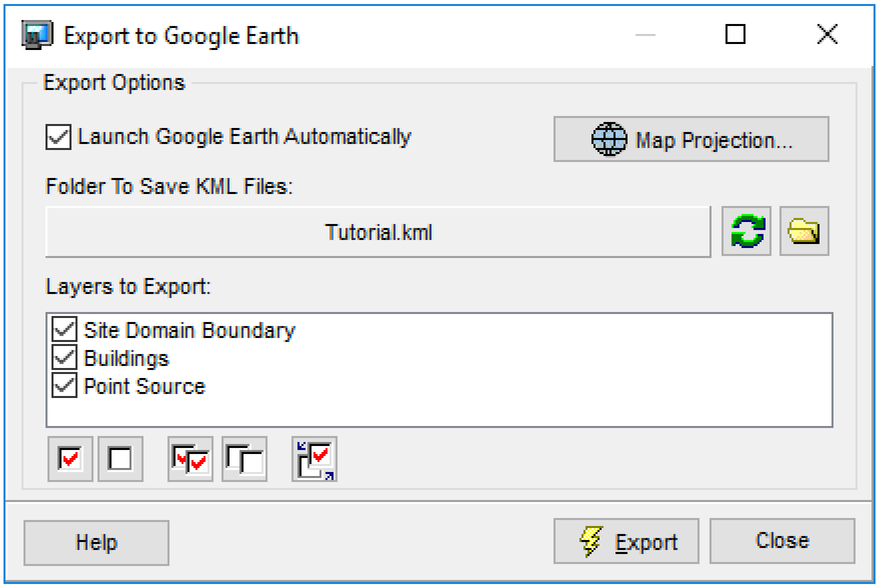
Visualization with 3D View

Visualization with Google Earth

Follow the steps below on how to visualize your project in Google Earth:

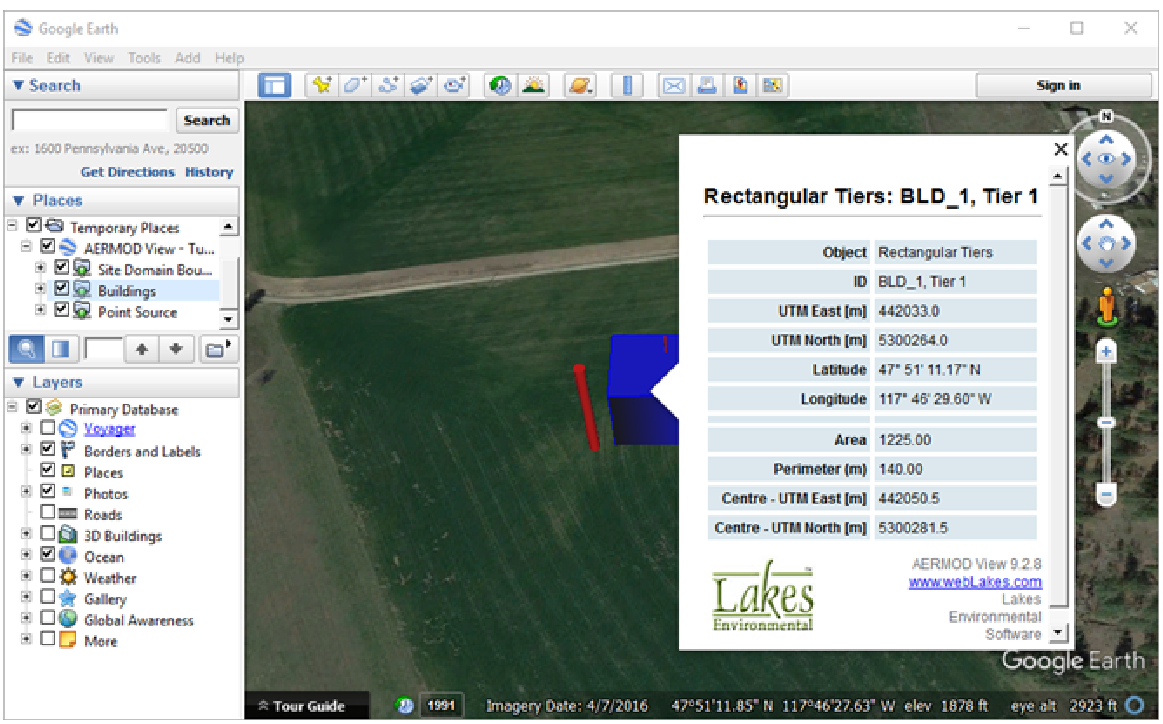
Step 1: Click the **3D View**  button and select **Google Earth...** The Export to Google Earth dialog is displayed.





Step 2: Click the  button.

Step 2: **Google Earth** will be launched automatically and you will be able to visualize your building and stacks. Within the **Google Earth** application, the information exported by AERMOD View is displayed under the **Temporary Places** folder. Clicking on the object link (e.g., STCK2) will display the information bubble containing object parameters. Close **Google Earth** when you are done.



Visualization with Google Earth

Control Pathway

Now that we have defined the sources of emissions and the building, we can finish setting up the project by specifying receptors and additional modeling options in the several pathways.

For this tutorial, we will input data into the five pathways in the following sequence:

- Control Pathway
- Source Pathway
- Receptor Pathway
- Meteorology Pathway
- Output Pathway

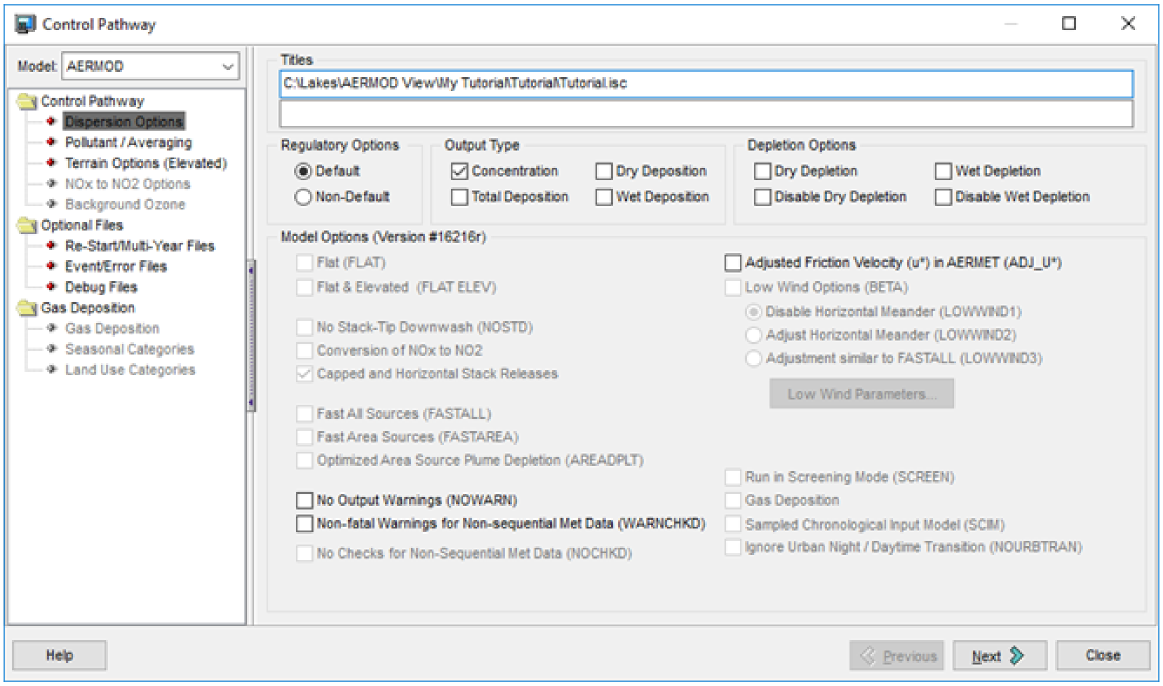
The **Control Pathway** allows you to specify the overall job control options such as dispersion options, pollutant, and averaging times. The **Control Pathway** can be accessed by clicking the **Control** menu toolbar button or by selecting **Data | Control Pathway...** from the menu. The **Control Pathway** dialog will open.

Before you start inputting data, you should note the indicators that show where you are in the pathways:

- The name of the pathway is shown at the top of the window.
- In the top left corner of the window, the current model is shown. This can be changed at any time by selecting another model from the drop-down list.
- The tree view on the left indicates the specific window you are currently in and also allows you to jump to any other window within the pathway.

CO – Dispersion Options

In the **Dispersion Options** window, you can specify project titles and dispersion options such as output type, plume depletion and non-default options.



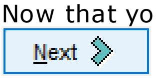
Control Pathway – Dispersion Options

Step 1: In the **Titles** fields enter the following project title: *XYZ Company - Concentration Calculation - 1986 Met Data*.

Step 2: In the **Dispersion Options** panel, select the **Regulatory Default Options**. This is the default option for regulatory applications.

Step 3: For this tutorial we are going to calculate only concentration results. Select the **Concentration** option from the **Output Type** panel. A summary of the parameters to be specified in this window is shown in table below:

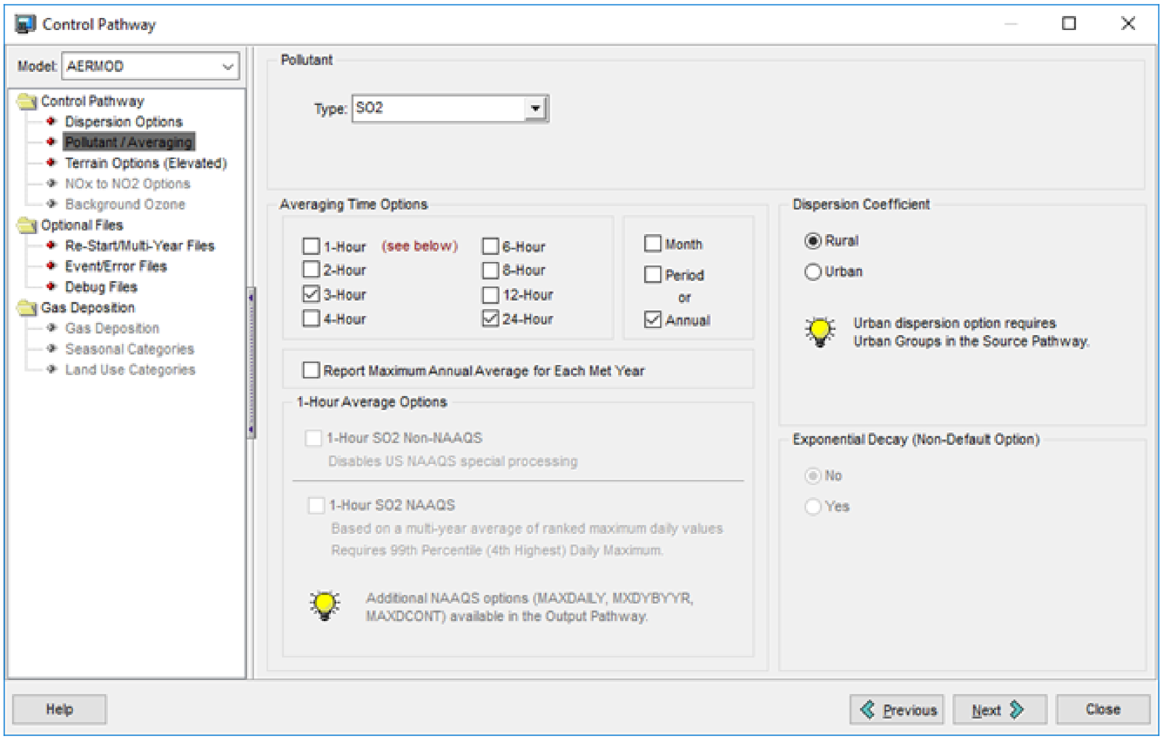
Parameter	Value
Dispersion Options	Regulatory Default Options
Output Type	Concentration

Step 4: Now that you have finished the data input in the **Dispersion Options** window, click the 

button, or select the **Pollutant/Averaging** option from the tree view.

CO – Pollutant/Averaging

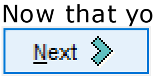
In the **Pollutant/Averaging** window, you can specify the pollutant being modeled, averaging time, and dispersion coefficient.



Control Pathway – Pollutant/Averaging

Step 5: A summary of the parameters to be specified in this window is shown in table below:

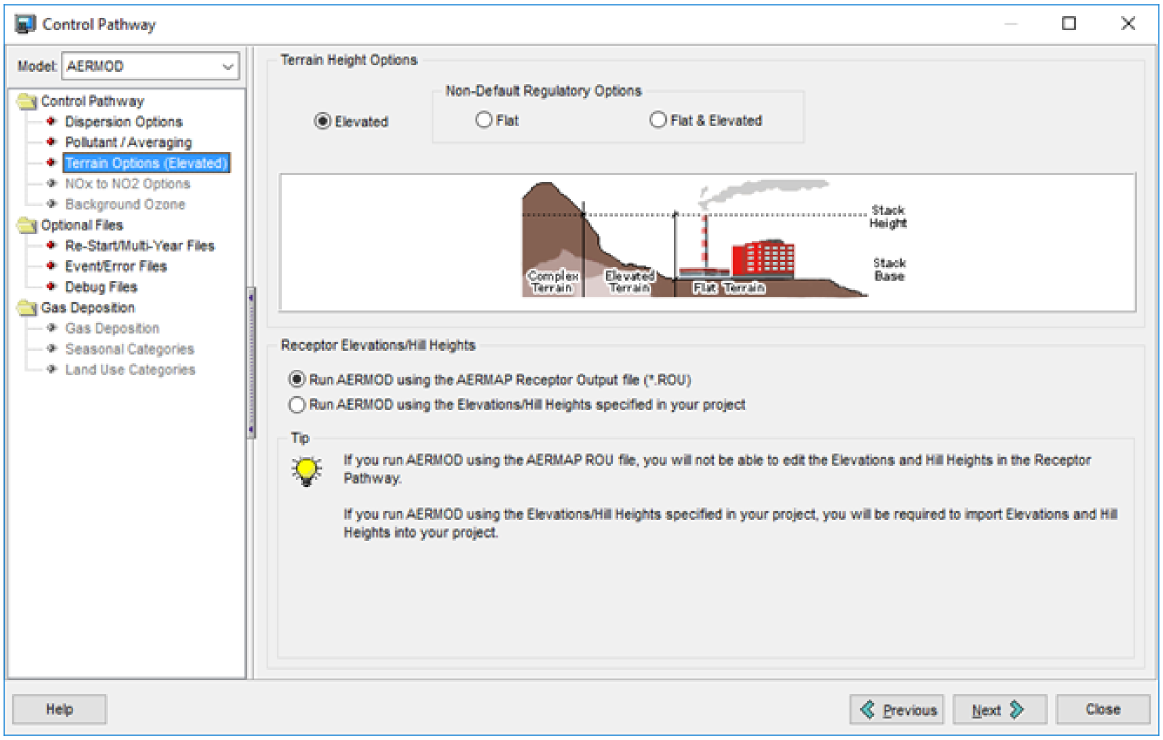
Parameter	Value
Pollutant Type	SO2
Averaging Time	3-Hour, 24-Hour, and Annual
Dispersion Coefficient	Rural **Note:** If the land use types including industrial, commercial and residential, account for 50% or more of an area within 3 km radius from the source, the site is classified as urban; otherwise, it is classified as rural.

Step 6: Now that you have finished the data input in the **Pollutant/Averaging** window, click the 

button, or select the **Terrain Options** from the tree view.

CO – Terrain Options

In the **Terrain Options** window, you can specify whether you will be modeling the effects of flat or elevated terrain.



Control Pathway – Terrain Options

Step 9: Make sure the **Elevated** option is selected for the **Terrain Height Options**.

Parameter	Value
Terrain Height Option	Elevated
Receptor Elevations/ Hill Heights	Run AERMOD using AERMAP Receptor Output file (*.ROU)

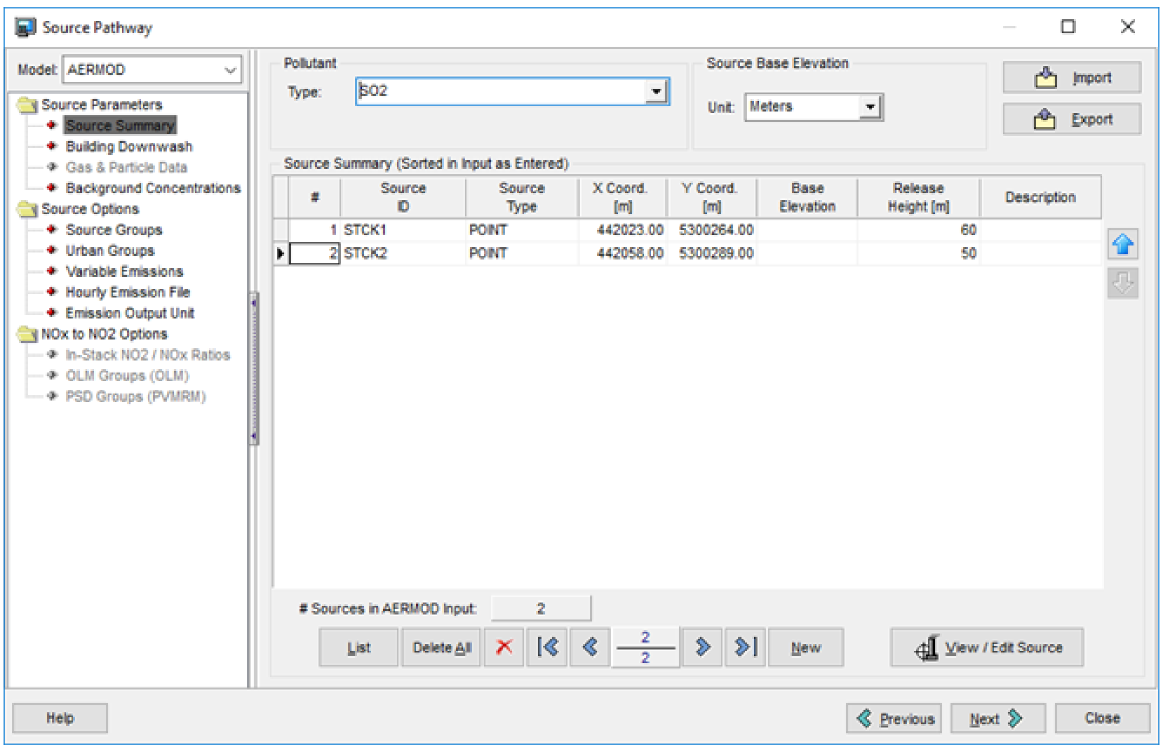
Step 10: You have now finished specifying all the data required in the **Control Pathway** for this tutorial and do not need to specify information in any of the other windows. Click on the **Close** button to exit the **Control Pathway** dialog and save your selections.

Source Pathway

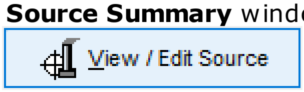
The **Source Pathway** allows you to specify the source input parameters and source group information such as source types, building downwash, and variable emissions. The **Source Pathway** can be accessed by clicking the **Source** menu toolbar button or by selecting **Data | Source Pathway...** from the menu. The **Source Pathway** dialog will open.

SO – Source Summary

In the **Source Summary** window, you can specify information on the number of sources specified for your current project, pollutant information, and source base elevation information.



Source Pathway – Source Summary

Step 1: The **Source Summary** window will display the two sources you previously defined. Click on the 

button to open the **Source Inputs** dialog where you can review the information specified for the two stacks. Click on the **Close** button when you have finished.

Step 2: You have now finished specifying all the data required in the **Source Pathway** for this tutorial and do not need to specify information in any of the other windows. Click on the **Close** button to exit the **Source Pathway** dialog and save your selections.

Receptor Pathway

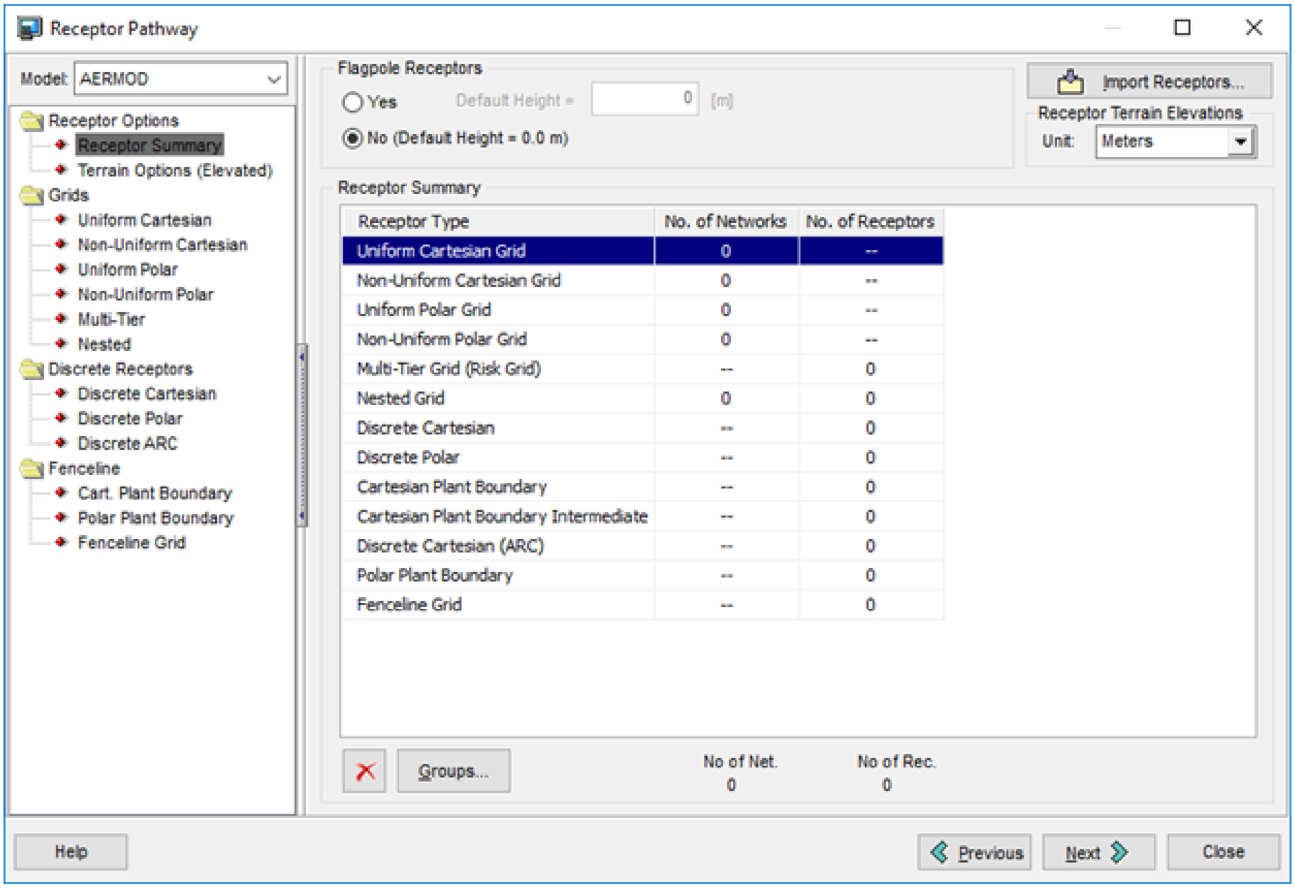
The **Receptor Pathway** allows you to specify the receptor locations. Receptors are locations where you want the model to calculate concentration results. The **Receptor Pathway** can be accessed by clicking the **Receptor** menu toolbar button or by selecting **Data | Receptor Pathway...** from the menu. The **Receptor Pathway** dialog will open.

For this tutorial, we are going to define an **Uniform Cartesian Receptor Grid** centered on one of the stacks.

RE – Receptor Summary

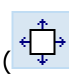
In the **Receptor Summary** window, you can view a summary of the type of receptors, number of grids, and total number of receptors specified for the current project. You can also define flagpole receptors, import receptors, and specify receptor terrain elevation units.f

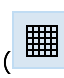
The **Receptor Summary** table displays all the receptors and grids defined in the project. There are two ways you can define a uniform Cartesian receptor grid, graphically or in text mode. For this tutorial we will define the grid in graphical mode.



Receptor Pathway – Receptor Summary

Step 1: Click the **Close** button to exit the **Receptor Pathway** dialog and return to the drawing area, where we can graphically define the receptor grid.

Step 2: Click the **View Max. Extents** () button on the **Annotation Toolbar** to zoom to the full extents of the modeling region.

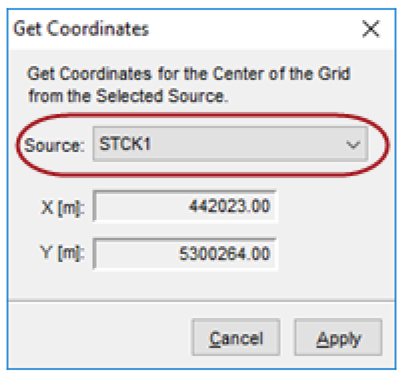
Step 3: Select the **Uniform Cartesian Grid** tool () located on the **Application Toolbar**.

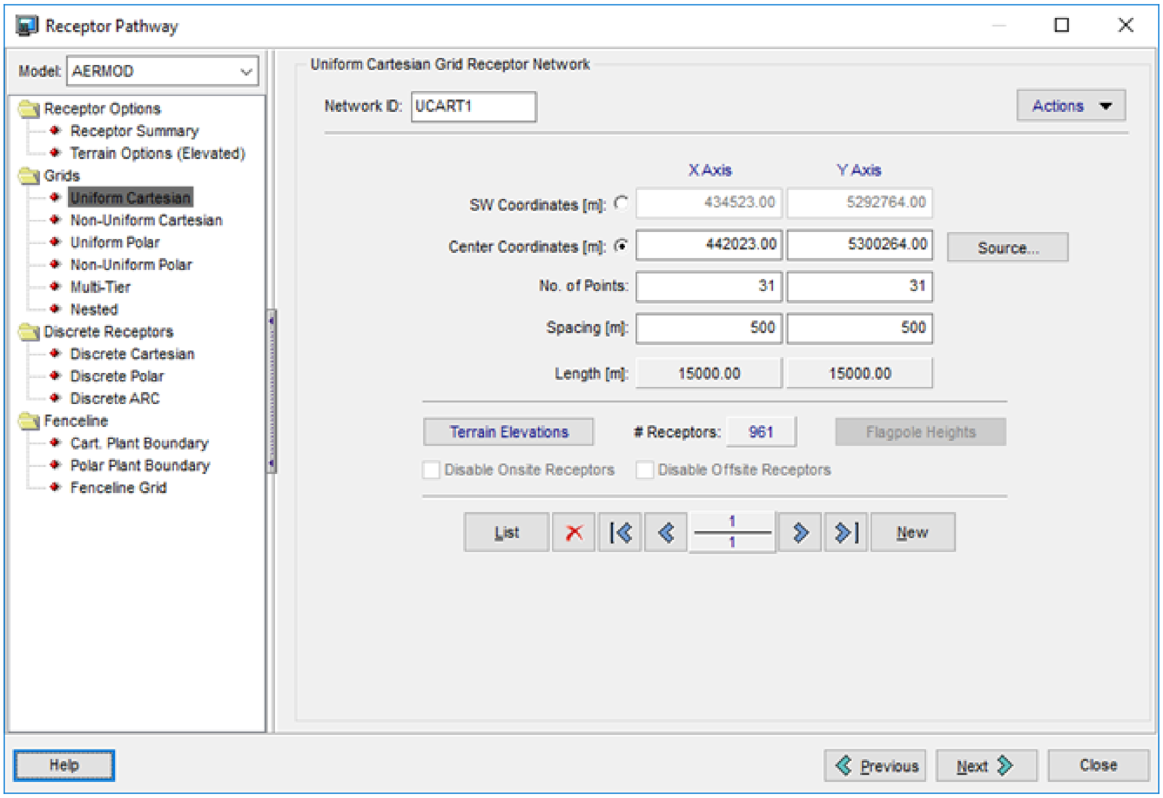
Step 4: With the mouse pointer click on the drawing area at the location you want one of the corners of your grid to be located. Holding the left mouse button drag the mouse pointer diagonally until you reach the desired grid size. Release the left mouse button. The **Uniform Cartesian Grid** window is displayed to allow you to adjust the coordinates of the grid, the number of points, and the spacing.

RE – Uniform Cartesian Grid

In the **Uniform Cartesian Grid** window, you can define Cartesian grid receptor networks with uniform grid spacing.

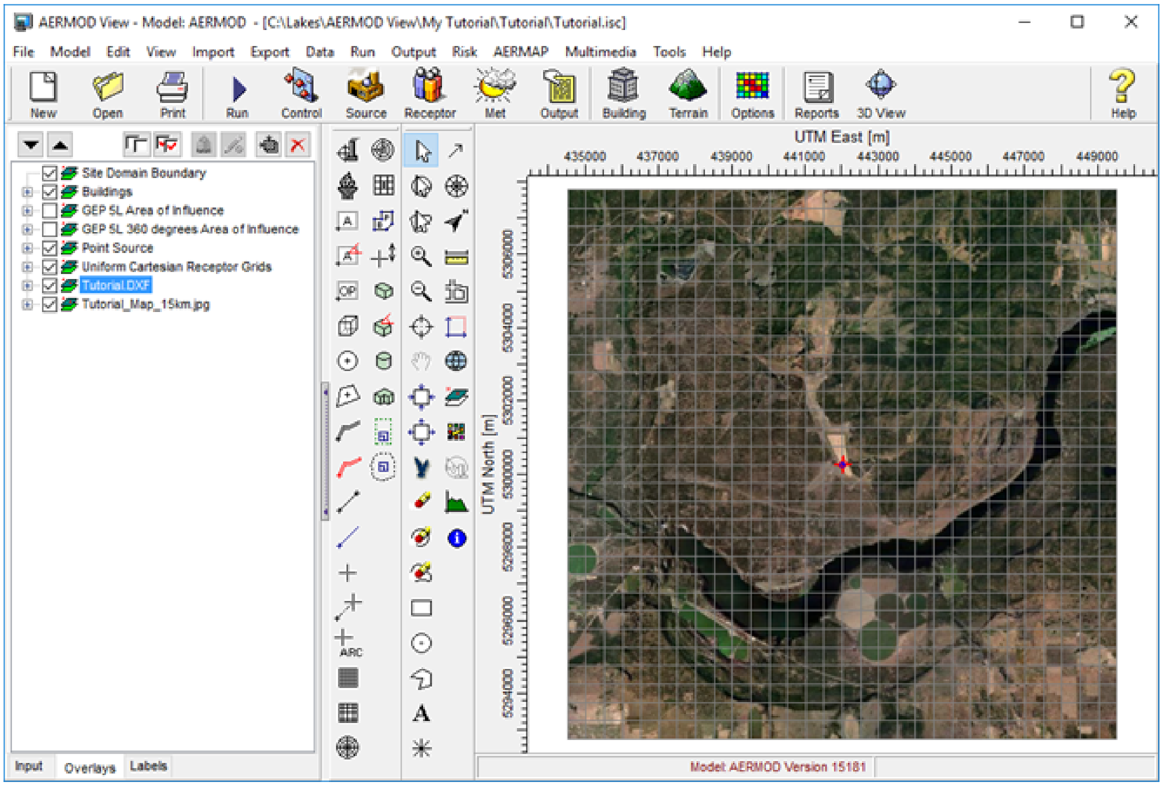
Step 5: Using the information below, you can edit the grid in the **Uniform Cartesian Grid** window.

Parameter	Value
Network ID	UCART1
Center Coordinates	X: 442023.00, Y: 5300264.00 Select the option to specify the **Center Coordinates** of grid and click the **Source** button. Under the **Get Coordinates** dialog, select **STCK1** and click Apply. 
No. of Points	31 x 31
Spacing	500 meters, 500 meters
Length	15000 by 15000 meters



Receptor Pathway – Uniform Cartesian

Step 5: You have now finished specifying all the data required in the **Receptor Pathway** for this tutorial and do not need to specify information in any of the other windows. Click on the **Close** button to exit the **Receptor Pathway** dialog and save your selections. The defined uniform Cartesian grid should now be displayed in the drawing area.



Meteorology Pathway

The **Meteorology Pathway** allows you to specify the input meteorological data file and other meteorological variables, including the period to process for the meteorological files. The **Meteorology Pathway** can be accessed by clicking the **Met** button or by selecting **Data | Meteorology Pathway...** from the menu. The **Meteorology Pathway** dialog will open.

The AERMOD model requests the **Surface** and the **Profile** met data files. These are the files you generated in the AERMET View Tutorial.

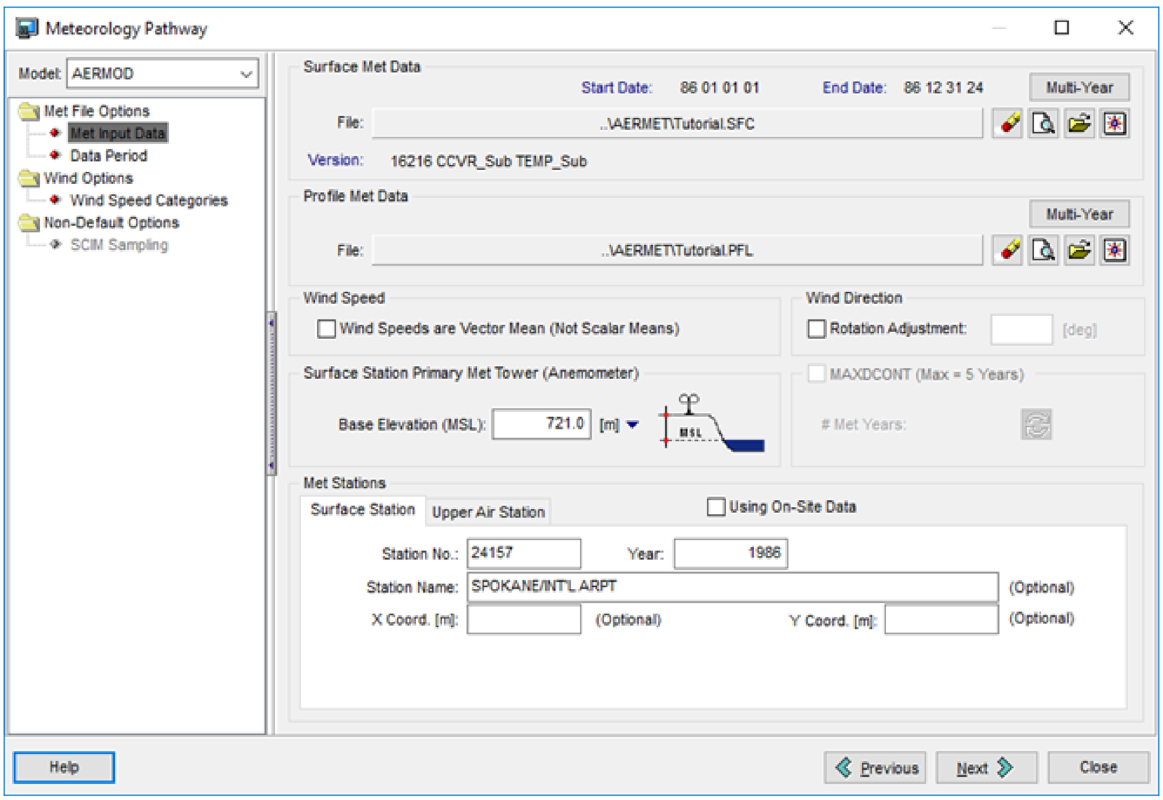
ME – Met Input Data Window

The **Met Input Data** page allows you to specify the meteorological data file and information on the meteorological stations.

Step 1: Click on the  button in the **Surface Met Data** panel and select the surface met data file that you processed using AERMET View (*.sfc). AERMOD View automatically loads the **Profile Met Data** file (*.PFL) if this file is located in the same folder as the **Surface Met Data** file (*.SFC).

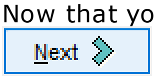
Step 2: A summary of the parameters to be specified in this window is shown in table below:

Parameter	Value
Folder Location	C:\Lakes\AERMOD View\My Tutorial\AERMET
Surface Met File	Tutorial.SFC
Profile Met File	Tutorial.PFL
Station Base Elevation (MSL)	721 m



Meteorology Pathway – Met Input Data

AERMOD View reads the Station No. and Year from the specified met data files and places these values in the appropriate fields under the **Met Stations** panel.

Step 3: Now that you have finished the data input in the **Met Input Data** window, click the 

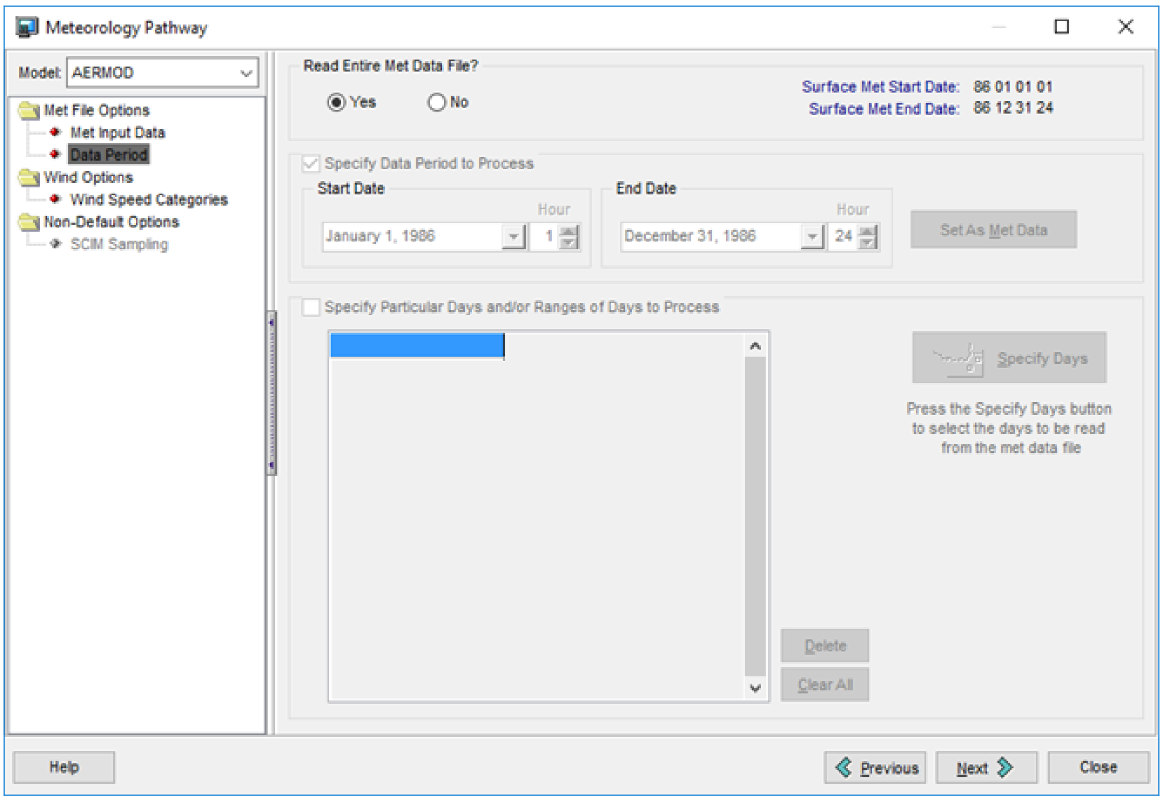
button, or select the **Data Period** option from the tree view.

ME – Data Period

The **Data Period** window allows you to specify particular days or ranges of days to process from the sequential meteorological file input if you do not want to run the entire met data file.

Step 4: By default, the model will read the entire met data file. For this tutorial we will read the entire data file, therefore keep the Yes option selected.

Parameter	Value
Read Entire Met Data File?	Yes

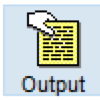


Meteorology Pathway – Data Period

Step 5: You have now finished specifying all the data required in the **Meteorology Pathway** for this tutorial and do not need to specify information in any of the other windows. Click on the **Close** button to exit the **Meteorology Pathway**.

Output Pathway

The **Output Pathway** allows you to specify the output options for a particular run such as contour plot files and threshold violation files. The **Output Pathway** can be accessed by clicking the **Output** menu toolbar button or by selecting **Data | Output Pathway...** from the menu. The **Output Pathway** dialog will open.



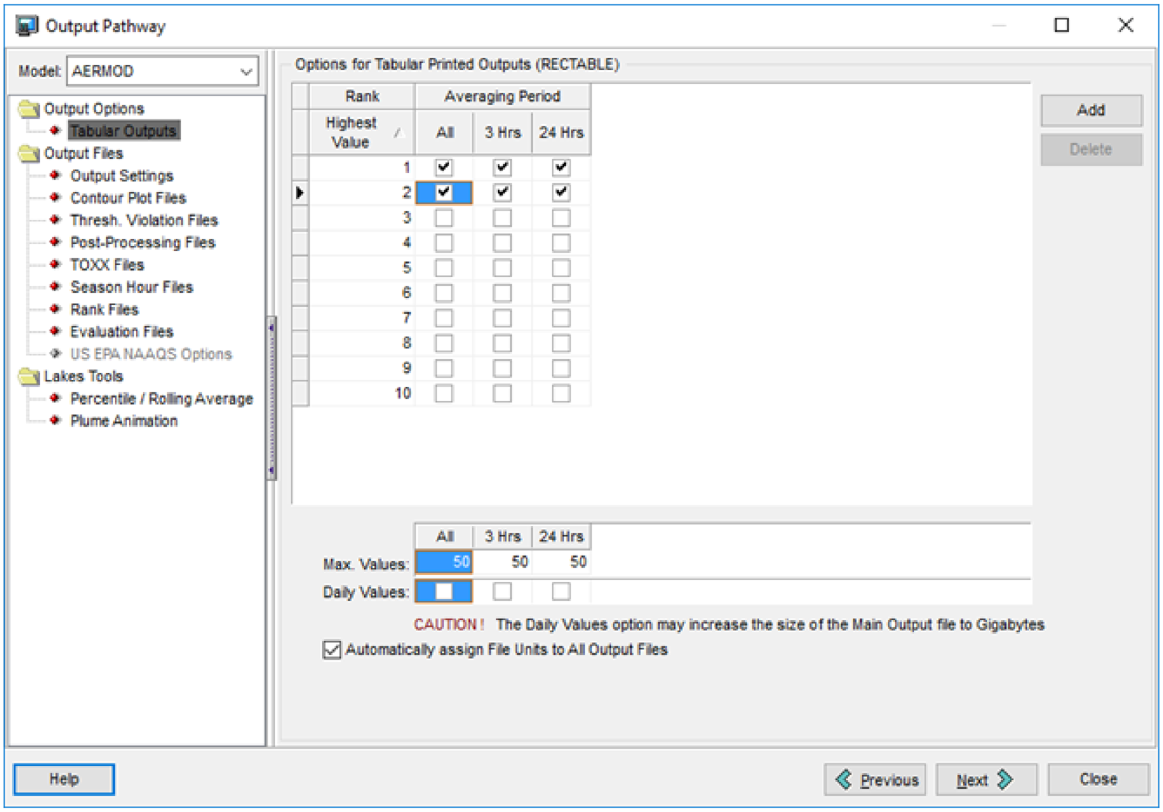
For this tutorial we will be specifying **High Values**, **Maximum Values**, and **Contour Plot Files**.

OU – Tabular Outputs

In the **Tabular Outputs** window you can define tabular output options for each short-term averaging period selected in the **Pollutant/Averaging** window of the **Control Pathway**. These options will affect the type of results to be included in the output file after the model run.

Step 1: The **Highest Values** option controls outputs for the high value summary tables by receptor. Make sure the **1st** highest values by receptor, for all short-term averages (**3** hrs and **24** hrs), is selected. Note that if you check the **1st** and **2nd** boxes for the **ALL** option, then all check boxes for that specific high value are automatically checked for all short-term averages.

Step 2: The **Maximum Values** option controls output for overall maximum value summary tables. Specify **50** maximum values for all short-term averages.



Output Pathway – Tabular Outputs

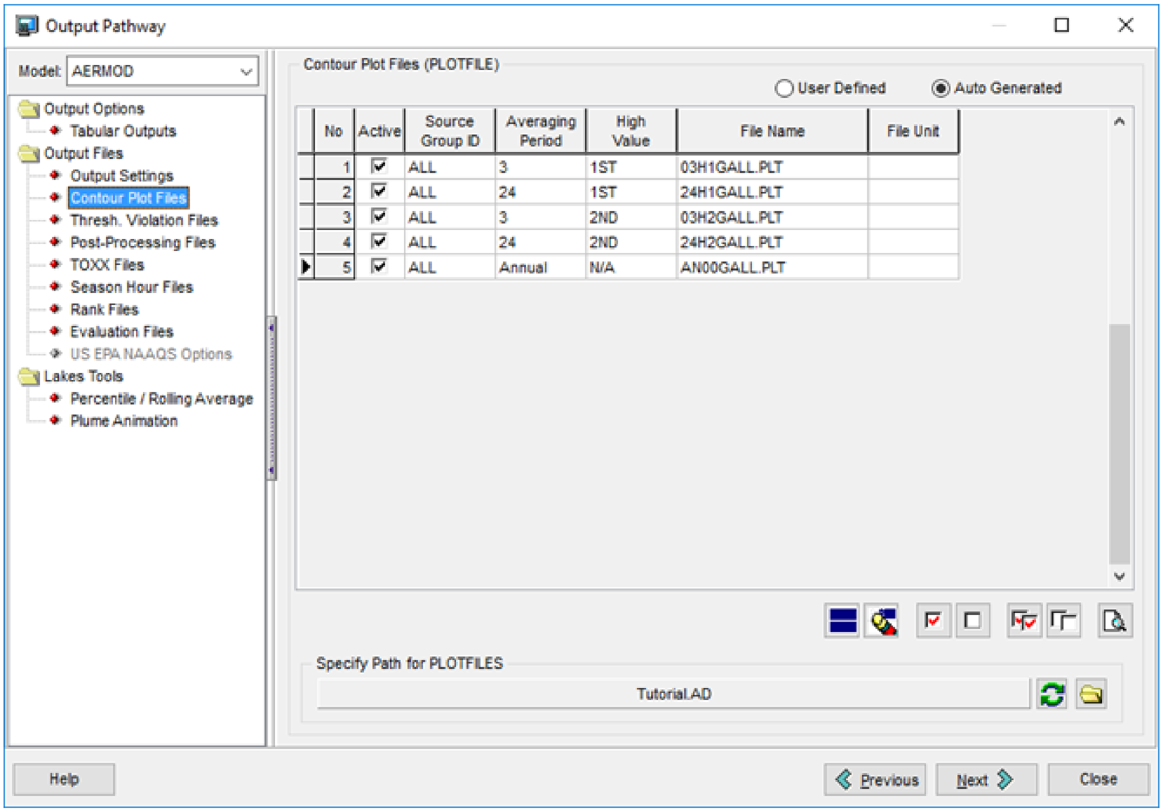
Step 3: Now that you have finished specifying your **Tabular Output** options, click the 

button, or select the **Contour Plot Files** option from the tree view.

OU – Contour Plot Files

The **Contour Plot Files** screen allows you to produce files that are suitable for generating contour plots of your modeling results.

Step 4: AERMOD View automatically generates all possible combinations of plot files that can be setup for the current run with the **Auto Generated** option. You can discard one or more of these plot files by un-checking the **Active** field. For this tutorial project we will use all auto-generated plot files.



Output Pathway – Contour Plot Files

In the **Specify Path for PLOTFILES** panel, you can specify the location where all the plot files are to be placed. By default, AERMOD View specifies that these files should be placed on the project directory in the following folders:
ProjectName.AD for the Plot Files generated by the AERMOD model
ProjectName.PR for the Plot Files generated by the ISC-PRIME model
ProjectName.IS for the Plot Files generated by the ISCST3 model

Step 5: You have now finished specifying all the data required in the **Output Pathway** for this tutorial and do not need to specify information in any of the other windows. Click on the **Close** button to exit the **Output Pathway**.

Terrain Processor

In order to calculate terrain elevations for all receptor locations and base elevations for all sources and building, you must run the US EPA AERMAP model (terrain Preprocessor for AERMOD). You will need to specify the digital terrain file that covers the modeling domain and process it using the AERMAP model. In AERMOD View this is done within the **Terrain Processor**.

Click on the **Terrain** menu toolbar button or select **Data | Terrain Processor** from the menu to open the **Terrain Processor**.

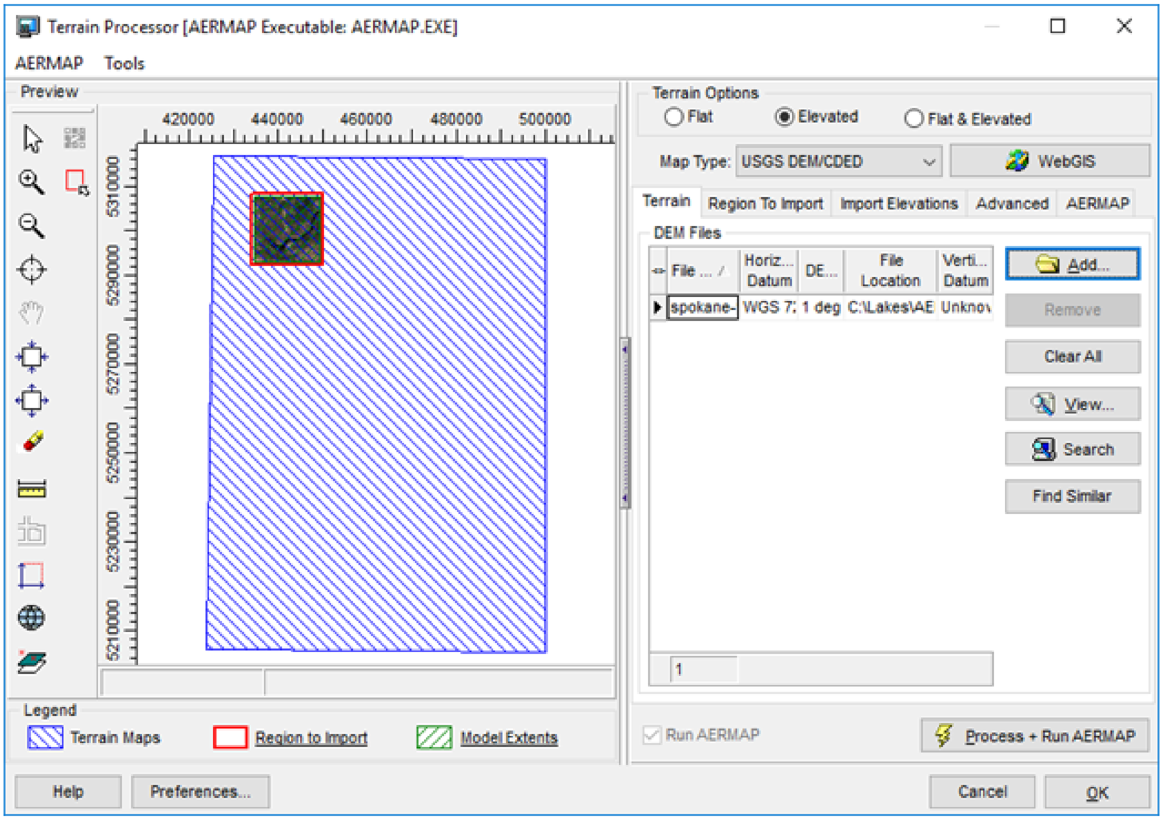
On the left side of the **Terrain Processor** window is the **Preview** panel, which displays a preview of the project modeling domain or **Model Extents** (green hatched region). On the right side of the **Terrain Processor** window, there are several tabs where you can specify options related to the preprocessing of terrain data.

Step 1: Under the **Terrain** tab, click on the 

button to specify the digital terrain data file according to information provide in table below:

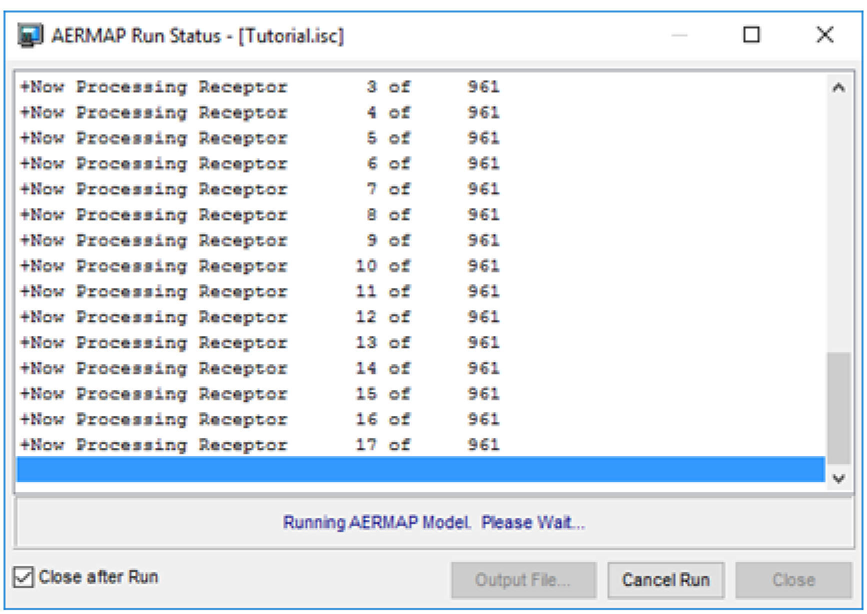
Parameter	Value
Map Type	USGS DEM
Folder Location	C:\Lakes\AERMOD View\Tutorial\Terrain
File Name	spokane-w.DEM

Step 2: The extents of the imported terrain file will be displayed in the **Preview** panel as a blue hatched region.

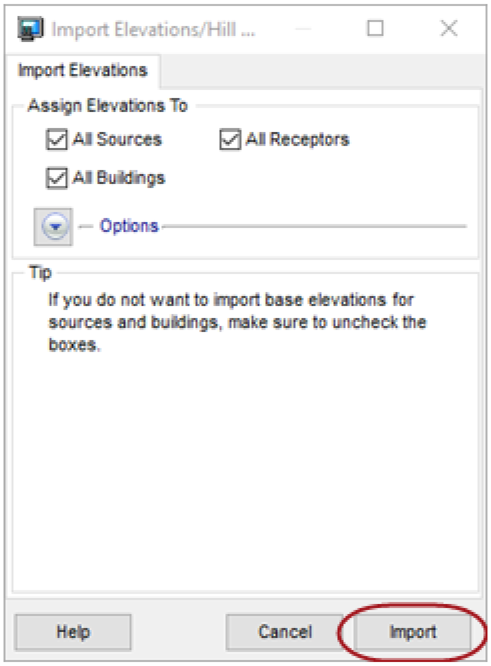


Terrain Processor – Terrain tab

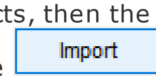
Step 3: You can now process your terrain. Click the Process + Run AERMAP button. The AERMAP run will take several minutes.



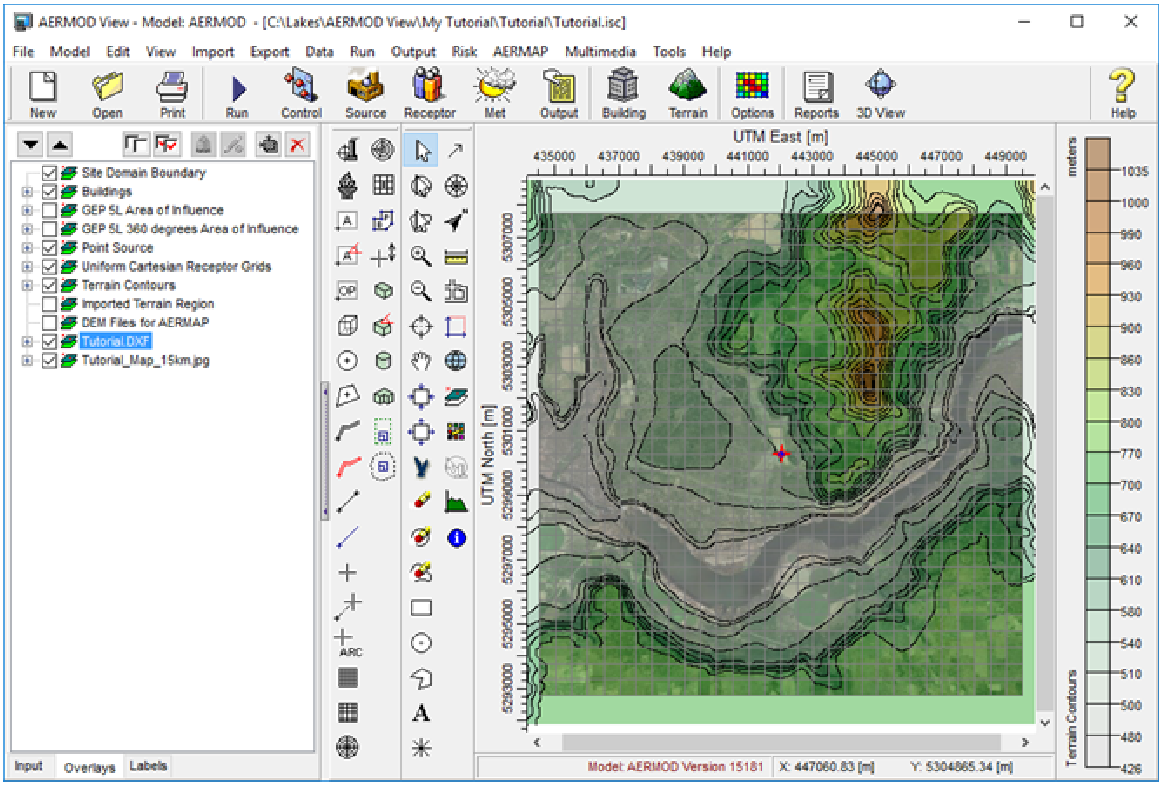
Step 4: Once processing is complete the **Import Elevations/Hill Heights** dialog will be displayed. This dialog gives you the option to import elevations for all sources, receptors, and buildings.



Import Elevations/Hill Heights dialog

Step 5: Since we want to import elevations for all objects, then the **All Sources**, **All Buildings**, and **All Receptors** boxes must be checked. Click the 

button. Now that you have imported terrain elevations, you can open the **Receptor Pathway**, **Source Pathway**, and **Building Inputs** dialog to view the imported elevations.



Terrain Contours

For future projects, you can automatically download digital terrain data by pressing the WebGIS button and then selecting the terrain format appropriate for your modeling area. The automatic downloads of NED terrain data (NED 1/3 and NED 1) as well as STRM1 terrain data are only available to users in current maintenance.

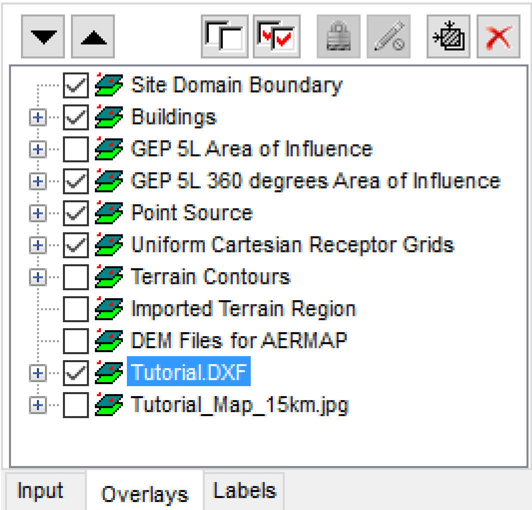
Running BPIP

Now that you have specified the required information for all five pathways and have processed the terrain data, you can begin running your project. We will start by first running the U.S. EPA Building Input Profile Program (BPIP) to obtain building downwash calculation results.

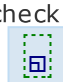
Follow the steps below to run the BPIP model:

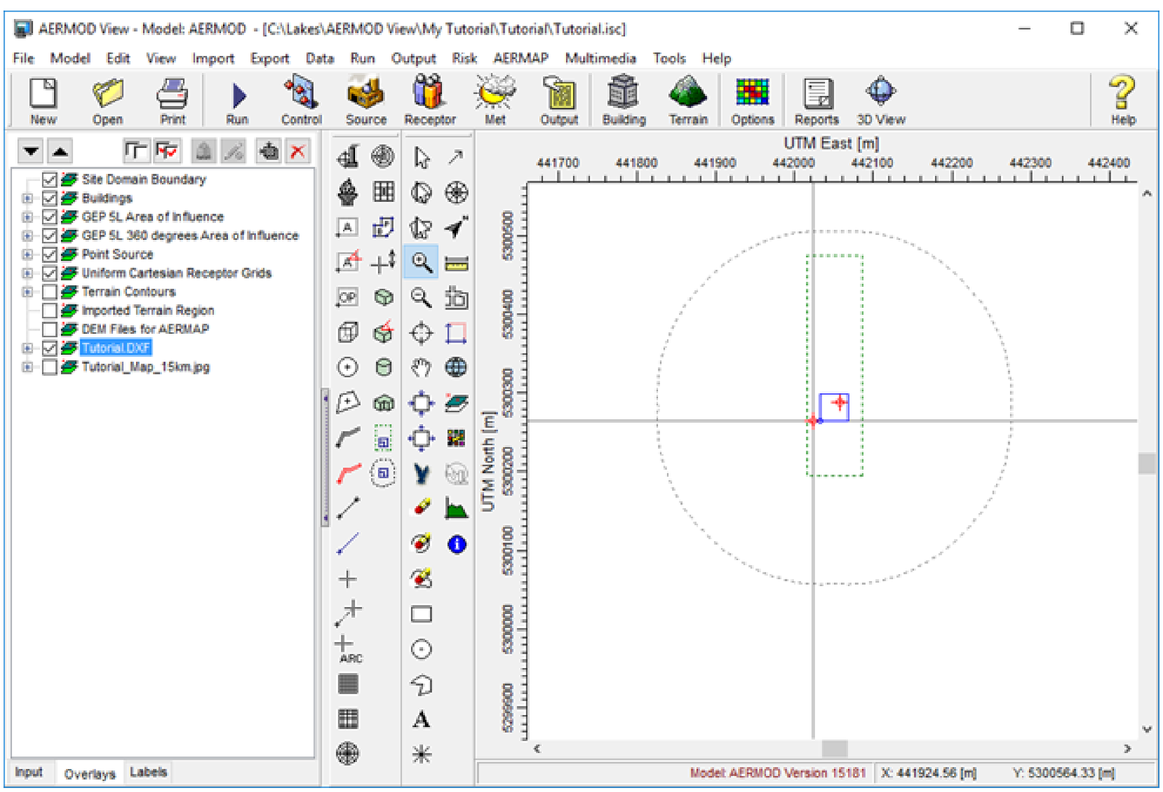
Step 1: Before we run BPIP, we need to do one last thing, which is to visualize the GEP 5L Area of Influence for the building. This is just for visualization purposes. Click the tool to display the **GEP 5L Area of Influence**.

Step 2: Turn off the **Terrain Contours** and **Tutorial_Map_15km.jpg** overlays by unchecking them in the **Overlays** tab.

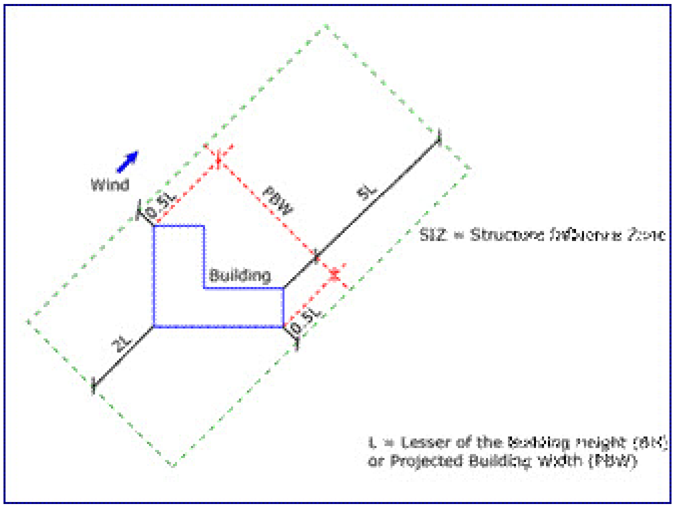


Step 3: Zoom in, so you can clearly see the building and sources.

Step 4: As image below shows, both your stacks are within the area of influence of the building. You can also check the **Structure Influence Zone** for the building specified in this tutorial by clicking the  tool.



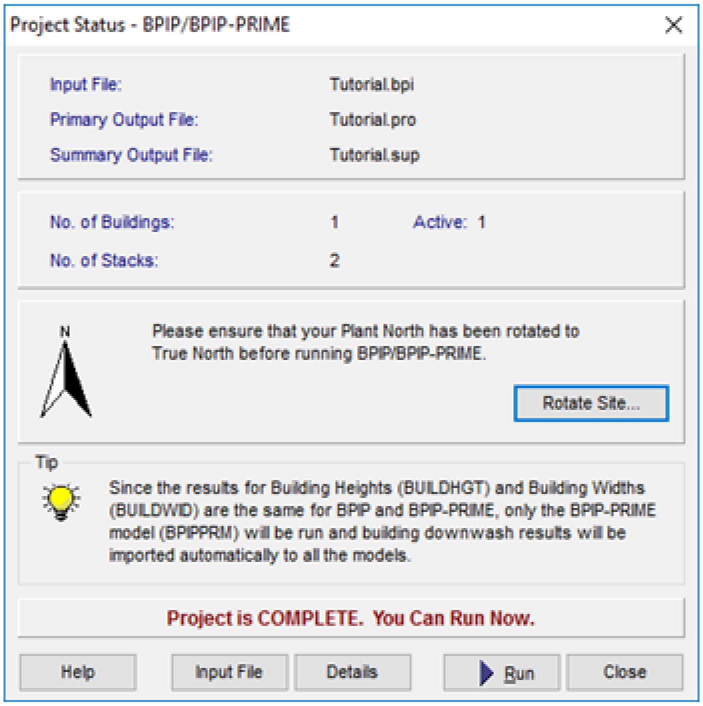
Structure Influence Zone (SIZ): For downwash analyses with direction-specific building dimensions, wake effects are assumed to occur if the stack is within a rectangle composed of two lines perpendicular to the wind direction, one at 5L downwind of the building and the other at 2L upwind of the building, and by two lines parallel to the wind direction, each at 0.5L away from each side of the building, as shown below. L is the lesser of the building height or projected building width. This rectangular area has been termed Structure Influence Zone (SIZ).



Structure Influence Zone (SIZ) Boundary Line

GEP 5L Area of Influence: Each structure produces an area of wake effect influence that extends out to a distance of five times L directly downwind from the trailing edge of the structure, where L is the lesser of the BH or PBW. As the wind rotates full circle, each direction-specific area of influence changes and is integrated into one overall area of influence termed the GEP 5L Area of Influence. Any stacks that are on or within the limit line are affected by GEP wake effects for some wind direction or range of wind directions.

Step 2: Select **Run | Run BPIP...** from the menu. The **Project Status** dialog is displayed showing the number of buildings and stacks that are going to be included in the run. On the bottom panel of the dialog, a message identifies if your project is complete or not. If your project is complete click on the  button.



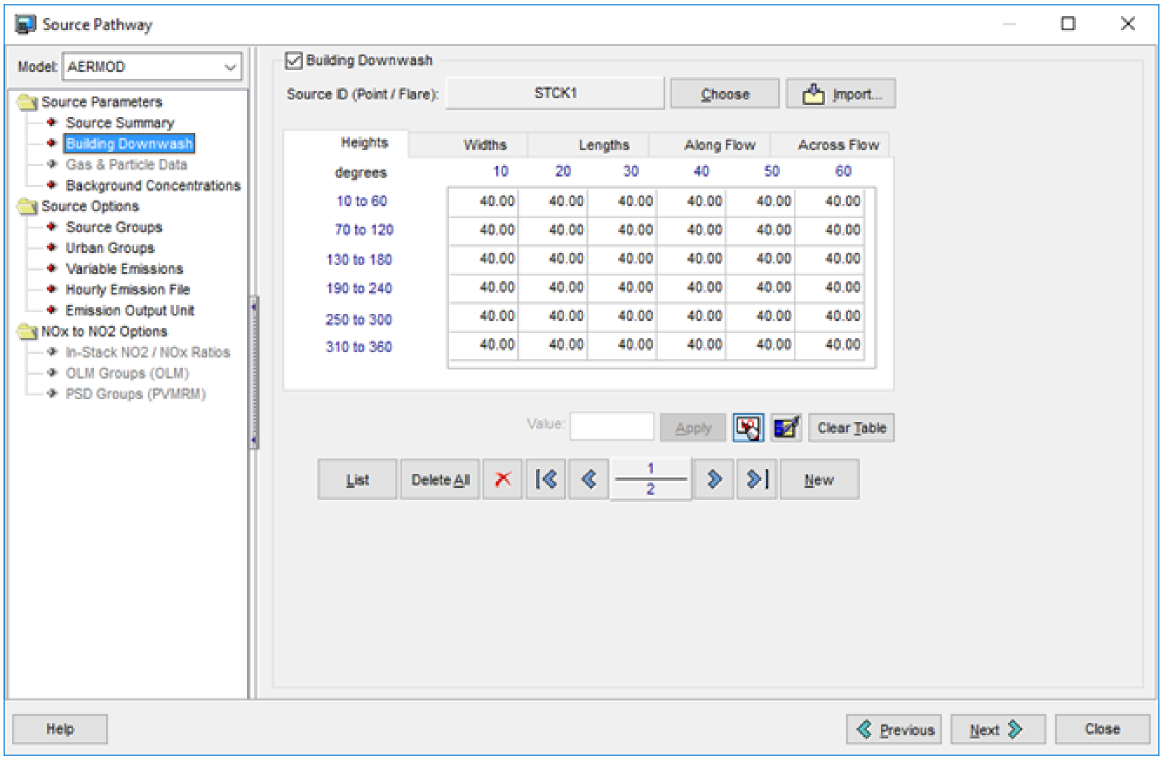
Project Status dialog

Step 3: When the run finishes, select **Yes** when asked if you want to see output file – this will open the BPIP output file (Tutorial.pro) for inspection.

Step 4: You can also view BPIP summary file that is also generated by BPIP by selecting **Output | BPIP Summary File** from the menu. This file contains information such as which building tiers are affecting a particular stack for a particular wind flow direction.

Step 5: We can check to make sure the building downwash information has been added to the **Source Pathway**. Click on the **Source** menu toolbar button to open the **Source Pathway** dialog and select the **Building Downwash** option from the tree view.

Step 6: Make sure the **Building Downwash** option box is checked. This option indicates to the model that the building influence on the plume should be taken into account. When you ran the BPIP model, AERMOD View automatically places all the information contained in the BPIP Output file into the tables. You should have two building downwash records created, one record for STCK1 and one record for STCK2.

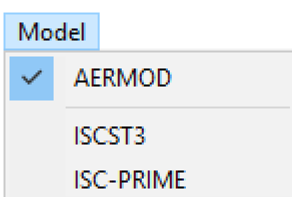


Source Pathway – Building Downwash screen

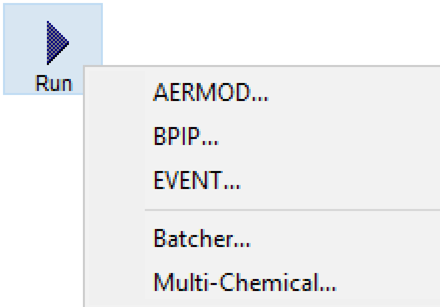
Running AERMOD

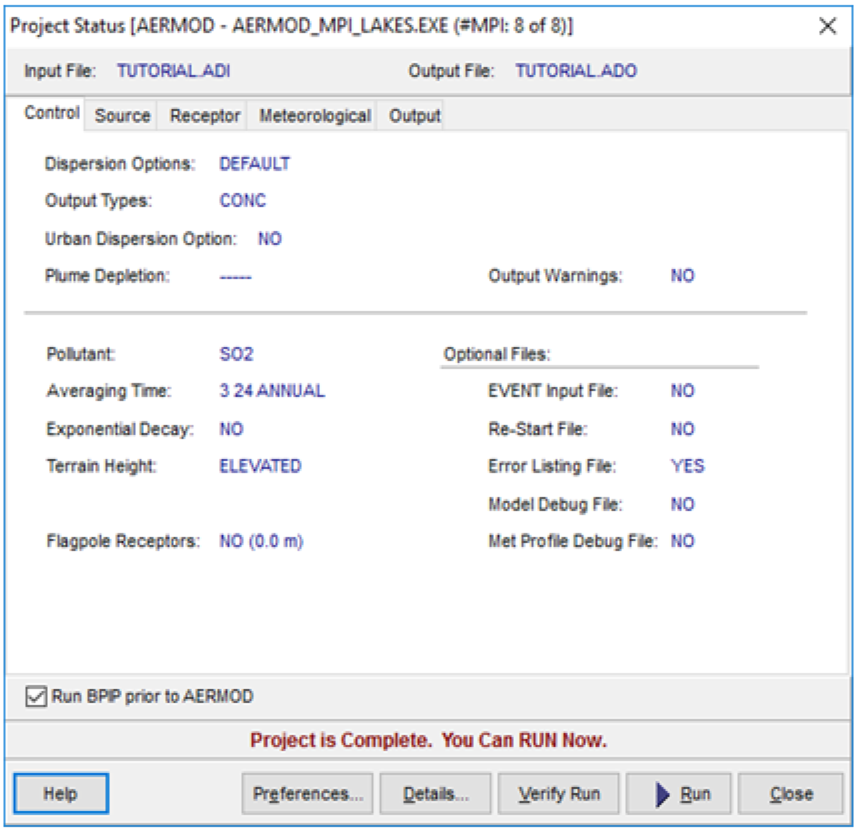
Your project should be complete by now. Before running your project, we suggest that you follow these steps:

Step 1: Make sure you are in the AERMOD model by selecting **Model | AERMOD** from the menu.



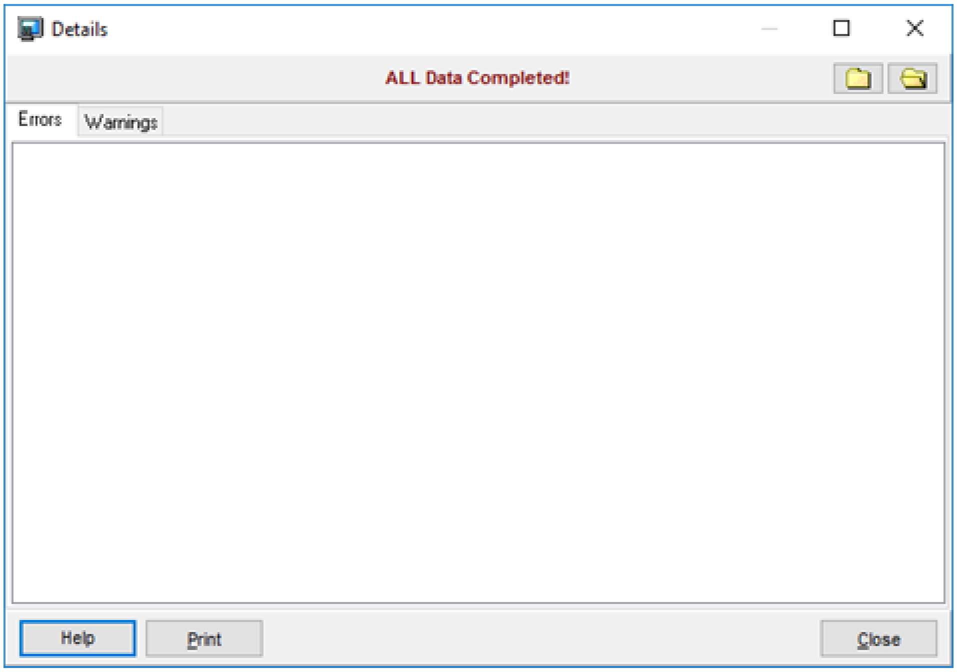
Step 2: Check the status of your project to make sure your options are correct. To do this, select **Run | Status | AERMOD** from the menu, or simply click the **Run** menu toolbar button. The **Project Status** dialog is displayed.



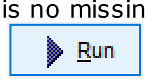


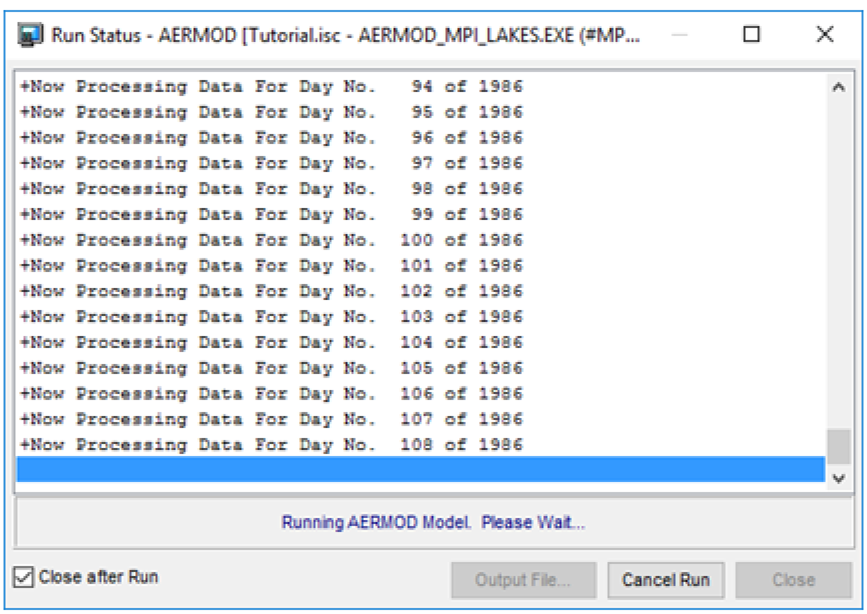
Project Status dialog

Step 3: Check the details of your project. If any crucial piece of information is missing, it will be displayed on the **Details** dialog. Click on the Details... button to open the **Details** dialog.



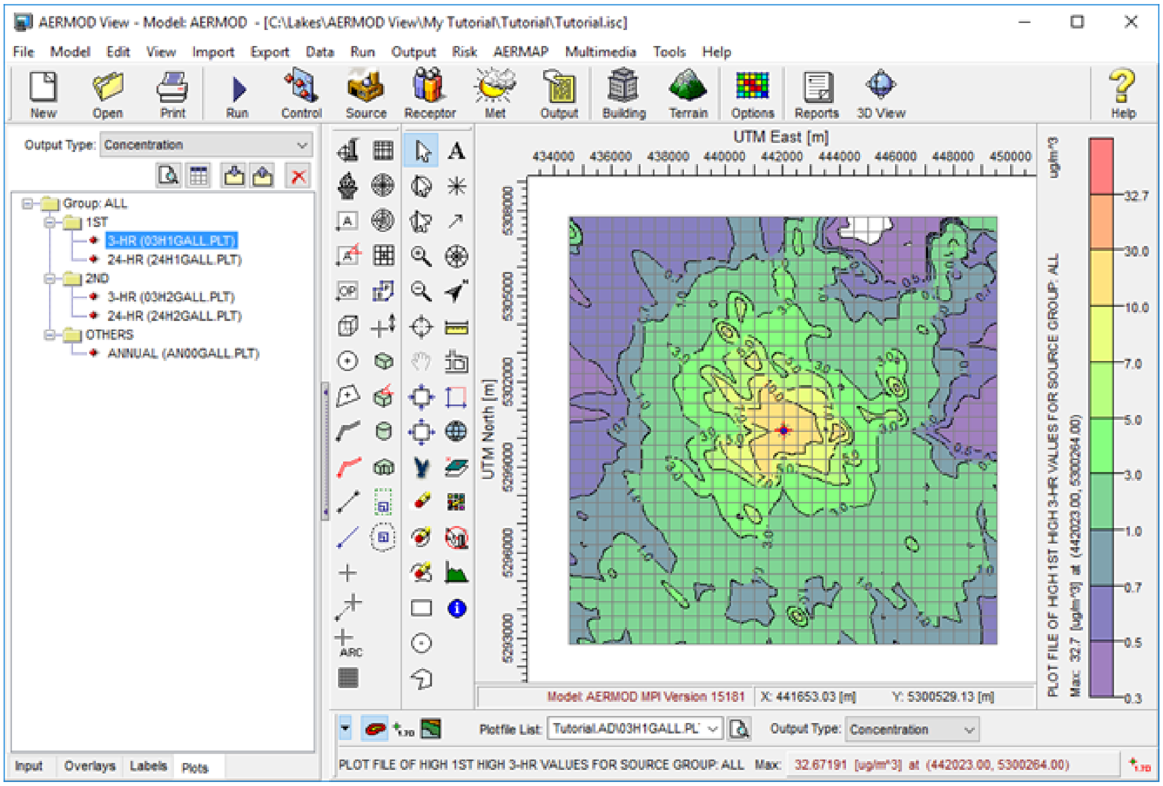
Details dialog

Step 4: If there is no missing information and your project is complete, close the **Details** dialog and click the  button located on the bottom of the **Project Status** dialog.



Run Status dialog

Step 5: Once the run has finished successfully, an information dialog will pop-up indicating your run was successful. Click on the **OK** button. Your contour plot results will be displayed in the drawing area. Zoom out until you can see the entire modeling domain.

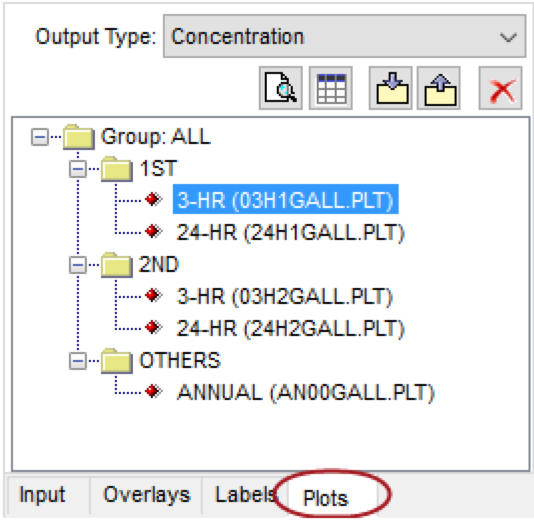


Contour Plot Results - AERMOD Model

Postprocessing of Results

After running the AERMOD model, results from the model run are displayed in the drawing area.

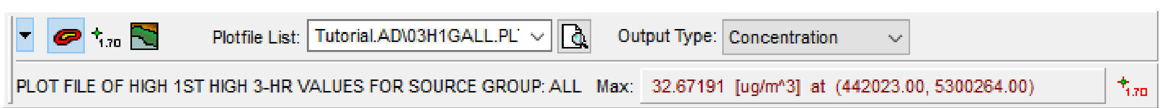
Step 1: From the Tree View located on the left side of the AERMOD View main window, notice that the **Plots** tab was added to the Tree View after successfully running the model. This tab lists all available plot files generated for the current run, grouped by source group and high value.



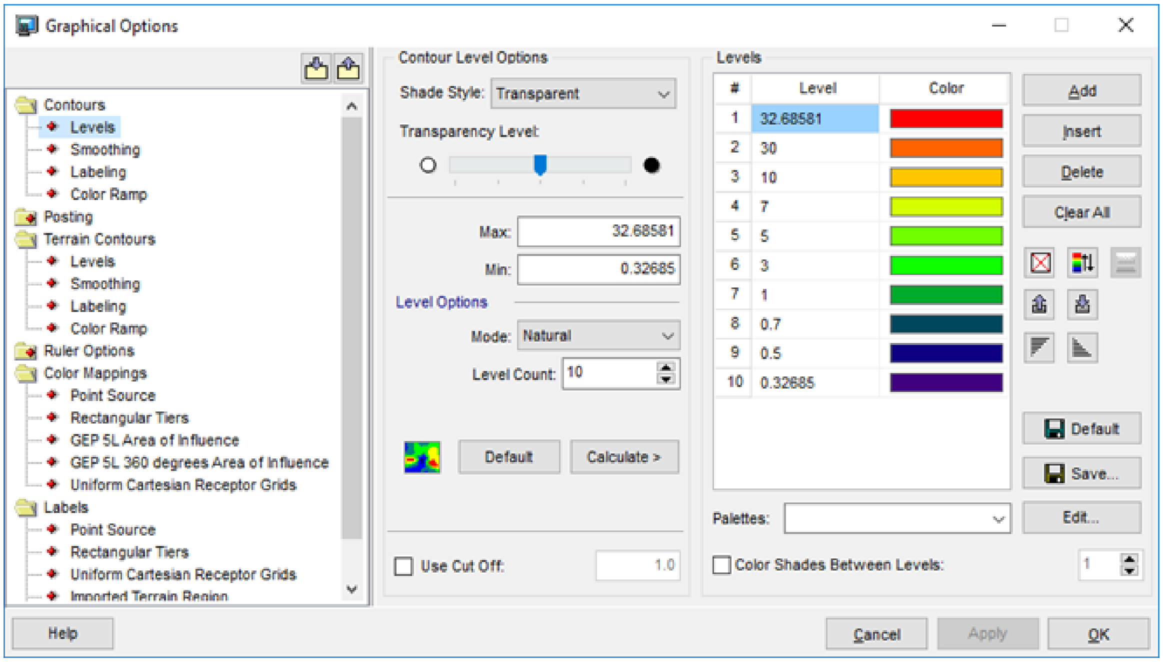
Step 2: From the **Plots** tab, select each plot file to view the output contour results in the drawing area. The results in the drawing area are for the plot file currently highlighted in the Plots tab. As you select a different output plot file the drawing area is refreshed and new contour results are displayed.

Step 3: At the bottom of the interface note that the Graphical Output Toolbar is now displayed.

The Graphical Output Toolbar allows you to easily turn on and off contouring ( ), posting of results ( ), and acts as a quick reference to the maximum result value. This toolbar also allows you to select the desired plot files and output type to be displayed. As you select different output types, the drawing area is refreshed and results are displayed for the new option. Play around with these options and see how the drawing area changes.



Step 4: You can also modify the default options for your contours such as contour levels, labeling, contour lines, and grid size in the Graphical Options dialog. Click on the Graphical Options tool ( ) located on the Annotation Toolbar, or select **Output | Graphical Options...** from the menu. Open this dialog and play around with the several options. Click **OK** when done.

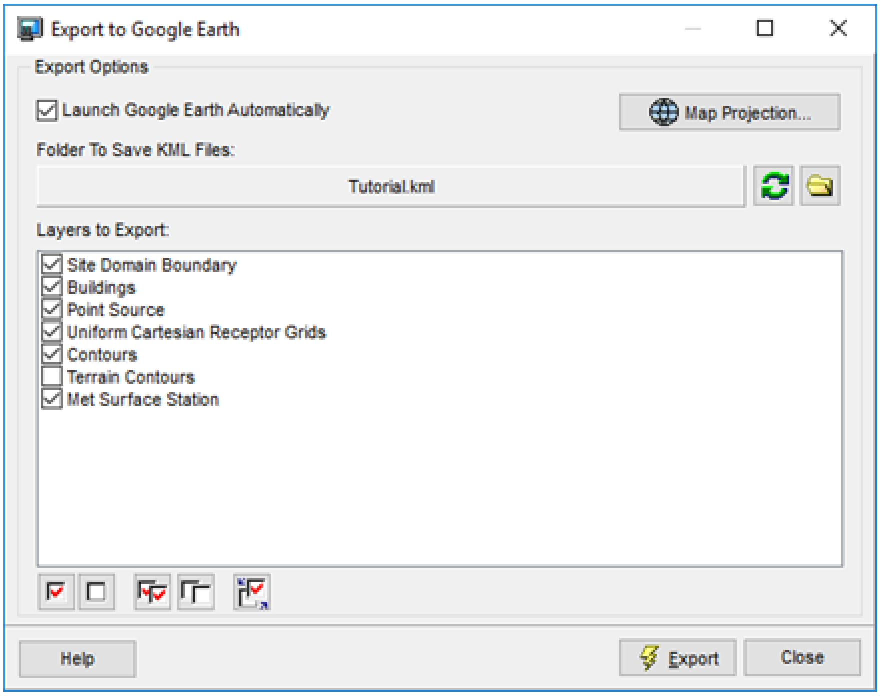


Exporting to Google Earth

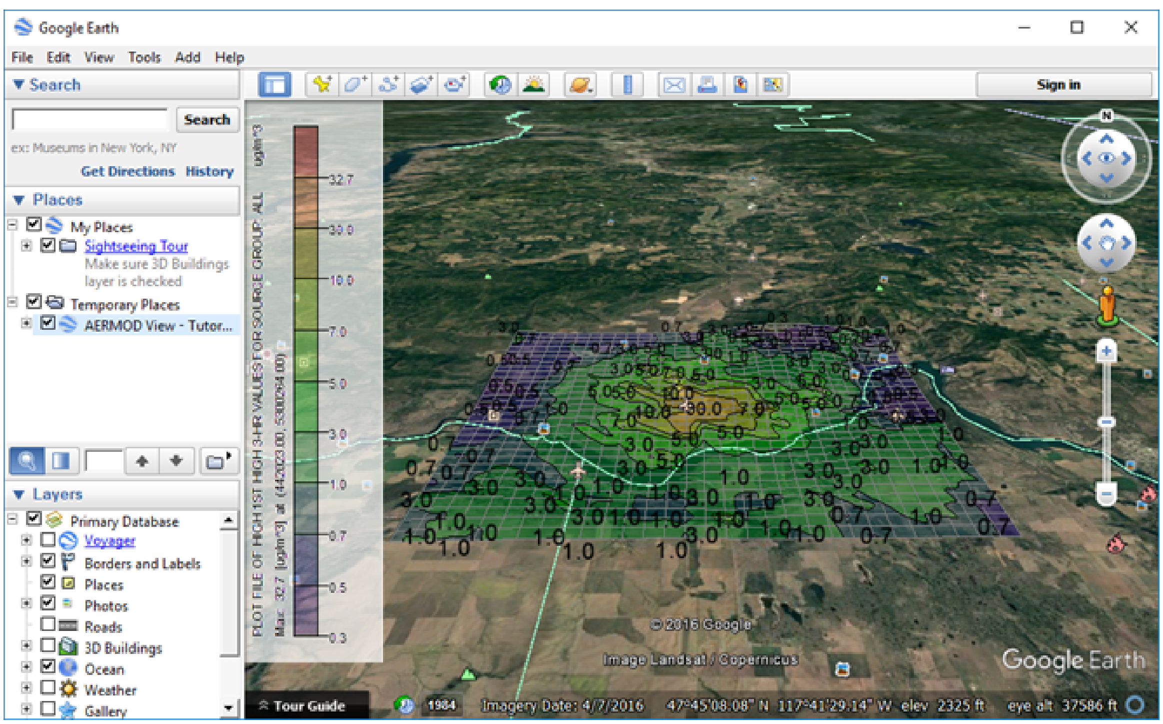
Follow the steps below to visualize your project in Google Earth:

Step 1: Select **Export | Google Earth...** from the menu. The **Export to Google Earth** dialog is displayed. Click the 

button.



Step 2: Google Earth will be launched automatically and you will be able to visualize your entire project in Google Earth (building, stacks, grid, terrain contours, and concentration result contours). You can turn different layers on and off to achieve the desired presentation.



Visualization in Google Earth

Quick Steps to Complete the ISCT3 & ISC-PRIME Tutorials

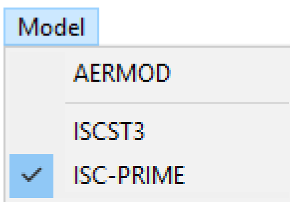
Once you have finished running the AERMOD model, it is time for you to run the ISCST3 and ISC-PRIME models.

The PRIME algorithms have been integrated into the ISCST3 model and called the ISC-PRIME model which contains the same basic options as the ISCST3 model but with enhanced building downwash analysis.

The ISCST3 and ISC-PRIME models request the same met information and both use the same preprocessed meteorological data file. RAMMET View allows you to preprocess the meteorological data file used by the ISCST3 and ISC-PRIME models. Please see the RAMMET View Tutorial for a brief tutorial on how to process met data using RAMMET View.

To complete the ISC-PRIME tutorial, follow the steps below:

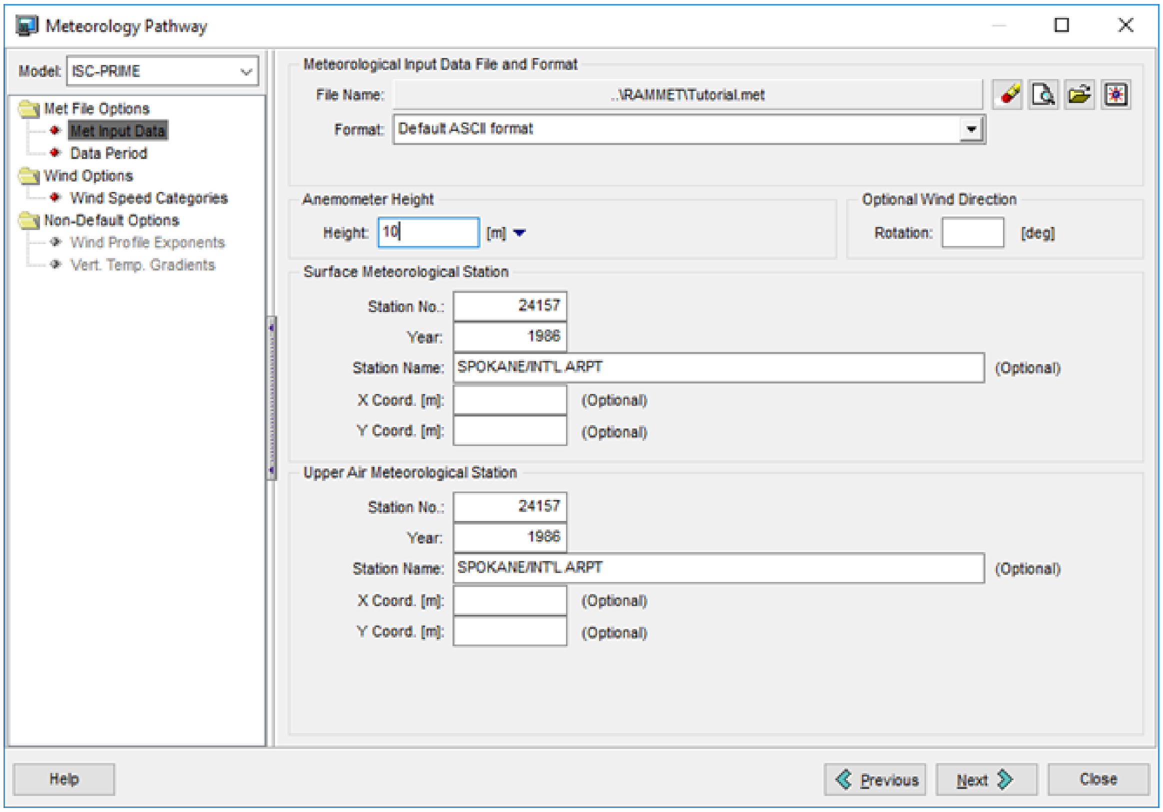
Step 1: Make sure you are in **ISC-PRIME** mode by selecting **Model | ISC-PRIME** from the menu.



Step 2: Click the **Met** menu toolbar button or select **Data | Meteorology Pathway...** from the menu to open the **Meteorology Pathway** dialog.

Step 3: Click on the  button in the **Meteorological Input Data File** panel and select the **Tutorial.met** file that has been preprocessed using RAMMET View (*.met). Specify the additional parameters according to table below:

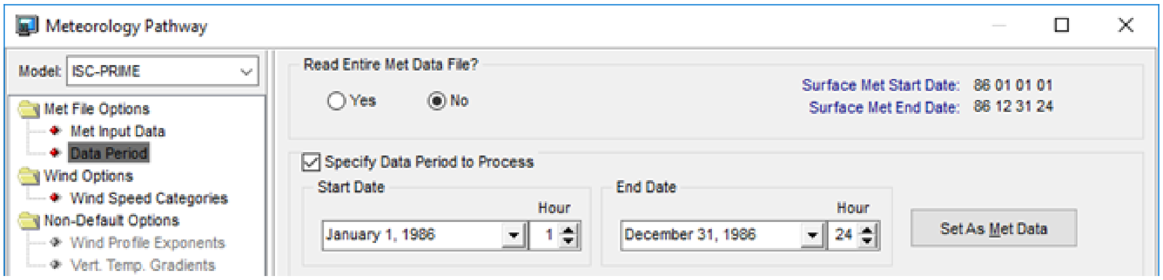
Parameter	Value
Folder Location	C:\Lakes\AERMOD View\Tutorial\RAMMET\
File Name	Tutorial.met
Format	Default ASCII Format
Anemometer Height	10 m



Meteorology Pathway – Met Input Data window

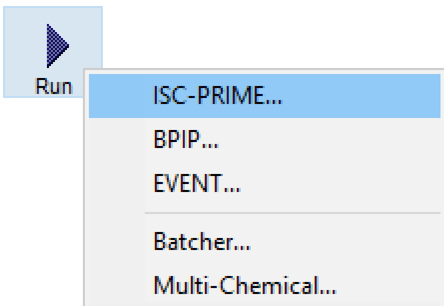
Step 4: Go to the **Data Period** window by pressing the  button. The met data preprocessed under the RAMMET View Tutorial was for a 5-year period. Because we want to compare results among the models, we need to specify that we only want to run the model for year 1986. Specify the information provide in table below. Close the **Meteorology Pathway** window when done.

Parameter	Value
Read Entire Met Data File?	No
Specify Data Period to Process	**Start Date:** January 1, 1986 - Hour 1 **End Date:** December 31, 1986 - Hour 24



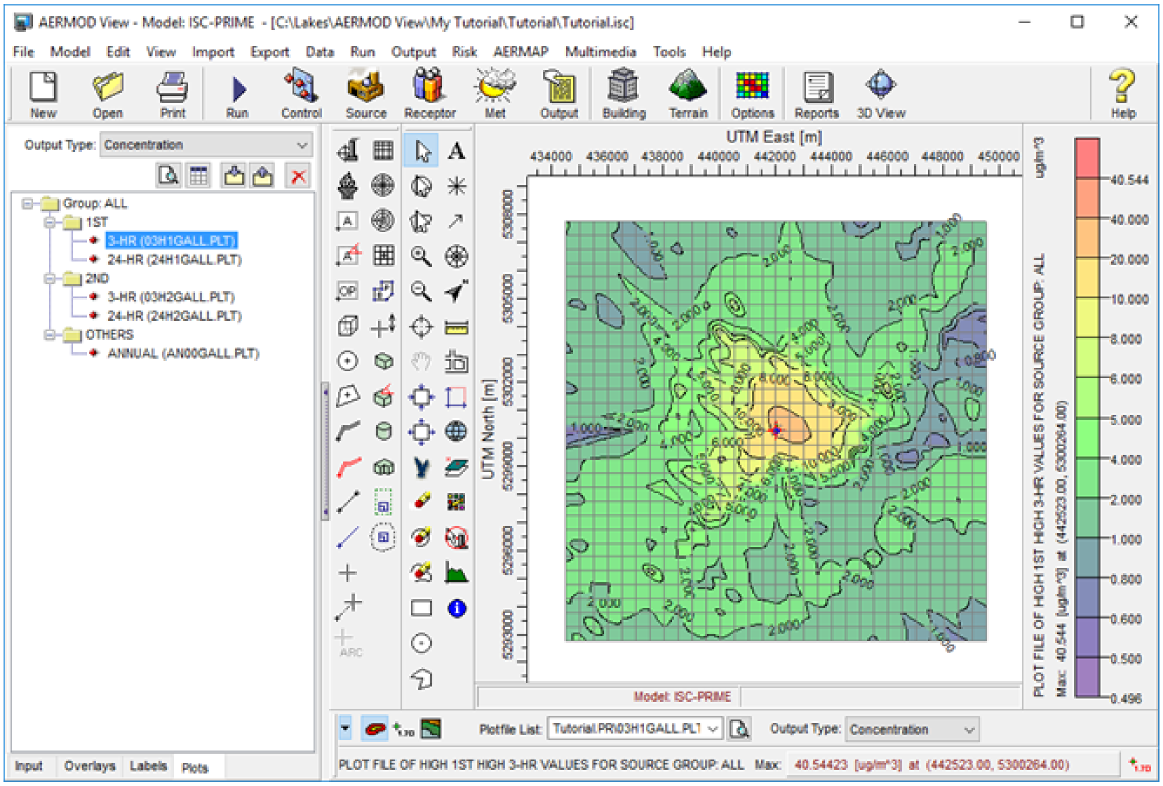
Meteorology Pathway – Data Period window

Step 5: From the AERMOD View main window, press the **Run** menu toolbar button and select **ISC-PRIME** from the pop-up menu. The **Project Status** dialog is displayed.



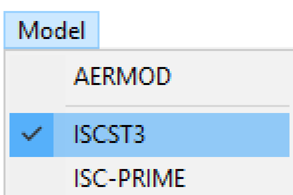
Step 6: If there is no missing information and your project is complete, click the 

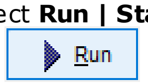
button located on the bottom of the **Project Status** dialog. Once the run has finished successfully, your contour plot results for the ISC-PRIME model will be displayed in the drawing area.



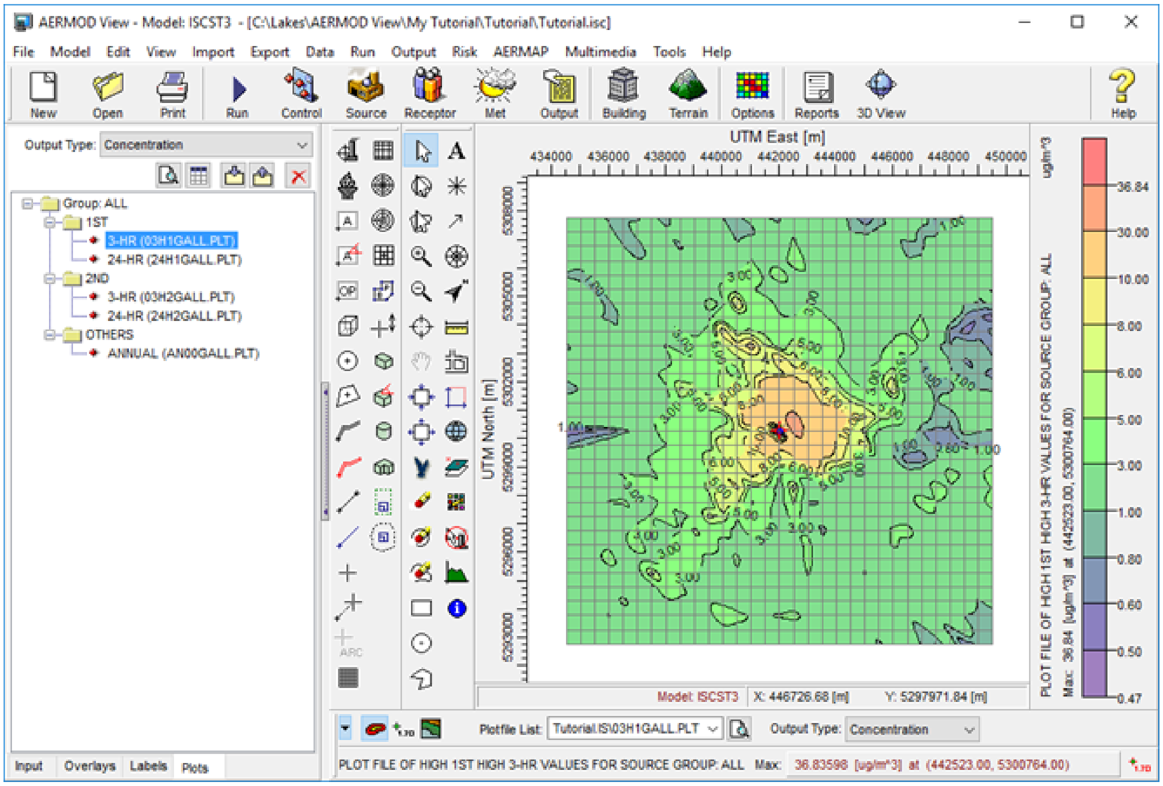
Contour Plot Results - ISC-PRIME Model

Step 7: Now you can run the **ISCST3** model. Make sure you are in the ISCST3 mode by selecting **Model | ISCST3** from the menu.



Step 8: Select **Run | Status | ISCST3** from the menu. The **Project Status** dialog is displayed. Click the  button located on the bottom of the **Project Status** dialog.

Step 9: Once the run has finished successfully, your contour plot results for the ISCST3 model will be displayed in the drawing area.



Contour Plot Results - ISCST3 Model

Comparison of Model Results

Now that you have run all three models, you can carry out a comparison of the model results. Use the tables below to easily enter and compare the model results:

	Concentration Results (ug/m3)		
Averaging Period and High Value	**AERMOD**	**ISCST3**	**ISC-PRIME**
3-Hour - 1st High			
24-Hour - 1st High			
Annual			

Multi-Chemical Tutorial

The AERMOD View Tutorial presented in previous chapters were processed using a single pollutant. In this chapter, we will see how to model emissions from multiple pollutants using the **Multi-Chemical Utility**.

The **Multi-Chemical Tutorial** has the following steps, which should be followed in the order provided below.

Contents:

About the Multi-Chemical Utility

The **Multi-Chemical Utility** is designed to allow each source to emit multiple chemicals, each at a specified emission rate, allowing you to model the contributions from each pollutant quickly and concisely. These chemicals, and emission rates, can be different for each source. This feature is very powerful, as the alternative is to run separate projects for each different emitted chemical.

The **Multi-Chemical Utility** works by creating individual batch/input files for each source, running them through the specified model (AERMOD, ISCST3, or ISC-PRIME), and combining them into a single result to be viewed. The contour plot files contain the contribution from each individual pollutant for all the sources. It should be noted that output files are created for each source, with unitized emission rates, and that chemical specific output files are not created.

While it is very useful and can save a lot of time, the Multi-Chemical Utility does have limitations, namely that it does not support all of the features of the AERMOD, ISCST3, and ISC-PRIME models. Features that are NOT supported include:

- Special pollutant options, such as SO2, NOx, and PM10
- Optional control files, such as Re-Start files
- TOXICS/Gas Dry Deposition when the deposition velocity is calculated by the model
- TOXICS/SCIM Option
- Source Groups other than Group ALL
- Discrete Polar Receptors and Polar Plant Boundary Receptors

- MAXTABLE (Maximum Values) and DAYTABLE (Daily Values) options
- Only chemical specific plot files are generated, any other type of output file is not created (example: Main Output File, Summary File, etc.)
- The main output files that are generated are specific to only one source with a unitized emission rate and should not be used for your modeling concentration results.

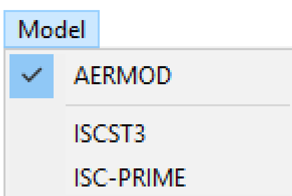
> Since not all options are supported in a multi-chemical run, it is important to inspect your input files. Any information in the input file not supported by the Multi-Chemical Run will be commented out (**). If this is information that is required, you will need to do a regular model run.

Using the Multi-Chemical Utility

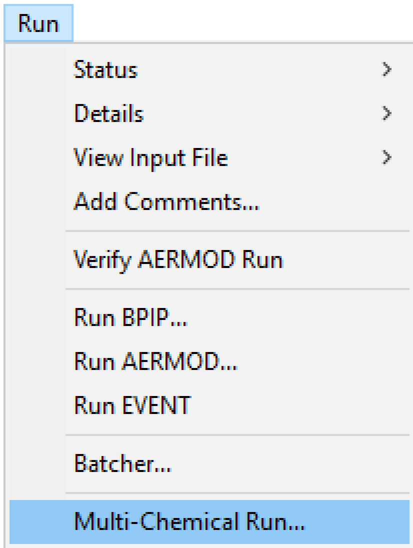
In this tutorial, the project that has been created in the previous chapters will be modified to emit multiple chemicals using the Multi-Chemical Utility. We will be using the AERMOD model for this example.

Step 1: Start the AERMOD View application and open the **Tutorial** project.

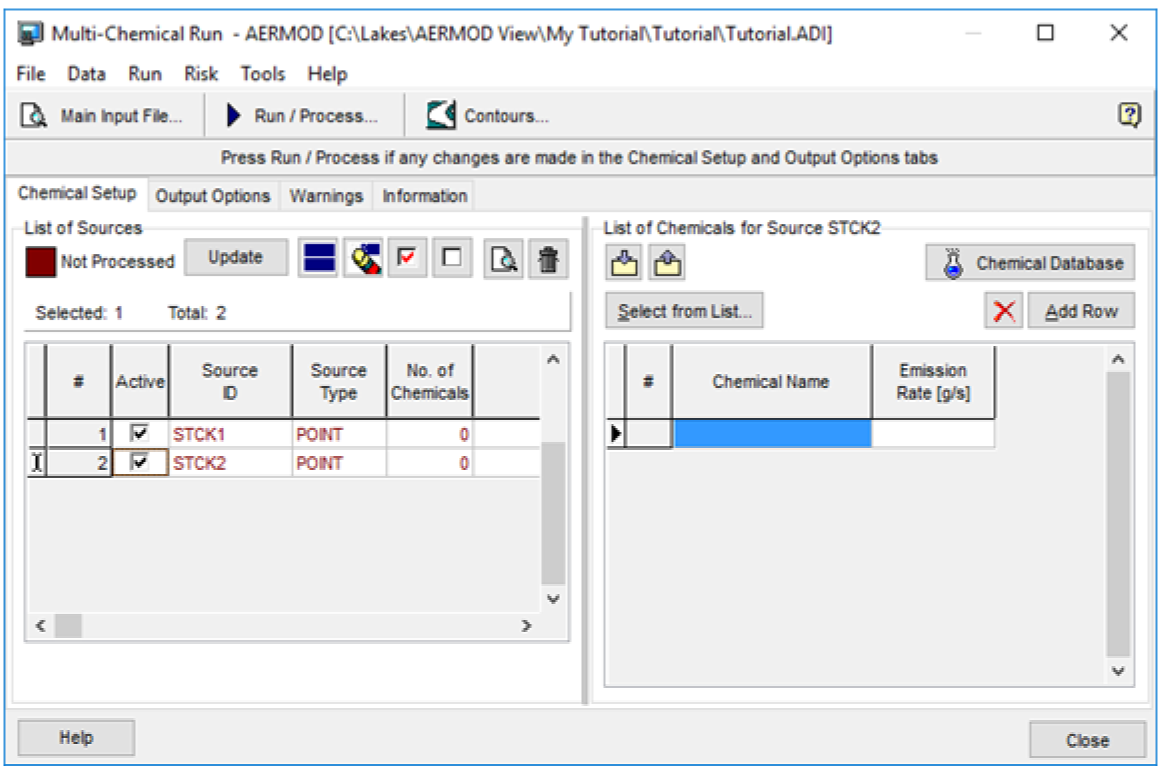
Step 2: Make sure the **AERMOD** model is selected from the **Models** menu.



Step 3: Select **Run | Multi-Chemical Run...** from the menu.



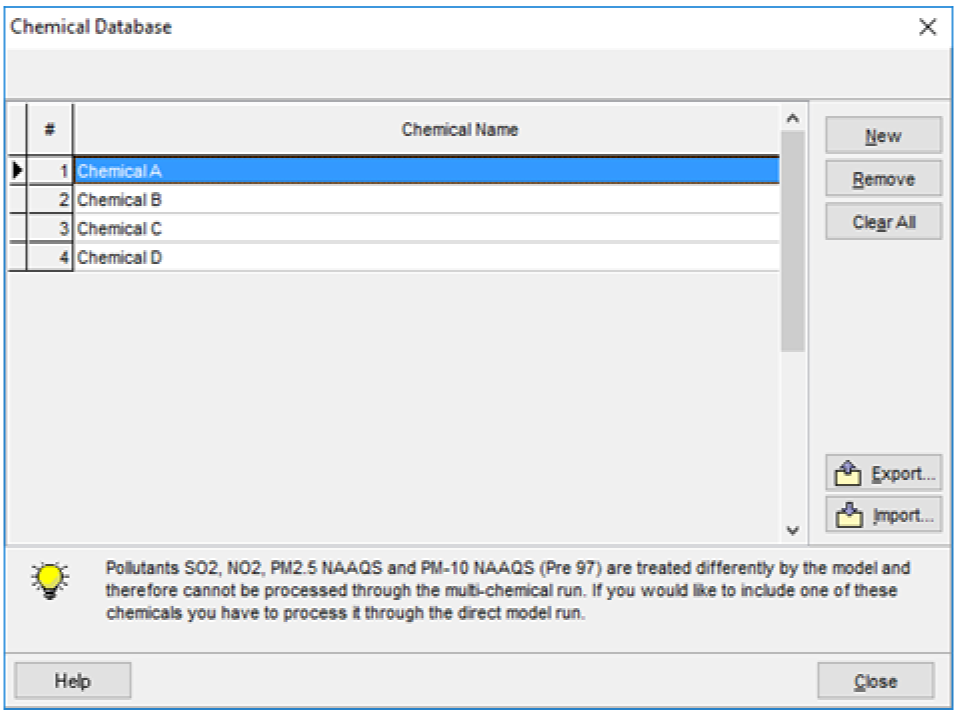
Step 4: The **Multi-Chemical Run** Utility is displayed. The stacks that were specified in the project will be displayed in the **List of Sources** table. Check the **Active** box for all stacks to be included in the **Multi-Chemical** run.



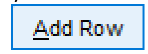
Multi-Chemical Utility – Chemical Setup tab

Step 5: Before emissions can be assigned to the stacks, chemicals will need to be added to the chemical database. Click on the Chemical Database button and specify the chemicals according to table below. To add a new chemical to the database, click on the **New** button. Click on the **Close** button when done.

Parameter	Value
Chemical Names	Chemical A Chemical B Chemical C Chemical D

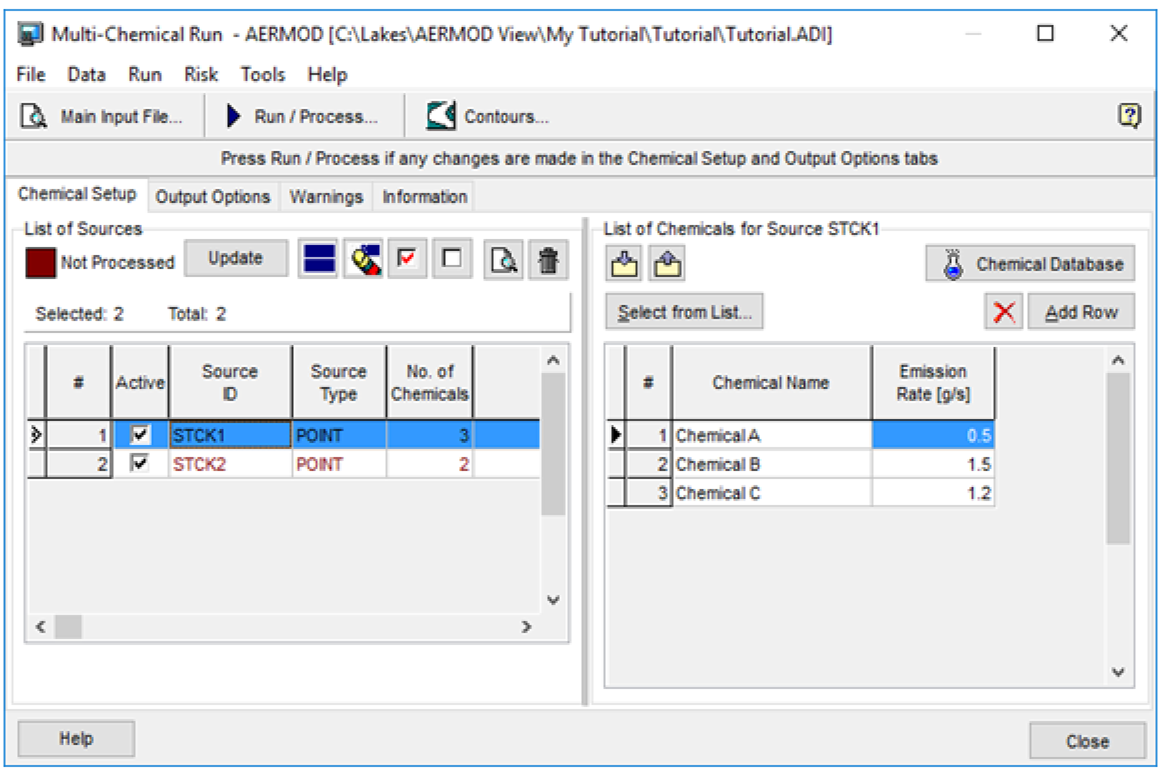


Chemical Database dialog

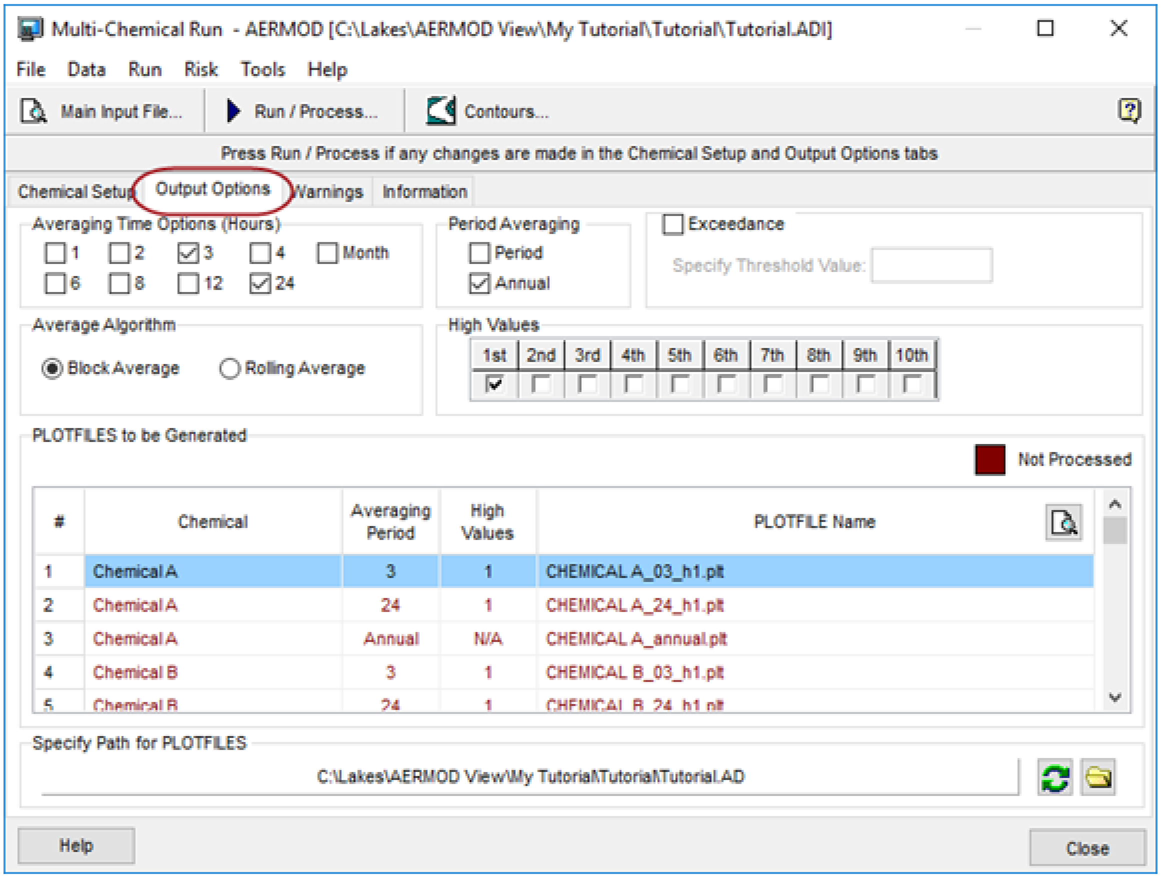
Step 6: You can now specify the chemicals and emission rates for each source. Select **STCK1** from the **List of Sources** table. In the **List of Chemicals** for Source STCK1 window, click on the **Chemical Name** cell to display a drop-down list. You will need to click on the 

button to add successive chemicals. Specify the following chemical and emission rates for the sources:

Source	Chemical Name	Emission Rate [g/s]
STACK1	Chemical A	0.5
	Chemical B	1.5
	Chemical C	1.2
STACK2	Chemical A	1.0
	Chemical D	0.75

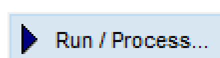


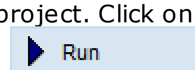
Step 7: Once you have specified the chemicals and emission rates for each source, go to the **Output Options** tab. This tab allows you to specify additional output options for the Multi-Chemical Run.



Multi-Chemical Utility – Output Options tab

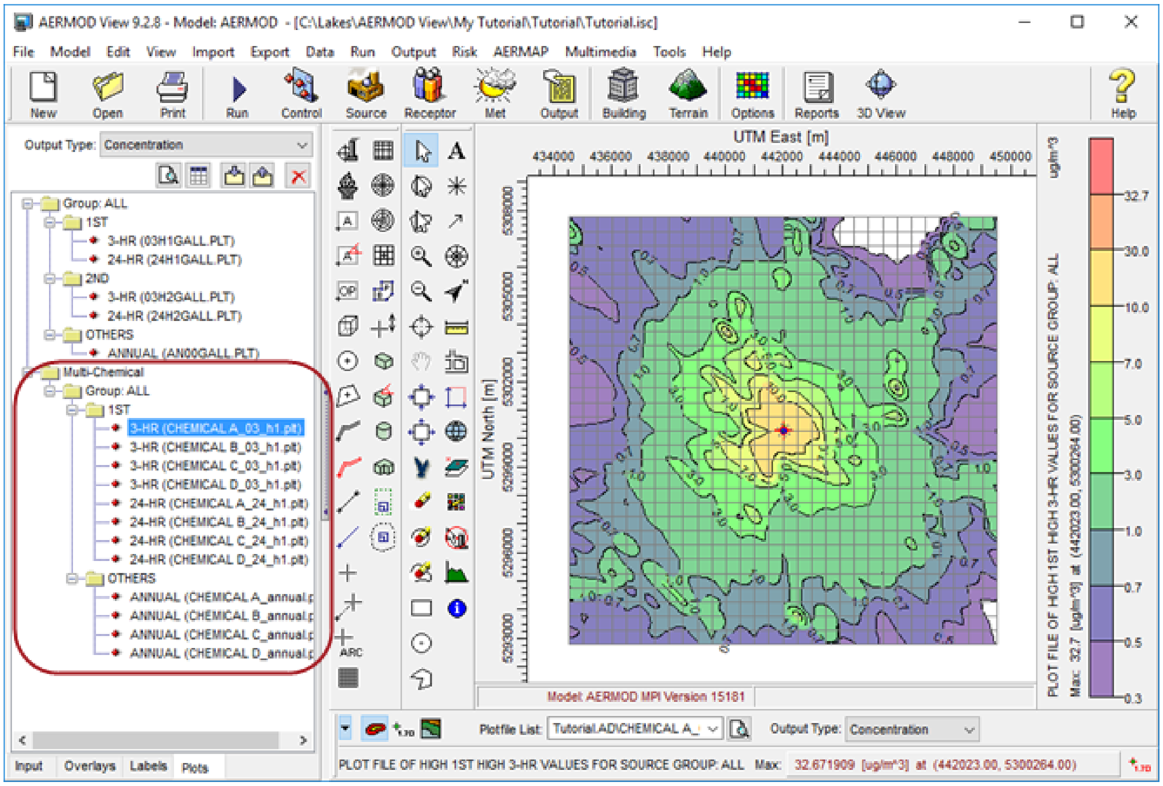
Step 8: The **Warnings** tab displays the information that was ignored from the AERMOD/ISC input file by the Multi-Chemical Run utility and the **Information** tab displays information regarding the options that are not supported in the current version of the Multi-Chemical Run utility. You can browse through these two tabs. For this tutorial we will disregard any warnings.

Step 9: You are now ready to run your project. Click on the 

button. The **AERMOD Batcher** is displayed. Press the 

button.

Step 10: The **Multi-Chemical Run Utility** will automatically generate the required plot files when the **AERMOD Batcher** is closed. When this process is complete, you can close the **Multi-Chemical Utility** and view the generated contour plots in the main AERMOD View interface.



Contour Plot Results for Multi-Chemical Run - AERMOD Model

You have now completed the Multi-Chemical Tutorial Project!

Technical Support

Lakes Environmental is dedicated to providing full technical support for users in **current maintenance**.

> If your software is out of maintenance, please contact our sales team at sales@webLakes.com to renew it.

> Please include the **Version #** of the software you are inquiring about, as well as the **Serial #** that was provided to you with the purchase.

Hours of Operation

Monday - Friday

9:00 a.m. - 5:00 p.m EST See also

Lakes Environmental Software

Tel.: +1.519.746.5995
Fax: +1.519.746.0793

e-mail: support@webLakes.com
Website: www.webLakes.com

> See also System Requirements for AERMOD View.

License Agreement

Please carefully read the following license and warranty information. By installing, archiving copies of, or otherwise using the licensed software, Licensee agrees to be bound by the terms and conditions of this license. Lakes Environmental Software, a division of Lakes Environmental Consultants Inc., retains the ownership of this copy of software. This copy is licensed to you for use under the following conditions:

COPYRIGHT NOTICE

This software is owned by Lakes Environmental Software and is protected by both Canadian copyright law and international treaty provisions. You must treat this software like any other copyrighted material (e.g., a book or musical recording). Lakes Environmental Software authorizes you to make archive copies of the software to protect it from loss. Licensee may not distribute, rent, sub-license, lease, alter, modify, or adapt the software or documentation, including, but not limited to, translating, decompiling, disassembling, or creating derivative works without the prior written consent of Lakes Environmental Software. Licensee agrees that in case of transference of ownership of the software, the transferee must expressly accept all terms and conditions of this agreement and written notice of transference must be provided to Lakes Environmental Software.

The associated warranty is not applicable to any freely available software that may be included as part of the software installation package. Lakes Environmental Software assumes no liability for the use of any freely available software applications.

WARRANTY AND LIABILITY

Lakes Environmental Software warrants that, under normal use, the software application and its' documentation will be free of defects in materials and workmanship for a period of 60 days from the date of purchase. In the event of notification of defects in material or workmanship, Lakes Environmental Software will replace the defective material or documentation.

The above warranty is in lieu of all other warranties, whether written, express, or implied. Lakes Environmental Software specifically excludes all implied warranties, including, but not limited to, loss of profit, and fitness for a particular purpose. In no case shall Lakes Environmental Software assume any liabilities with respect to the use, or misuse, or the interpretation, or misinterpretation, of any results obtained from this software, or for direct, indirect, special, incidental, or consequential damages resulting from the use of this software.

Specifically, Lakes Environmental Software is not responsible for any costs including, but not limited to, those incurred as a result of lost profits or revenue, loss of data, the costs of recovering programs or data, the cost of any substitute program, claims by third parties, or for other similar costs. In no event will Lakes Environmental Software's liability exceed the amount of the license fee.

SOFTWARE OPTIONS AND SERVICES ENABLED ONLY FOR LICENSES UNDER CURRENT MAINTENANCE

This software application contains features and services which may be restricted to use by users with a current maintenance agreement.

In the event that this software contains one or more of the options listed below, these software options will only be available for licenses that are currently in maintenance:

1. Import of Tile Maps
2. Download of NED Terrain Data in Terrain Processor
3. Download of SRTM1 (Version 3) Terrain Data in Terrain Processor
4. Download of CORINE Land Use Data in AERMET View's Land Use Creator
5. Download of EOSD Land Use Data in AERMET View's Land Use Creator

The following services will only be available for licenses that are currently in maintenance:

1. Access to the Online Knowledgebase
2. Software Updates
3. Technical Support (support@webLakes.com)

The high resolution imagery maps, under "Lakes Satellite", are provided generated and provided by large number of copyright holders (the Licensors). Licensors include satellite companies, data aggregators, and mapping companies. Therefore, Lakes Environmental is obligated to pass these copyright owner's use restrictions:

1. The map Content is owned and/or controlled by Lakes and its respective Licensors and is protected by local and international intellectual property laws. Users cannot resale, give away, or cache these maps outside the current application.

2. Licensors grant a non-exclusive, non-transferable license, revocable at any time at Licensors' sole discretion, to access and use the maps strictly in accordance with the terms.

3. Users may not distribute, transfer the right to use, modify, translate, reproduce, resell, sublicense, rent, lease, reverse engineer, or otherwise attempt to discover the source code of or make derivative works of the data and maps from the application.

4. The Data is restricted for use in the specific system for which it was created. Except to the extent explicitly permitted by mandatory laws, you may not extract or reutilize any parts of the contents of the Data, nor reproduce, copy, modify, adapt, translate, disassemble, decompile or reverse engineer any portion of the Data.

5. Use the Content in accordance with the restrictions set out in the applicable local and international laws.

6. Software from Lakes Environmental may provide links to other sites. Lakes Environmental has no control over the third party content, sites, or services and assumes no responsibility for services provided or material created or published on these third-party sites or services. A link to a third-party site does not imply that Lakes Environmental endorses the site or the products or services referenced in the site.

7. The Data may contain inaccurate or incomplete information due to the passage of time, changing circumstances, sources used and the nature of collecting comprehensive geographic data, any of which may lead to incorrect results. The Data does not include or reflect information on - inter alia - travel time and may not include neighborhood safety; law enforcement; emergency assistance; construction work; road or lane closures; road slope or grade; bridge height, weight or other limits; road conditions; or special events depending on the navigation system brand that you possess. Users must assess suitability of the maps for the specific purpose of use.

8. To the extent permitted by local law, Lakes Environmental and/or Licensors, including their respective map and data suppliers, disclaim any warranties, express or implied, of quality, performance, merchantability, fitness for a particular purpose, or non-infringement.

9. Users may not use any automated systems or means, except for those provided by Lakes Environmental Software, for the selection or downloading of the Content.

10. You agree to use the Data in compliance with all applicable laws, rules and regulations, including local laws, rules and regulations of the country or region in which you reside or in which you obtain or use the Data. You agree not to export from anywhere any part of the Data or any direct product thereof, except in compliance with, and with all licenses and approvals required under, applicable export laws, rules and regulations, including but not limited to the laws, rules and regulations administered by the Office of Foreign Assets Control of the U.S. Department of Commerce and the Bureau of Industry and Security of the U.S. Department of Commerce. To the extent that any such export laws, rules or regulations prohibit your, your company, Lakes Environmental, and Licensors, or your supplier from complying with any of its obligations to deliver or distribute the Data, such failure shall be excused and shall not constitute a breach of this Agreement.

The Map Service is provided on “AS IS” and “AS AVAILABLE” basis. Lakes Environmental and Licensors do not warrant that the Map Service will be uninterrupted or error free. No warranty of any kind, either express or implied, including but not limited to warranties of title, non-infringement, merchantability, or fitness for a particular purpose, is made in relation to the availability, accuracy, reliability, information or content of the Service. You expressly agree and acknowledge that the use of the Map Service is at your sole risk and that you may be exposed to content from various sources.

WEB LICENSE

In the event this software was distributed with a web activated license, the software application must be able to periodically establish connectivity to the web license server, maintained by Lakes Environmental, in order for the software application to properly function. Specifically, the software application must successfully communicate with the web license server at a minimum of once every two weeks. Otherwise, software applications are not able to verify security and licensing credentials, which are necessary to maintain an active license status for the software application and registered user.

GOVERNING LAW

This license agreement shall be construed and enforced in accordance with the laws of the Province of Ontario, Canada. Any terms or conditions of this agreement found to be unenforceable, illegal, or contrary to public policy in any jurisdiction will be deleted, but will not affect the remaining terms and conditions of the agreement.

ENTIRE AGREEMENT

This agreement constitutes the entire agreement between you and Lakes Environmental Software.

System Requirements

AERMOD View is a Microsoft® Windows®-based program that can be installed in the following Windows operating systems (32-bit and 64-bit):

- Microsoft Windows 10
- Microsoft Windows 8 & 8.1
- Microsoft Windows 7 (Home Premium, Professional and Ultimate versions)

Before you install **AERMOD View**, make sure you have the following minimum requirements:

- 1 GHz or faster processor
- At least 20 GB of available hard disk space
- At least 2 GB of RAM (4 GB recommended)

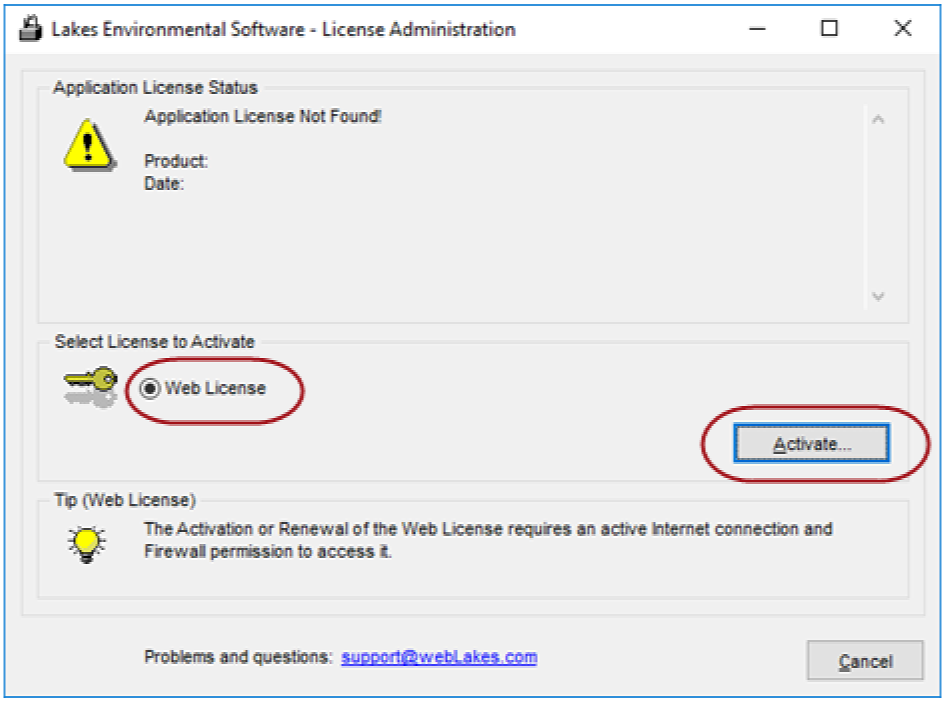
Web License Administration

Find below descriptions on how you can activate and deactivate your AERMOD View web license:

How to Activate the Web License:

Follow the steps below to be able to activate your AERMOD View web license:

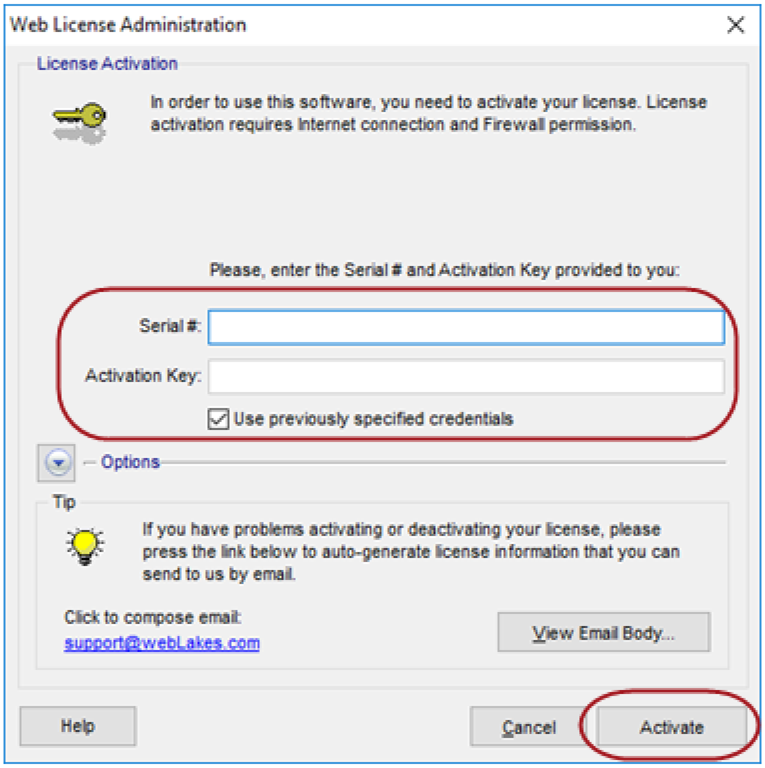
1. After installing AERMOD View and rebooting your computer, login as the user who will be using AERMOD View.
2. Start the AERMOD View application by selecting from the Windows Start Menu **All Programs | Lakes Environmental | AERMOD View | AERMOD View**.
3. The **License Administration** tool should now be displayed. Select **Web License** and click the **Activate** button.



License Administration dialog

4. The **Web License Administration** dialog should now be displayed.

5. Enter the **Serial #** and **Activation Key** provided to you by e-mail and click the **Activate** button. This will open the AERMOD View welcome screen.



Web License Administration dialog

6. Simply click **OK** to begin using **AERMOD View**.

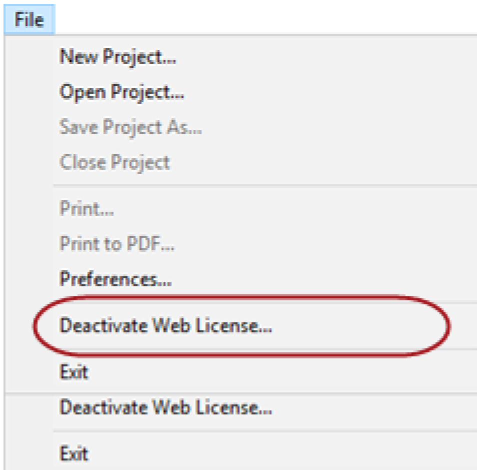
> **Note:** If you are using a proxy server to access the internet, then read the section **How to Use Proxy Options** below.

How to Deactivate the Web License:

Follow the steps below to be able to deactivate your AERMOD View web license:

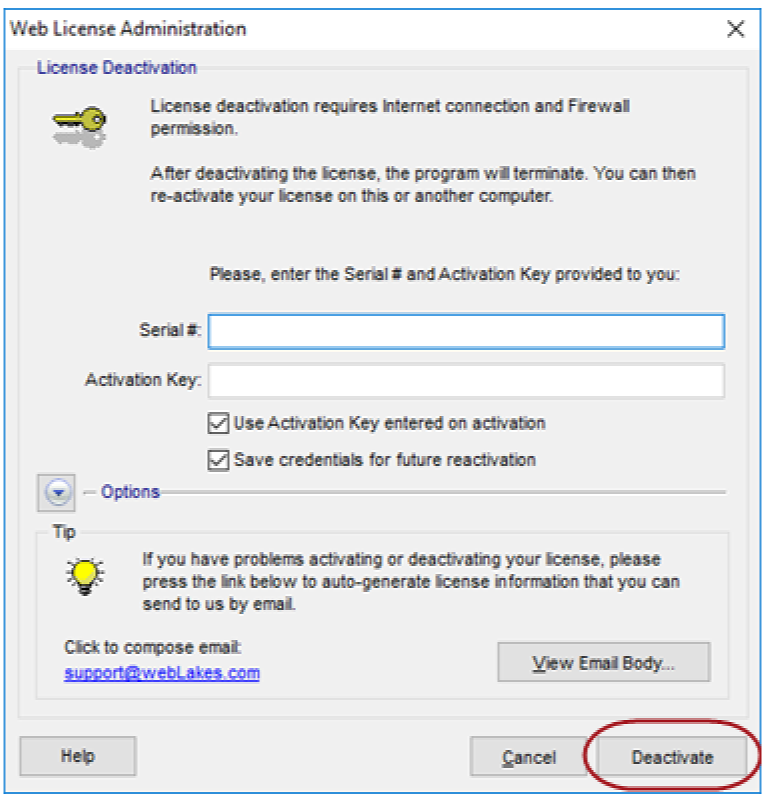
1. Start the AERMOD View application by selecting from the Windows Start Menu **All Programs | Lakes Environmental | AERMOD View | AERMOD View**.

2. Select the menu option **File | Deactivate Web License**.



File Menu

3. The **Web License Administration** dialog is displayed. Click the **Deactivate** button.



Web License Administration dialog

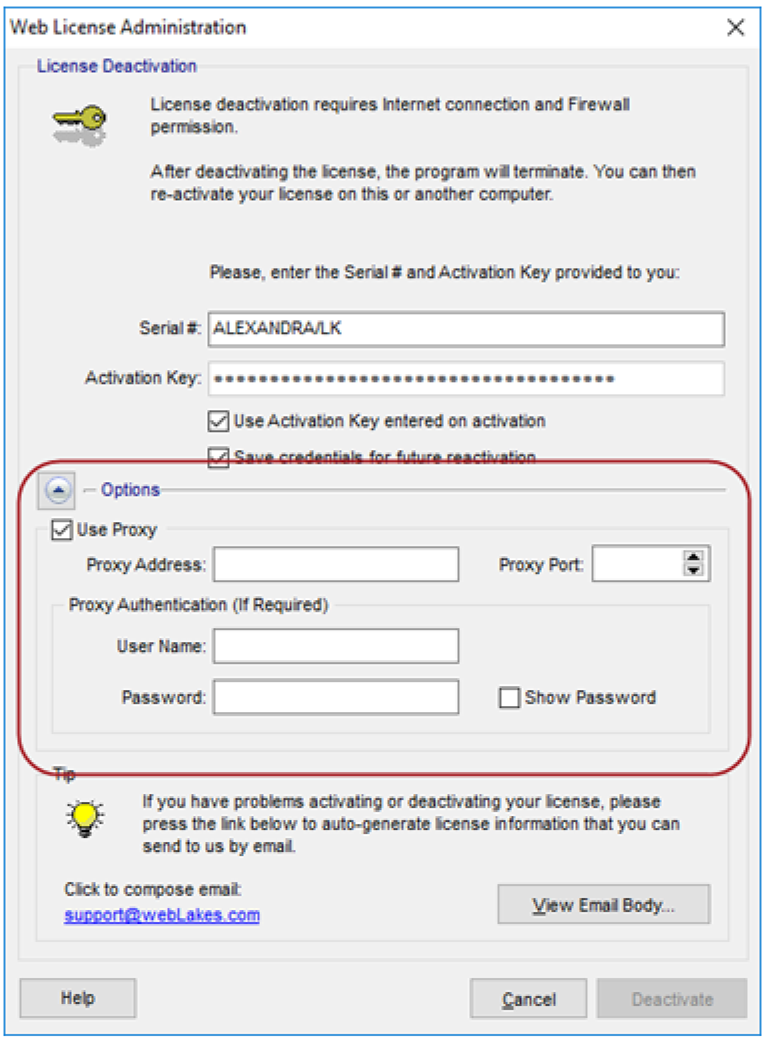
4. Now you can activate your Web License in another computer.

How to Use Proxy Options:

When trying to activate your AERMOD View web license, you may need to specify your organization's proxy address and port in case you are using a proxy server to access the internet.

1. Under the **Web License Administration** dialog, click on the  to expand the **Options** section.

2. Select the **Use Proxy** box and enter the **Proxy Address** and **Proxy Port** for your organization.



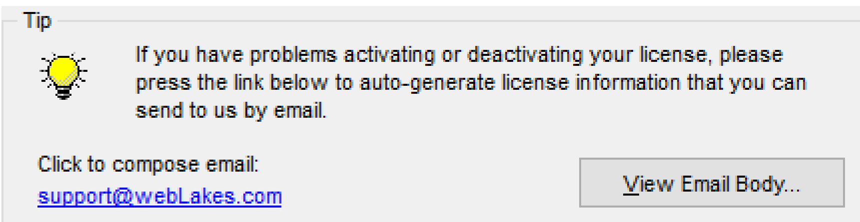
3. If your proxy server requires authentication, enter the **Proxy User Name** and **Proxy Password** in the fields available under the **Proxy Authentication** frame.

4. Click the **Activate** button. This will open the AERMOD View welcome screen.

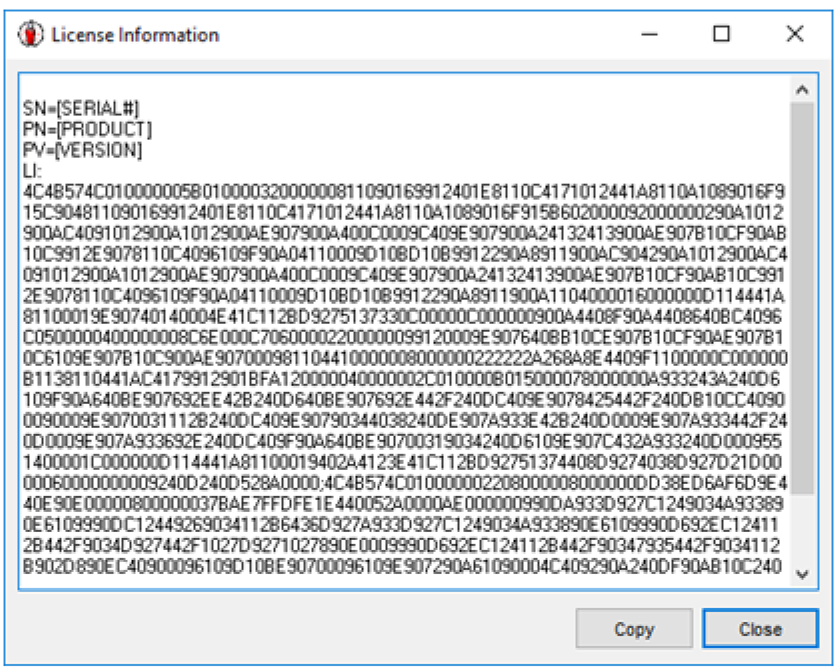
Having Web License Activation Problems?

If you are having problems trying to activate your AERMOD View web license, then follow the steps below:

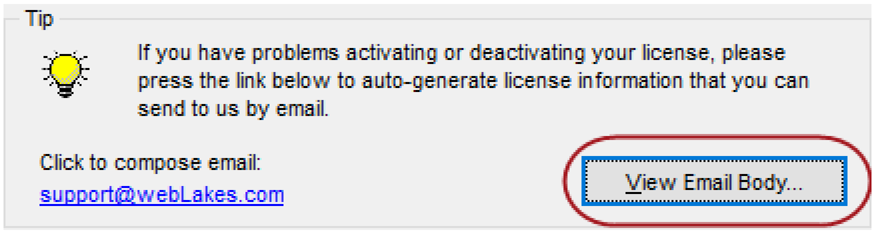
1. Under the Web License Administration dialog, click on the email link: **support@webLakes.com**



2. This will open your email client and will automatically place in the body of the email information about your license.



3. If the body of your email is blank, you can press the **View E-mail Body** button to display the license information that you can copy & paste to your email and send back to Lakes Environmental.



4. Lakes Environmental will receive your web license information and will contact you with a solution to the specified problem.

Available Features with Current Maintenance

The following features are **only** available to users with current maintenance:

- Import of Tile Maps
- Access to the Online Knowledgebase
- Software Updates
- Technical Support (support@webLakes.com)
- Automated Import of NED Terrain Data (NED 1/3 - USA: ~10 m | NED 1- USA, Canada, Mexico: ~30 m) in Terrain Processor
- Automated Import of SRTM1 (Version 3) Terrain Data (Global: ~30 m) in Terrain Processor
- Automated Import of CORINE Land Use Data (Europe: 100 m or 250 m) in AERMET View's Land Use Creator
- Automated Import of EOSD Land Use Data (Canada: 25 m) in AERMET View's Land Use Creator

References

AERMOD View is an interface for the ISCST3, ISC-PRIME, and AERMOD air dispersion models. We have referenced from the following publications:

1. Air/Superfund National Technical Guidance Study Series, Volume 4: *Guidance for Ambient Air Monitoring at Superfund Sites*, Revised EPA Number: 451R93007, NTIS number PB93-199214

2. Air/Superfund National Technical Guidance Study Series: *Emission Factors for Superfund Remediation Technologies*. EPA/450/1-91/001, NTIS PB91-190975/XAB.

3. Eckhoff, P.A., May 1993. *Evaluation of Computer Programs For Calculating Projected Building Widths (Draft)*. U.S. Environmental Protection Agency, Research Triangle Park, North Carolina 27711.

4. Jindal, M., Heinold, D., 1991, "*Development of Particulate Scavenging Coefficients to Model Wet Deposition from Industrial Combustion Sources*". Paper 91-59.7, 84th Annual Meeting - Exhibition of AWMA, Vancouver, BC, June 16-21.

5. Lee, Russell F., July 1, 1993. *Stack-Structure Relationships--Further clarification of our memoranda dated May 11, 1988 and June 28, 1989*. Memorandum to Richard L. Daye.

6. Memorandum (dated October 10, 1985). *Questions and Answers on Implementing the Revised Stack Height Regulation*. From G. T. Helms to Chief, Air Branch Regions I-X.

7. Ontario Ministry of the Environment, July 2005. Air Dispersion Modelling Guideline for Ontario, Version 1.0. PIBS#5165e. Ontario Ministry of the Environment, Toronto, Ontario.

8. Schulman, L.L., and J.S. Scire, 1980: *Modelling Plume Rise from Low-level Buoyant Line and Point Sources*. Proceedings, Second Joint Conference on Applications of Air Pollution Meteorology, 24-28 March, New Orleans, LA. pp. 133-139.

9. Sheih, C.M., M.L. Wesley, and B.B. Hicks, 1979. *Estimated Dry Deposition Velocities of Sulfur Over the Eastern U.S. and Surrounding Regions*. Atmos. Environ., 13: 361-368.

10. Tikvart, Joseph A., May 11, 1988. *Stack-Structure Relationships*. Memorandum to Richard L. Daye.

11. Tikvart, Joseph A., June 28, 1989. *Clarification of Stack-Structure Relationships*. Memorandum to Regional Modeling Contacts, Regions I-X.

12. U.S. Environmental Protection Agency, 1985. *Guideline for Determination of Good Engineering Practice Stack Height (Technical Support Document for the Stack Height Regulations)*. Revised EPA-450/4-80-023R. U.S. Environmental Protection Agency, Research Triangle Park, North Carolina 27711.

13. U.S. Environmental Protection Agency, 1986. *User's Guide to the Building Profile Input Program*. Revised EPA-454/R-93-038. U. S. Environmental Protection Agency, Research Triangle Park, NC.

14. U. S. Environmental Protection Agency, 1987. *Guidelines on Air Quality Models (Revised) and Supplement A*. EPA-450/2-78-027R. U. S. Environmental Protection Agency, Research Triangle Park, NC.

15. U.S. Environmental Protection Agency, 1992a. *User's Guide For the Industrial Source Complex (ISC2) Dispersion Models - Volume I*. EPA-450/4-92-008a, Office of Air Quality Planning and Standards, Research Triangle Park, North Carolina 27711.

16. U.S. Environmental Protection Agency, 1992b. *Workbook of Screening Techniques for Assessing Impacts of Toxic Air Pollutants (Revised)*. EPA-454/R-92-024. Office of Air Quality Planning and Standards, Research Triangle Park, North Carolina.

17. U.S. Environmental Protection Agency, 1992c. *Screening Procedures For Estimating The Air Quality Impact Of Stationary Sources (Revised)*. EPA-454/R-92-019, U.S. Environmental Protection Agency, Research Triangle Park, North Carolina 27711.

18. U.S. Environmental Protection Agency, 1995a. *User's Guide for the Industrial Source Complex (ISC3) Dispersion Models - Volume 1 (Revised)*. EPA-454/B-95-003a. Office of Air Quality Planning and Standards, Research Triangle Park, NC.

19. U.S. Environmental Protection Agency, 1995b. *SCREEN3 Model User's Guide*. EPA-454/B-95-004. Office of Air Quality Planning and Standards, Research Triangle Park, NC.

20. U.S. Environmental Protection Agency, 1995c. *User's Guide to the Building Profile Input Program*. Revised EPA-454/R-93-038. U. S. Environmental Protection Agency, Research Triangle Park, NC.

21. U.S. Environmental Protection Agency, 1995d. *User's Guide for the Industrial Source Complex (ISC3) Dispersion Models - Volume I – User Instructions*. EPA-454/B-95-003a. U.S. Environmental Protection Agency. Research Triangle Park, NC 27711.

22. U.S. Environmental Protection Agency, 1995e. *User's Guide for the Industrial Source Complex (ISC3) Dispersion Models - Volume II – Description of Model Algorithms*. EPA-454/B-95-003a. U.S. Environmental Protection Agency. Research Triangle Park, NC 27711.

23. U.S. Environmental Protection Agency, 1997. *Addendum to ISC3 User's Guide - The Prime Plume Rise and Building Downwash Model*. Submitted by Electric Power Research Institute. Prepared by Earth Tech, Inc., Concord, MA.

24. U. S. Environmental Protection Agency, 1999. *Addendum - User's Guide for the Industrial Source Complex (ISC3) Dispersion Models - Volume 1*. Office of Air Quality Planning and Standards, Research Triangle Park, NC.

25. U. S. Environmental Protection Agency, 2002. *Addendum for the User's Guide for the Industrial Source Complex (ISC3) Dispersion Models - Volume 1 – User Instructions*. Office of Air Quality Planning and Standards, Research Triangle Park, NC.

26. U.S. Environmental Protection Agency, 2004. *User's Guide to the Building Profile Input Program*. Revised EPA-454/R-93-038. U. S. Environmental Protection Agency, Research Triangle Park, NC.

27. U.S Environmental Protection Agency, 2011. *Addendum - User's Guid For The AERMOD Terrain Preprocessor (AERMAP).* EPA-454/B-03-003*.* Office of Air Quality Planning and Standards, Research Triangle Park, NC

28. U.S. Environmental Protection Agency, 2017. *User's Guide for the AMS/EPA Regulatory Model (AERMOD)*. EPA-454/B-16-011. U.S. Environmental Protection Agency, Office of Air Quality Planning and Standards, Air Quality Assessment Division, Air Quality Modeling Group.

29. Wark, K. and C. Warner, 1981. *Air Pollution: Its Origin and Control, 2nd Edition*, Harper Collins Publishers.

EPA Error Message Codes

The EPA models check the input runstream to identify parameters that are missing or potentially in error, and the input source and meteorological data are checked and flagged for possible erroneous values. Error messages are reported in two ways:

- **Detailed Message Listing File:** You can request a detailed list of all messages generated by the model in a separate output file in the **Control Pathway - Event/Error Files** window.

- **Message Summary:** The models output a summary of messages within the main output file. This message table gives the number of messages of each type, together with a detailed list of all the fatal errors and warning messages.

There are two message summaries provided in the standard output file. The first is a summary for the model setup, located after the echo of input runstream file images and before the input data summary. If there are no errors or warnings generated during the setup processing, then the model simply reports that "**SETUP Finishes Successfully**."

```
 --------- Summary of Total Messages --------

A Total of          0 Fatal Error Message(s)
A Total of          0 Warning Message(s)
A Total of          0 Informational Message(s)


   ******** FATAL ERROR MESSAGES ********
              ***  NONE  ***


   ********   WARNING MESSAGES   ********
              ***  NONE  ***

***********************************
*** SETUP Finishes Successfully ***
***********************************
```

The second message is a summary table of the model execution and is located at the very end of the standard output result file. It sums up the messages generated by the complete model run - both setup processing and run-time processing.

```
*** Message Summary : ISCST3 Model Execution ***

 --------- Summary of Total Messages --------

A Total of           86 Fatal Error Message(s)
A Total of           16 Warning Message(s)
A Total of          240 Informational Message(s)

A Total of            0 Calm Hours Identified

A Total of           15 Missing Hours Identified


   ******** FATAL ERROR MESSAGES ********
CO E100     8 EXPATH: Invalid Pathway Specified. The Troubled Pathway is FF
MX E510  3479 METEXT:Fatal Error Occurs During Reading of the File of    MET-INP
MX E450  3490 CHKDAT:Error in Meteor. File - Record Out of Sequence at  98052610

   ********   WARNING MESSAGES   ********
SO W320    88 PPARM :Input Parameter May Be Out-of-Range for Parameter       QS
RE W282    98 CHK_EL:RecElev < SrcBase; See non-DFAULT HE>ZI option in  MCB#9
MX W410   210 METQA :Flow Vector Out-of-Range.   KURDAT=                98010918
MX W430  1651 METQA :Ambient Temperature Data Out-of-Range.  KURDAT=    98031019

   ****************************************
   *** ISCST3 Finishes UN-successfully ***
   ****************************************
```

If there are any fatal errors or warning messages a more detailed summary is provided. This summary provides a message count for each type of message, and a detailed listing of each fatal error and warning message generated. Three types of messages can be produced by the models during the processing of input runstream images and during model calculations. These are described briefly below:

- **Fatal Errors:** These will halt any further processing, except to identify additional error conditions (type E);

- **Warnings:** These do not halt processing but indicate possible errors or suspect conditions (type W); and

- **Informational messages:** There may be of interest to the user but have no direct bearing on the validity of the results (type I).

The messages have a consistent structure which contains the pathway ID, indicating which pathway the messages are generated from; the message type followed by a three-digit message number; the line number of the input runstream image file for setup messages (or the meteorology hour number for runtime messages); the name of the module (e.g. the subroutine name) from which the message is generated; a detailed message corresponding to the message code; and an 8-character simple hint to help the user spot the possible source of the problem.

The following is an example of a detailed message generated from the CO pathway with the message syntax is explained in more detail below:

CO E100 8 EXPATH: Invalid Pathway Specified. The Troubled Pathway is FF

or

PW Txxx LLLL mmmmmm: MESSAGE Hints

where:

- **PW:** Pathway ID (CO, SO, RE, ME, EV, or OU) (1:2) or MX for met data extraction, or CN for calculation messages
- **T:** Message type (E, W, I) (4:4)
- **xxx:** Numeric message code (a 3 digit number) (5:7)
- **LLLL:** The line number of the input runstream image file where the message occurs; If message occurs in runtime operation, the hour number of the meteorology file is given (9:12)
- **mmmmmm:** Name of the code module from which the message is generated (14:19)
- **MESSAGE:** Detailed message for this code (22:71)
- **Hints:** Hints to help you determine the nature of errors (keyword, pathway where the error occurs,...etc.) (73:80)

The 3-digit message codes are grouped into general categories corresponding to the different stages of the processing. See the following sections for a description of each of these messages:

- Input Runstream Image Structure Processing (100 - 199)
- Parameter Setup Processing (200 - 299)
- Data and Quality Assurance Processing (300 - 399)
- Run Time Message Processing (400 - 499)
- Input/Output Message Processing (500 - 599)

Input Runstream Image Structure Processing (100 - 199)

Extracted from AMS/U.S. EPA User's Guide for the AERMOD Regulatory Model (EPA 2004)

This type of message indicates problems with the basic syntax and/or structure of the input runstream image. Typical messages include errors like "Missing mandatory keyword", "Illegal Keyword", ..., etc. If a fatal error of this kind is detected in a runstream image, a fatal error message is written to the message file and any attempt to process data is prohibited, although the remainder of the runstream file is examined for other possible errors. If a warning occurs, data may still be processed, although the inputs should be checked carefully to be sure that the condition causing the warning does not indicate an error.

100: Invalid Pathway Specified. The pathway ID should be a 2 character string. It should be one of the following: CO for control pathway, SO for source pathway, RE for receptor pathway (or EV for event pathway for EVENT processing), ME for meteorology data setting pathway, and OU for output format pathway. Its position is normally confined to columns 1 and 2 (1:2) of the input runstream file. However, the model does allow for a shift of the entire input runstream file of up to 3 columns. If the inputs are shifted, then all input records must be shifted by the same amount. The invalid pathway is repeated at the end of the message.

105: Invalid Keyword Specified. The keyword ID should be an 8-character string. Its position is normally confined to columns 4 to 11 (4:11) of the input runstream file. However, the model does allow for a shift of the entire input runstream file of up to 3 columns. If the inputs are shifted, then all input records must be shifted by the same amount. There should be a space between keyword ID and any other data fields. For a list of valid keywords, refer to Appendix A or Appendix B. The invalid keyword is repeated at the end of the message.

110: Keyword is Not Valid for This Pathway. The input keyword is a valid 8-character string, but it is not valid for the particular pathway. Refer to Appendix A, Appendix B or Section 3 for the correct usage of the keyword. The invalid keyword is repeated at the end of the message.

115: Starting and Finishing Statements do not match. Only One STARTING and one FINISHED statement,respectively, is allowed at the very beginning and the very end of each pathway block. Check the position and frequency to make sure the input runstream file meets the format requirement. The pathway during which the error occurs is included at the end of the message.

120: Pathway is Out of Sequence. The pathways are not input in the correct order. The correct order is CO, SO, RE, ME, and OU for the AERMOD and AERMOD model, and CO, SO, ME, EV, and OU for EVENT processing. The offending pathway is given as a hint.

125: Missing FINISHED Statement - Runstream file is incomplete. One or more FINISHED statements are missing. A 5-digit status variable is given as a hint. Each digit corresponds to a pathway in the appropriate order, and is a '1' if the pathway is complete and a '0' if the FINISHED is missing. For example, a status of '10111' indicates that the SO pathway was missing a FINISHED statement. Normally such an error will generate additional messages as well.

130: Missing Mandatory Keyword. To run the model, certain mandatory keywords must present in the input runstream file. For a list of mandatory keywords, see Appendix A or Appendix B. For more detailed information on keyword setup, see the description of message code 105. The missing keyword is included with the message.

135: Duplicate Non-repeatable Keyword Encountered. More than one instance of a non-repeatable keyword is encountered. For a list of non-repeatable keywords, see Appendix A or Appendix B. The repeated keyword is included with the message.

140: Invalid Placement of Keyword. A keyword has been placed out of the acceptable order, or a STARTING or FINISHED keyword has been placed in an INCLUDED file. The order for most keywords is not critical, but the relative order of a few keywords is important for the proper interpretation of the input data. The keyword reference in Section 3 identifies any requirements for the order of keywords. The keyword that was improperly placed is included with the message.

145: Conflicting Options: MULTYEAR and Re-Start Option. The multiple year option for processing PM-10 values makes use of the re-start routines in the model with some slight changes to handle the period averages from year to year. As a result, the MULTYEAR keyword cannot be specified with either the SAVEFILE or INITFILE keywords.

150: Conflicting Options: MULTYEAR for Wrong Pollutant. The multiple year option is provided specifically for the processing of PM-10 values to obtain the "high-sixth-high in five years" design value. Its treatment of the high short term values for multiple year periods is not consistent with existing air quality standards for other pollutants. To use the MULTYEAR option, the user must specify a pollutant type (on the CO POLLUTID card) of PM-10, PM10, or OTHER.

152: ELEVUNIT card must be first for this pathway. The ELEVUNIT card must be the first non-commented card after STARTING when used on the SO or RE pathway. This requirement is made in order to simplify reviewing runstream files to determine the elevation units used for sources and receptors.

154: Conflicting Options: SCIM option cannot be used with the specified option, causing a fatal error.

155: Conflicting Decay Keyword. The AERMOD model allows for the user to specify the rate of exponential decay either in terms of the half-life (HALFLIFE keyword) or the decay coefficient (DCAYCOEF keyword). If both keywords are specified, then only the first one will be used, and inputs for the second one will be ignored.

156: Option ignored - not valid with SCIM. The specified option is not valid with the SCIM option and is ignored. This is not a fatal err

157: Wet SCIM option is not operational yet and the input is ignored. Since the model does not include wet deposition algorithms yet, the wet SCIM option is not operational.

158: Wet SCIM option is not operational yet and the input is ignored. Since the model does not include wet deposition algorithms yet, the wet SCIM option is not operational.

160: Duplicate ORIG Secondary Keyword for GRIDPOLR. Only one origin card may be specified for each grid of polar receptors. The network ID for the effected grid is included with the message.

170: Invalid Secondary Key for Receptor GRID. The network ID for the effected grid is included with this message. Refer to Appendix B for the correct syntax of secondary keywords.

175: Missing Secondary Keyword END for Receptor Grid. The END secondary keyword is required for each grid of receptors input by the user (keywords GRIDCART and GRIDPOLR). It signals the end of inputs and triggers the processing of data for that particular network.

180: Conflicting Secondary Keyword for Receptor Grid. Two incompatible secondary keywords have been input for the same grid of receptors, e.g. GDIR and DDIR for the keyword GRIDPOLR, where GDIR specifies to generate directions with uniform spacing, and DDIR specifies that discrete, non-uniform directions are being specified.

185: Missing Receptor Keywords. No Receptors Specified. Since none of the RE pathway keywords are mandatory, a separate error check is made to determine if any of the RE keywords are specified. At least one of the following keywords must be present: GRIDCART, GRIDPOLR, DISCCART, DISCPOLR, or EVALCART.

190: No Keywords for OU Pathway and No PERIOD or ANNUAL Averages. All of the OU pathway keywords are optional, and in fact the model will run if no keywords are specified on the OU pathway as long as PERIOD or ANNUAL averages are being calculated. However, if there are no OU keywords and no PERIOD or ANNUAL averages, then there will be no output generated by the model, and this fatal error message will be generated.

195: Incompatible Option Used With SAVEFILE or INITFILE. Either a non-fatal message to warn the user that DAYTABLE results will be overwritten if the model run is re-started, or a fatal error message generated if the TOXXFILE option is selected with either the SAVEFILE or INITFILE options.

197: Post-1997 PM10 processing option without the MAXIFILE option is incompatible with the EVENTFIL option. Threshold violations generated through the MAXIFILE option are the only events that are compatible with post-1997 PM10 processing for EVENT processing.

198: The non-default TOXICS option is required in order to use the specified option, such as the SCIM option.

Parameter Setup Processing (200 - 299)

Extracted from AMS/U.S. EPA User's Guide for the AERMOD Regulatory Model (EPA 2004)

This type of message indicates problems with processing of the parameter fields for the runstream images. Some messages are specific to certain keywords, while others indicate general problems, such as an invalid numeric data field. If a fatal error of this kind is detected in a runstream image, a fatal error message is written to the message file and any attempt to process data is prohibited, although the remainder of the runstream file is examined for other possible errors. If a warning occurs, data may still be processed, although the inputs should be checked carefully to be sure that the condition causing the warning does not indicate an error.

200: Missing Parameter(s). No options were selected for the indicated keyword. Check Appendix B for the list of parameters for the keyword in question.

201: Not Enough Parameters Specified For The Keyword. Check if there are any missing parameters following the indicated keyword. See Appendix B for the required keyword parameters.

202: Too Many Parameters Specified For The Keyword. Refer to Appendix B or Section 3 for the list of acceptable parameters.

203: Invalid Parameter Specified. The inputs for a particular parameter are not valid for some reason. Refer to Appendix B or Section 3. The invalid parameter is included with the message.

204: Option Parameters Conflict. Forced by Default to: Some parameters under the indicated keyword conflict with the other model parameters setting. Refer to Appendix B or Section 3 for the correct parameter usage. The default setting is specified with the message.

205: No Option Parameter Setting. Forced by Default to: No setting was specified for a particular parameter. Refer to Appendix B or Section 3 for the correct parameter usage. The default setting is specified with the message.

206: Regulatory DFAULT Specified With Non-default Option. The DFAULT option on the CO MODELOPT card always overrides the specified non-default option, and a warning message is generated.

207: No Parameters Specified. Default Values Used For. The keyword for which no parameters are specified is included with the message. Refer to Appendix B or Section 3 for a discussion of the default condition.

208: Illegal Numerical Field Encountered. The model may have encountered a non-numerical character for a numerical input, or the numerical value may exceed the limit on the size of the exponent, which could potentially cause an underflow or an overflow error.

209: Negative Value Appears For A Non-negative Variable. The effected variable name is provided with the message.

210: Number of Short Term Averages Exceeds Maximum. The user has specified more short term averages on the CO AVERTIME card than the model array limits allow. This array limit is dynamically allocated at model runtime based on model inputs, and is stored in the NAVE variable. While the model still performs this test, this message should never occur.

211: Duplicate Parameter(s) Specified for Keyword. A duplicate parameter or set of parameters has been specified for the indicated keyword. For example, if more than one POSTFILE keyword is included for the same averaging period and source group, then this error message will be generated.

212: END Encountered Without (X,Y) Points Properly Set. This error occurs during setting up the grid of receptors for a Cartesian Network. This message may occur for example if X-coordinate points have been specified without any Y-coordinate points for a particular network ID.

213: ELEV Inputs Inconsistent With Option: Input Ignored. This happens when the user inputs elevated terrain heights for receptors when the FLAT option is specified. The input terrain heights are ignored and the model proceeds with FLAT terrain modeling.

214: ELEV Inputs Inconsistent With Option: Defaults Used. This happens when the user does not input elevated terrain heights for receptors when the default option of incorporating elevated terrain effects is used. The model assumes that the missing terrain heights are at 0.0 meters for those receptors and proceeds with ELEV terrain modeling.

215: FLAG Inputs Inconsistent With Option: Input Ignored. This happens when the user inputs receptor heights above ground for flagpole receptors when the FLAGPOLE keyword option has not been specified. The input flagpole heights are ignored in the model calculations.

216: FLAG Inputs Inconsistent With Option: Defaults Used. This happens when the user does not input receptor heights above ground for flagpole receptors when the FLAGPOLE keyword option has been specified. The model assumes that the missing flagpole heights are equal to the default value specified on the CO FLAGPOLE card. If no default height is specified on the FLAGPOLE card, then a default of 0.0 meters is assumed.

217: More Than One Delimiter In A Field. For example, 12//34 is an illegal input data item for the DAYRANGE card, and STACK1--STACK-20 is an illegal specification for a range of sources.

218: Number of (X,Y) Points Not Match With Number Of ELEV Or FLAG. Check the number of elevated terrain heights or flagpole receptor heights for the gridded network associated with the indicated line number in the runstream file.

219: Number Of Receptors Specified Exceeds Maximum. The user has specified more receptors on the RE pathway than the model array limits allow. This array limit is dynamically allocated at model runtime based on model inputs, and is stored in the NREC variable. While the model still performs this test, this message should never occur.

220: Missing Origin (Use Default = 0,0) In GRIDPOLR. This is a non-fatal warning message to indicate that the ORIG secondary keyword has not been specified for a particular grid of polar receptors. The model will assume a default origin of (X=0, Y=0).

221: Missing Distance Setting In Polar Network. No distances have been provided (secondary keyword DIST) for the specified grid of polar receptors.

222: Missing Degree Or Distance Setting In Polar Network. Missing a secondary keyword for the specified grid of polar receptors.

223: Missing Distance or Degree Field. No data fields have been specified for the indicated secondary keyword.

224: Number of Receptor Networks Exceeds Maximum. The user has specified more receptor networks of gridded receptors on the RE pathway than the model array limits allow. This array limit is dynamically allocated at model runtime based on model inputs, and is stored in the NNET variable. While the model still performs this test, this message should never occur.

225: Number of X-Coords Specified Exceeds Maximum. The user has specified more X-coordinate values for a particular grid of receptors than the model array limits allow. This array limit is dynamically allocated at model runtime based on model inputs, and is stored in the IXM variable. While the model still performs this test, this message should never occur.

226: Number of Y-Coords Specified Exceeds Maximum. The user has specified more Y-coordinate values for a particular grid of receptors than the model array limits allow. This array limit is dynamically allocated at model runtime based on model inputs, and is stored in the IYM variable. While the model still performs this test, this message should never occur.

227: No Receptors Were Defined on the RE Pathway. Either through lack of inputs or through errors on the inputs, no receptors have been defined.

228: Default(s) Used for Missing Parameters on Keyword. Either an elevated terrain height or a flagpole receptor height or both are missing for a discrete receptor location. Default value(s) will be used for the missing parameter(s).

229: Too Many Parameters - Inputs Ignored on Keyword. Either an elevated terrain height or a flagpole receptor height or both are provided when the corresponding option has not been specified. The unneeded inputs are ignored.

231: Too Many Numerical Values Specified. Too many values have been specified for the type of input indicated.

232: Number Of Specified Sources Exceeds Maximum. The user has specified more sources than the model array limits allow. This array limit is dynamically allocated at model runtime based on model inputs, and is stored in the NSRC variable. While the model still performs this test, this message should never occur.

233: Building Dimensions Specified for a Non-POINT Source. Building dimensions can only be specified for a POINT source, since the VOLUME and AREA source algorithms do not include building downwash.

234: Too Many Sectors Input. For example, the user may have input too many building heights or widths for a particular source.

235: Number of Source Groups Specified Exceeds Maximum. The user has specified more source groups than the model array limits allow. This array limit is dynamically allocated at model runtime based on model inputs, and is stored in the NGRP variable. While the model still performs this test, this message should never occur.

236: Not Enough BUILDHGTs Specified for a Source ID. There should be 36 building heights.

237: Not Enough BUILDWIDs Specified for a Source ID. There should be 36 building widths.

239: Not Enough QFACTs Specified for a Source ID. The number of variable emission rate factors specified for a particular source is less than the model expects based on the variable emission rate flag. Check the EMISFACT keyword on the SO pathway in Appendix B of Section 3 for the appropriate number.

241: Not Enough BUILDLENs Specified for a Source ID. There should be 36 building lengths specified.

246: Not Enough XBADJs Specified for a Source ID. There should be 36 along-flow distances from the stack to the center of the upwind face of building specified.

247: Not Enough YBADJs Specified for a Source ID. There should be 36 across-flow distances from the stack to the center of the upwind face of building specified.

248: No Sources Were Defined on the SO Pathway. There must be at least one LOCATION card and one SRCPARAM card to define at least one source on the SO pathway. Either no cards were input or there were errors on the inputs.

250: Duplicate XPNT/DIST or YPNT/DIR Specified for GRID. One of the grid inputs, either an X-coordinate, Y-coordinate, polar distance range or polar direction, has been specified more than once for the same grid of receptors. This generates a non-fatal warning message.

252: Duplicate Receptor Network ID Specified. A network ID for a grid of receptors (GRIDCART or GRIDPOLR keyword) has been used for more that one network.

254: Number of Receptor Arcs Exceeds Maximum. This array limit is dynamically allocated at model runtime based on model inputs, and is stored in the NARC variable. While the model still performs this test, this message should never occur.

256: EVALFILE Option Used Without EVALCART Receptors. The EVALFILE output option provides model results designed for model evaluation purposes based on receptors grouped by arc. Such receptors must be identified using the EVALCART keyword. If EVALFILE is selected without any EVALCART receptors, this fatal error message will be generated.

260: Number of Emission Factors Exceeds Maximum. The user has selected an option for variable emission rate factors that exceeds the array storage limit for emission rate factors. This array limit is dynamically allocated at model runtime based on model inputs, and is stored in the NQF variable. While the model still performs this test, this message should never occur.

262: First Vertex Does Not Match LOCATION for AREAPOLY Source. The coordinates of the first vertex defined for an AREAPOLY source must match the source location coordinates provided on the SO LOCATION card.

264: Too Many Vertices Specified for an AREAPOLY Source. The number of vertices specified on the AREAVERT cards must match the number given on the SRCPARAM card for that source.

265: Not Enough Vertices Specified for an AREAPOLY Source. The number of vertices specified on the AREAVERT cards must match the number given on the SRCPARAM card for that source.

270: Number of High Values Specified Exceeds Maximum. The user has selected a high short term value on the OU RECTABLE card that exceeds the array storage limit for high values by receptor. This array limit is dynamically allocated at model runtime based on model inputs, and is stored in the NVAL variable. While the model still performs this test, this message should never occur.

280: Number of Maximum Values Specified Exceeds Maximum. The user has selected a value for the number of overall maximum values on the OU MAXTABLE card that exceeds the array storage limit for overall maximum values. This array limit is dynamically allocated at model runtime based on model inputs, and is stored in the NMAX variable. While the model still performs this test, this message should never occur.

294: PERIOD and ANNUAL averages are both selected on the AVERTIME card. The user can only specify one long-term averaging option, either PERIOD or ANNUAL average.

295: Invalid Averaging Period Specified for the SCREEN Mode. The SCREEN mode of AERMOD can only be used with 1-hour averages.

298: Error Allocating Storage for Setup Arrays. An error occurred while allocating storage for the setup arrays, indicating that there is insufficient memory available on the computer for the model to run. Try closing other applications and/or reducing the size of the run. An estimate of memory requirements is provided on the first page of the output file.

299: Error Allocating Storage for Result Arrays. An error occurred while allocating storage for the result arrays, indicating that there is insufficient memory available on the computer for the model to run. Try closing other applications and/or reducing the size of the run. An estimate of memory requirements is provided on the first page of the output file.

Data and Quality Assurance Processing (300 - 399)

Extracted from AMS/U.S. EPA User's Guide for the AERMOD Regulatory Model (EPA 2004)

This type of message indicates problems with the actual values of the parameter data on the input runstream image. The basic structure and syntax of the input card is correct, but one or more of the inputs is invalid or suspicious. These messages include quality assurance checks on various model inputs. Typical messages will tell the consistency of parameters and data for the setup and run of the model. If a fatal error of this kind is detected in a runstream image, a fatal error message is written to the message file and any attempt to process data is prohibited. If a warning occurs, data may or may not be processed, depending on the processing requirements specified within the run stream input data.

300: Specified Source ID Has Not Been Defined Yet. The message indicates that the user attempts to use a source ID on a keyword before defining this source ID on a SO LOCATION card. It could indicate an error in specifying the source ID, an omission of a LOCATION card, or an error in the order of inputs.

310: Attempt to Define Duplicate LOCATION Card for Source. There can be only one LOCATION card for each source ID specified. The source ID is included with the message.

313: Attempt to Define Duplicate EVENTPER Card for the specified EVENT. There can be only one EVENTPER card for each EVENT specified. The EVENT name is included with the message.

315: Attempt to Define Duplicate SRCPARAM Card for Source. There can be only one SRCPARAM card for each source ID specified. The source ID is included with the message.

319: No Sources Included in the Specified Source Group. This is a non-fatal warning message indicating that a source group has been defined that does not include any sources. The source group ID is provided.

320: Source Parameter May Be Out-of-Range for Parameter. The value of one of the source parameters may be either too large or too small. The name of the parameter is provided with the message. Use the line number provided to locate the card in question.

325: Negative Exit Velocity (Set=1.0E-5) for Source ID. The exit velocity for the specified source ID was input as a negative value. Since the model currently cannot handle sources with downward momentum, the exit velocity is set to a very small value (1.0E-5 m/s) and modeling proceeds. This non-fatal message is generated to warn the user that the input may be in error.

330: Mass Fraction Parameters Do Not Sum to 1. (within +/- 2 percent) for a particular source.

332: Mass Fraction Parameter Out-of-Range for a particular source. Must be between 0.0 and 1.0, inclusive.

334: Particle Density Out-of-Range for a particular source. Must be greater than 0.0.

340: Possible error in the PROFBASE input. The value input as the base elevation (above MSL) for the potential temperature file is less than zero.

342: Source ID Mismatch in the Hourly Emissions File. The source ID read from the hourly emissions file does not match what is expected based on the SO HOUREMIS card. A source ID and/or date may be out of order or missing in the hourly emissions file. The source ID read from the file is provided with the error message.

344: Hourly Emission Rate is Zero. The emission rate read from the hourly emission file is zero for the specified date. This is written as an informational message, and does not halt processing of the data.

350: Julian Day Out Of Range. This error occurs if the Julian Day selected is less than zero or greater than 366. Check ME setup to ensure the Julian Day selection.

352: Missing Field on MULTYEAR Card for Pre-1997 PM10. When using the MULTYEAR card for pre-1997 PM10 processing, the keyword 'H6H' must be specified.

353: MULTYEAR Card for PM10 Processing Applies Only for Pre-1997 Applications. Use High-Fourth-High for Post-1997 PM10 Processing.

354: High-Fourth-High Only Required for Post-1997 PM10 Processing with the RECTABLE Card.

360: 2-digit Year Specified. Valid for the range 1950-2049. Four-digit years are valid for the entire range of Gregorian dates, but two digit years are accepted.

363: 24-Hr and ANNUAL Averages Only are Allowed for Post-1997 PM10 Processing on the AVERTIME Card.

365: Year Input is Greater than 2147. A four-digit year greater than 2147 has been input to the model, which will cause an integer overflow.

370: Invalid Date: 2/29 In a Non-leap Year. The year has been identified as a non-leap year, and a date of 2/29 (February 29) has been specified on the DAYRANGE card. Check the year and/or the date specification.

380: This Input Variable is Out-of-Range. The indicated value may be too large or too small. Use the line number to locate the card in question, and check the variable for a possible error.

381: Latitude in the Surface File is Not Valid. The latitude is read from the header record of the surface/scalar meteorological data file. If the value is not within the valid range of 0 to 90 degrees, then this error message is generated.

382: Error Decoding Latitude in the Surface File. The latitude is read from the header record of the surface/scalar meteorological data file. An error occurred trying to decode the latitude indicating a potential problem with the surface file.

385: Averaging period does not equal 1-hour averages for the TOXXFILE option for the AERMOD model. The AERMOD model will generate TOXXFILE outputs for other averaging periods, but the TOXX model component of TOXST currently supports only the 1-hour averages. This is a non-fatal warning message.

390: Averaging period specified on the EVENTPER card for EVENT processing must be less than or equal to 24.

391: Aspect ratio (length/width) of an area source is greater than 10. The area source algorithm in the AERMOD model allows for specifying area sources as elongated rectangles, however, if the aspect ratio exceeds 10 a warning message will be printed out. The user should subdivide the area so that each subarea has an aspect ratio of less than 10.

392: Error Decoding Latitude. The latitude is read from the header record of the surface/scalar meteorological data file. The value of the latitude should be followed by an 'N' or an 'S' for the northern or southern hemispheres. If the value cannot be decoded properly, then this error message is generate.

395: Met. Data Error; Incompatible Version of AERMET. Based on the version date given in the header record of the surface meteorological data file, the meteorological data were generated by an older, incompatible version of AERMET.

396: Met. Data Generated by Older Version of AERMET. Based on the version date given in the header record of the surface meteorological data file, the meteorological data were generated by an older version of AERMET. The data may be compatible with the current version of AERMOD, but should be updated with the current version of AERMET.

Run Time Message Processing (400 - 499)

Extracted from AMS/U.S. EPA User's Guide for the AERMOD Regulatory Model (EPA 2004)

This type of message is generated during the model run. Setup processing has been completed successfully, and the message is generated during the performance of model calculations. Typical messages will tell the information and error during the model run. If a fatal error of this kind is detected during model execution, a fatal error message is written to the message file and any further processing of the data is prohibited. The rest of the meteorological data file will be read and quality assurance checked to identify additional errors. If a warning occurs, data may or may not be processed, depending on the processing requirements specified within the run stream input data.

405: The Value of PHEE Exceeds 1.0. If the value of PHEE, the fraction of the plume material below Hcrit, is greater than 1.0, then this informational message is generated. The value of PHEE is then set to 1.0 for the specified date.

406: Increase NVMAX for complex AREAPOLY source. The number of vertices for an AREAPOLY source exceeds the value of NVMAX, initially set to 20.

410: Flow Vector Out-of-Range. The flow vector must be between 0 and 360 degrees, inclusive. The date of occurrence is provided with the message (in the form of year, month, day, hour as YYMMDDHH)

413: Number of Threshold Violation Events > 9999 for the Specified Averaging Period. This will result in duplicate event names being written to the EVENTFIL file. The duplicate events will have to be segregated before running the model in the EVENT processing mode.

420: Wind Speed Out-of-Range. The wind speed value may be either too large or too small. An error is generated if the speed is less than 0.0, and a warning is generated if the speed is greater than 30.0 m/s. The date of occurrence is provided with the message (in the form of year, month, day, hour as YYMMDDHH).

430: Ambient Temperature Data Out-of-Range. The ambient temperature value may be either too large or too small. A warning is generated if the temperature is less than 220.0 K or greater than 330 K. The date of occurrence is provided with the message (in the form of year, month, day, hour as YYMMDDHH).

432: Surface Friction Velocity Out-of-Range. The surface friction velocity may by too large. A warning is generated if the surface friction velocity is greater than 1.5 m/s. The date of occurrence is provided with the message (in the form of year, month, day, hour as YYMMDDHH).

435: Surface Roughness Length Out-of-Range. A warning is generated if the surface friction velocity is less than 0.001 meters, and the value is set to 0.001 meters to avoid a divide by zero. The date of occurrence is provided with the message (in the form of year, month, day, hour as YYMMDDHH).

438: Convective Velocity Scale Out-of-Range. The convective velocity scale may by too large. A warning is generated if the convective velocity scale is greater than 3.0 m/s. The date of occurrence is provided with the message (in the form of year, month, day, hour as YYMMDDHH).

440: Calm Hour Identified in Meteorology Data File. This message is generated if a calm hour is identified, and provides the date of occurrence (in the form of year, month, day, hour as YYMMDDHH). A calm hour is identified by a reference wind speed of 0.0 m/s in the surface meteorological data file.

450: Error in Meteorology File - Record Out of Sequence. There is an error in the sequence of the hourly meteorological data file. The message also provides the date of occurrence (in the form of year, month, day, hour as YYMMDDHH).

455: Date/Time Mismatch on Hourly Emission File. There is mismatch in the date/time field between the meteorological data file and the hourly emission file. The message also provides the date of the occurrence from the surface/scalar file (in the form of year, month, day, hour as YYMMDDHH).

456: Date/Time Mismatch on Scalar and Profile Data. There is mismatch in the date/time field between the surface/scalar and the profile meteorological data files. The message also provides the date of the occurrence from the surface/scalar file (in the form of year, month, day, hour as YYMMDDHH).

460: Missing Hour Identified in Meteorology Data File. At least one of the meteorological variables is missing or invalid for the hour specified (in the form of year, month, day, hour as YYMMDDHH). If the missing data processing option is not used, then this message will be generated and any further calculations with the data will be aborted. The model will continue to read through the meteorological data file and check the data.

465: Number of Profile Levels Exceeds the Maximum. The profile meteorological data file includes more than the maximum number of levels, specified by the MXPLVL PARAMETER in MODULE MAIN1. The value of MXPLVL is provided with the message.

470: Mixing Height Value is Less Than or Equal to 0.0. This is an informational message that may indicate an error in the meteorological data file. The message includes the hour of occurrence (in the form of year, month, day, hour as YYMMDDHH).

475: The Reference Height is Higher than 100 m. The reference height is read from the surface/scalar meteorological data file. This warning message is generated if the reference height is higher than 100 meters. Since data for the reference height are used in surface layer similarity profiles, the reference height should be within the surface layer (between about 7 and 100 times the surface roughness length).

480: Less Than 1 Year Found for ANNUAL Averages. The input meteorological data file consists of less than a full year of data. The ANNUAL average requires that a full year of data or multiple years be available.

485: Data Remaining After End of Year. The ANNUAL average requires full years of input data (not necessarily calendar years), and there are data remaining after the end of the year. The number of hours remaining is specified.

487: User-specified Start Date on STARTEND card is Earlier Than Start Date of Data File with the ANNUAL average or post-1997 PM10 processing option.

Input/Output Message Processing (500 - 599)

Extracted from AMS/U.S. EPA User's Guide for the AERMOD Regulatory Model (EPA 2004)

This type of message is generated during the model input and output. Typical messages will tell the type of I/O operation (e.g., opening, reading or writing to a file), and the type of file. If a fatal error of this kind is detected in a runstream image, a fatal error message is written to the message file and any attempt to process data is prohibited. If a warning occurs, data may or may not be processed, depending on the processing requirements specified within the run stream input data.

500: Fatal Error Occurs During Opening of the Data File. The file specified can not be opened properly. This may be the runstream file itself, the meteorological data file, or one of the special purpose output files. This may happen when the file called is not in the specified path, or an illegal file name is specified. If no errors are found in the file name specification, then this message may also indicate that there is not enough memory available to run the program, since opening a file causes a buffer to be opened which takes up additional memory in RAM. For the special purpose output files, the hint field includes character string identifying the type of file and the file unit number, e.g., 'PLTFL312'.

510: Fatal Error Occurs During Reading of the File. File is missing, incorrect file type, or illegal data field encountered. Check the indicated file for possible problems. As with error number 500, this message may also indicate that there is not enough memory available to run the program if no other source of the problem can be identified.

520: Fatal Error Occurs During Writing to the File. Similar to message 510, except that it occurs during a write operation.

530: CAUTION! Met. Station ID Mismatch with SURFFILE for SURFDATA, UAIRDATA or SITEDATA. The surface, upper air, or optional on-site station ID numbers specified on the ME pathway do not agree with the values on the first record of the surface meteorological data file. This is a non-fatal warning message, but the input data should be checked carefully to ensure that the correct data file has been used.

540: No RECTABLE/MAXTABLE/DAYTABLE for Averaging Period. No printed output options selected for a particular averaging period. This is a non-fatal warning condition for the AERMOD model.

550: File Unit/Name Conflict for the Output Option. This error indicates that a problem exists with the file name and file unit specification for one of the special purpose output files. The associated keyword is provided as a hint. The same file name may have been used for more than one file unit, or vice versa.

560: User Specified File Unit Less Than or Equal to 25 for OU Keyword. A file unit of less than or equal to 25 has been specified for the indicated special purpose output files. This is a fatal error condition. File units of less than or equal to 25 are reserved for system files. Specify a unit number in the range of 26 to 100.

565: Possible conflict With Dynamically Allocated FUNIT. A file unit specified for the indicated special purpose output files is in the range > 100, and may therefore conflict with file units dynamically allocated for special purpose files by the model. This is typically a non-fatal warning condition.

570: Problem Reading Temporary Event File for Event. The AERMOD model stores high value events in a temporary file that is used to create the input file for EVENT processing, if requested, and also to store the high values for the summary tables at the end of the printed output file. A problem has been encountered reading this file, possibly because the concentration value was too large and overflowed the fixed format field of F14.5.

580: End-of-File Reached Trying to Read a Data File. The AERMOD model has encountered an end-of-file trying the read the indicated file. This may appear when trying to "re-start" a model run with the CO INITFILE card if there is an error with the initialization file. Check the data file for the correct file name.

Glossary

3D View

3D View is a powerful 3D visualization tool that enables you to quickly visualize digital terrain data in conjunction with your project objects.

AERMAP

AMS/EPA Regulatory Model Terrain Pre-processor.

AERMET

Meteorological processor program used for regulatory applications capable of processing upper air data and hourly surface weather observations for use in dispersion models such as AERMOD.

AERMET View

Lakes Environmental Software Meteorological pre-processor for the AERMOD model. Aermet View supports the U.S. EPA AERMET model.

AERMOD

The **A**MS/**E**PA **R**egulatory **Mod**el (AERMOD) was specially designed to support the EPA's regulatory modeling programs. AERMOD is a regulatory steady-state plume modeling system with three separate components: AERMOD (AERMIC Dispersion Model), AERMAP (AERMOD Terrain Preprocessor), and AERMET (AERMOD Meteorological Preprocessor).

AERMOD Batcher

The AERMOD Batcher is a utility that allows you to run your project outside the AERMOD View interface and is designed to let you easily perform multiple modeling runs. Simply specify the input files for the projects you wish to run and AERMOD Batcher will run all your projects. This is ideal for large modeling runs, which is often required for risk assessment projects.

Air Quality

Ambient pollutant concentrations and their temporal and spatial distribution.

Area Source

Area sources are used to model low level or ground level releases with no plume rise (e.g., storage piles, slag dumps, and lagoons).

ASCII

American Standard Code for Information Interchange, a standard set of codes used by computers and communication devices. Sometimes used to refer to files containing only such standard codes, without any application-specific codes such as might be present in a document file from a word processor program.

Background Concentration

Ambient pollutant concentrations due to:

- Natural sources;
- Nearby sources other than the one(s) currently under consideration; and
- Unidentified sources.

Building Downwash

Downwash occurs when the aerodynamic turbulence induced by nearby buildings cause a pollutant emitted from an elevated source to be mixed rapidly toward the ground (downwash), resulting in higher ground-level concentrations. The building downwash option is only applicable to Point and Flare source types.

Calm

For purpose of air quality modeling, calm is used to define the situation when the wind is indeterminate with regard to speed or direction.

CD-144 Format

Card Deck-144 data format available from NCDC for National Weather Service surface observations commonly used for dispersion models. Each record represents an 80-column "card image".

CO Pathway

Control Pathway is a collective term for the group of input runstream images used to specify the overall job control options, such as titles, dispersion options and terrain options.

Complex Terrain

Terrain exceeding the height of the stack being modeled.

Definitions of Seasons

- **Spring:** Periods when vegetation is emerging or partially green. This is a transitional situation that applies for 1-2 months after the last killing frost in spring.
- **Summer:** Periods when vegetation is lush and healthy, typical of mid-summer, but also of other seasons where frost is less common.
- **Autumn:** Periods when freezing conditions are common, deciduous trees are leafless, crops are not yet planted or are already harvested (bare soil exposed), grass surfaces are brown, and no snow is present.
- **Winter:** Periods when surfaces are covered by snow, and when temperatures are sub-freezing.

Dispersion Model

A group of related mathematical algorithms used to estimate (model) the dispersion of pollutants in the atmosphere due to transport by the mean (average) wind and small scale turbulence.

Emission Rate

The rate of emissions of each pollutant. There are two basic methods to determine the emission rate for a source:

- On-site source emission testing, and
- The application of standard emission factors based on AP-42 methods.

EPA/U.S. EPA

United States Environmental Protection Agency.

EVENT Model

A model specifically designed to provide source contribution (culpability) information for specific events of interest, e.g., design values or threshold violations.

Flow Vector

The direction towards which the wind is blowing.

Height Scale

This is the terrain height and location that has the greatest influence on dispersion for an individual receptor.

Input File

The **Input File** is the file that is created according to the keyword/parameter approach to specifying the options and input data for running the U.S. EPA models ISCST3, AERMOD, and ISC-PRIME. The basic structure of the input runstream file is the same for the ISCST3, ISC-PRIME, and AERMOD models, although some options may differ slightly. You do not have to worry about learning all the keyword structure for preparing the input file, since AERMOD View will automatically create the appropriate input file for you.

The runstream file is divided into six functional "pathways". These pathways are identified by a two-character pathway ID placed at the beginning of each runstream image. The pathways and the order in which they are input to the model are as follows:

- **CO** - for specifying overall job COntrol options;
- **SO** - for specifying SOurce information;
- **RE** - for specifying REceptor information;
- **ME** - for specifying MEteorology information;
- **TG** - for specifying Terrain Grid information; and
- **OU** - for specifying OUtput options.

```
****************************************
** ISCST3 Input Produced by:
** ISC-AERMOD View Ver.
** Lakes Environmental Software Inc.
** Date: 5/13/2005
** File: C:\NewISC\Tutorial\tutorial.INP
****************************************
** ISCST3 Control Pathway
****************************************
**
CO STARTING
   TITLEONE C:\ISCView4\Tutorial\tutorial.isc
   MODELOPT DFAULT CONC  RURAL
   AVERTIME 1
   POLLUTID SO2
   TERRHGTS FLAT
   RUNORNOT RUN
CO FINISHED
**
****************************************
** ISCST3 Source Pathway
****************************************
**
SO STARTING
** Source Location **
** Source ID - Type - X Coord. - Y Coord. **
   LOCATION STCK1 POINT 439245.000 5298405.00
** Source Parameters **
   SRCPARAM STCK1 1 60.000 400.000 5.00000 2.
   SRCGROUP ALL
SO FINISHED
**
****************************************
** ISCST3 Receptor Pathway
****************************************
**
RE STARTING
   DISCCART 438200.00 5297300.00
RE FINISHED
**
****************************************
** ISCST3 Meteorology Pathway
****************************************
**
ME STARTING
   INPUTFIL C:\NewISC\Tutorial\tutorial.met
   ANEMHGHT 10 METERS
   SURFDATA 14826 1988 FLINT/BISHOP_ARPT
   UAIRDATA 14826 1988 FLINT/BISHOP_ARPT
ME FINISHED
**
****************************************
** ISCST3 Output Pathway
****************************************
**
OU STARTING
   RECTABLE ALLAVE 1ST
   RECTABLE 1 1ST
OU FINISHED
```

ISC-PRIME

Industrial **S**ource **C**omplex - **P**lume **Ri**se **M**odel **E**nhancement

ISCST3

Industrial **S**ource **C**omplex - **S**hort **T**erm Dispersion Model, Version **3**. The ISCST3 model provides options to model emissions from a wide range of sources that might be present at a typical industrial source complex.

ISCLT3

Industrial **S**ource **C**omplex **L**ong **T**erm model, version **3**.

Julian Day

The number of the day in the year, i.e., Julian Day = 1 for January 1 and 365 (or 366 for leap years) for December 31.

Land Use and Land Cover (LULC)

Land Use and Land Cover digital data provide information on nine major classes of land use such as urban, agricultural, or forest as well as associated map data such as political units and Federal land ownership. LULC digital data is currently available from the Earth Science Information Centers (ESIC) in the following scales:

- 1:250,000 and 1:100,000-scale Land Use and Land Cover and associated maps, and
- 1:250,000 scale Alaska Interim Land Cover.

The 1:250,000-scale mapping format is usually a quadrangle unit of 1º of latitude x 2º of longitude. The 1:100,000 scale mapping format has been established as a 30' x 60' quadrangle, normally a quarter of a 1:250,000 scale quadrangle. Both series are based on the Universal Transverse Mercator (UTM) projection.

Land Use and Land Cover data compilation is based upon the classification system and definitions of Level II Land Use and Land Cover.

ME Pathway

Collective term for the group of input runstream images used to specify the input meteorological data file and other meteorological variables, including the period to process from the meteorological file.

Met Conditions for Stability Categories

See table below for the meteorological conditions typically associated with each stability categories.

Stability Category	Classification	Natural Phenomena	Most Likely Occurs in Time of Day	Most Likely Occurs in Season
A	Extremely Unstable	Strong thermal instability, bright sunlight	Late Morning to Midafternoon	Spring and Summer
B	Moderately Unstable	Transitional period, moderate mixing	Daytime Transition	All Year
C	Slightly Unstable	Transitional periods, Slight mixing	Daytime Transition	All Year
D	Neutral	Strong wind, overcast, day/ night transitions	Daytime/Cloudy, Nighttime/Cloudy, High Wind, Daylight Transition	All Year
E	Slightly Stable	Transitional periods, nighttime moderate winds	Nighttime Transition	All Year
F	Moderately Stable	Clear nighttime skies, very limited vertical mixing, plume fanning and meandering	Nighttime, clear skies, light wind	All Year

Meteorological Data File

Any file containing meteorological data, whether it be mixing heights, surface observations or on-site data.

Mixing Heights

The depth through which atmospheric pollutants are typically mixed by dispersive processes.

Models

Models refer to the U.S. EPA ISCST3, ISC-PRIME, and AERMOD models.

MPRM

Meteorological **P**rocessor for **R**egulatory **M**odels, a program designed for the purpose of processing meteorological data to prepare them for input to the regulatory models, such as ISCST3. Produces a file comparable to the RAMMET pre-processor output, and also capable of producing STAR summaries.

MrSID Images

MrSID (**M**ulti**R**esolution **S**eamless **I**mage **D**atabase) imaging format has the unique function of storing information about every pixel of an image in an image database. This allows each successive resolution of a MrSID file (*.SID) to be created by supplementing the currently displayed image data with only the new data necessary to create the next resolution. This results in better image detail and clarity with each successive resolution.

NCDC

National Climate Data Center (NCDC)

Federal Building

151 Patton Avenue

Asheville, NC 28801-5001

United States of America

Customer Service: (828) 271 4800

e-mail: question@ncdc.noaa.gov

Web Site: lwf.ncdc.noaa.gov/oa/ncdc.html

NWS

National Weather Service

Open Pit Sources

Open Pit sources are used to model emissions from open pits, such as surface coal mines and rock quarries. The OPEN PIT algorithm uses an effective area for modeling pit emissions, based on meteorological conditions, and then utilizes the numerical integration area source algorithm to model the impact of emissions from the effective area sources. The ISCST3 model accepts rectangular pits with an optional rotation angle specified relative to a north-south orientation.

OU Pathway

Collective term for the group of input runstream images used to specify the receptor locations for a particular run.

Partial Input File

A partial input file contains the inputs written to the input file for a specific pathway. You may create a partial input file for any one of the following pathways:

- CO - for specifying overall job COntrol options;
- SO - for specifying SOurce information;
- RE - for specifying REceptor information;
- ME - for specifying MEteorology information;
- TG - for specifying Terrain Grid information; and
- OU - for specifying OUtput options.

See below an example of a SO Pathway Partial Input File.

```
**
****************************************
** ISCST3 Source Pathway
****************************************
**
**
SO STARTING
** Source Location **
** Source ID - Type - X Coord. - Y Coord. **
   LOCATION STCK1 POINT 439245.000 5298405.000 518.000
   LOCATION STCK2 POINT 439118.000 5298262.000 518.000
   LOCATION VOL1 VOLUME 438625.000 5297924.590 0.000
   LOCATION AREA1 AREA 401222.000 4585281.000 0.000
   LOCATION AREA2 AREA 401222.000 4585281.000 0.000
** Source Parameters **
   SRCPARAM STCK1 1 60.000 400.000 5.00000 2.000
   SRCPARAM STCK2 1 60.000 450.000 5.00000 2.000
   SRCPARAM VOL1 2 2.000 64.867 0.470
   SRCPARAM AREA1 1.66E-5 6.000 183.000 107.000 0.000
   SRCPARAM AREA2 -1.3E-5 6.000 183.000 107.000 0.000
   SRCGROUP ALL
SO FINISHED
```

Pasquill Stability Categories

A classification of the dispersive capacity of the atmosphere, originally defined using surface wind speed, solar insolation (daytime) and cloudiness (nighttime). Atmospheric stability is represented by 6 stability categories (A to F).

Pathway

There are six major functional divisions in the input runstream file for each of the models. These are Control, Source, Receptor, Meteorology, Terrain Grid, and Output.

PCRAMMET

The PCRAMMET program is a meteorological preprocessor used for regulatory applications capable of processing twice-daily mixing heights and hourly surface weather observations for use in dispersion models such as ISCST3 and ISC-PRIME.

Percent View

Percent View is a Lakes Environmental Windows program that allows you to generate percentile plots of a given averaging period contained within an ISCST3, ISC-PRIME, or AERMOD Post Processing File (POSTFILE). Percentile values are routinely used to express air quality standards in an international setting.

Point Sources

Point sources are used to model releases from sources like stacks and isolated vents.

POST View

POST View is a Lakes Environmental Windows post-processor specially designed to handle AERMOD plotfiles, gridding them and displaying them for you. POST View is a true Windows MDI (multiple-document interface) program which allows you to have multiple contour plots being generated in multiple windows, at one time.

RAMMET View

RAMMET View is the Lakes Environmental interface for PCRAMMET, the meteorological pre-processor for the ISCST3 and ISC-PRIME models and is part of the AERMOD View Package.

RE Pathway

Collective term for the group of input runstream images used to specify the receptor locations for a particular run.

Receptor

A location at which ambient air quality is measured or estimated.

Regulatory Model

A dispersion model that has been approved for use by the regulatory offices of the EPA, or another appropriate governing body. Specifically, a model that is included in Appendix A of the Guideline on Air Quality Models (Revised), (EPA, 1987b), such as the ISCST3 model.

RiskGen (ISC/AERMOD Risk Generator)

RiskGen (ISC/AERMOD Risk Generator) allows you to setup all the required input files needed for your IRAP-h View and EcoRisk View projects following the requirements of the U.S. EPA - OSW Human Health Risk Assessment Protocol and Screening Level Ecological Risk Assessment Protocol (HHRAP & SLERAP).

SCRAM BBS

Support **C**enter for **R**egulatory **A**ir **M**odels - **B**ulletin **B**oard **S**ystem, an electronic bulletin board system used by the U.S. EPA for disseminating air quality dispersion models, modeling guidance, and related information.

SDTS

SDTS formatted DEMs are a set of files containing the surface elevation information that is required by AERMAP when modeling elevated terrain. This file type is not accepted by AERMAP; however, it can be converted to an AERMAP-accepted DEM file by using the SDTS to DEM Converter utility in AERMOD View.

Simple Terrain

An area where terrain features are all lower in elevation than the top of the stack.

SO Pathway

Collective term for the group of input runstream images used to specify the source input parameters and source group information.

Source Range

The source range is defined by two source ID separated by a dash, e.g., STACK1-STACK10. The model separates the source IDs into three parts: an initial alphabetical part, a numerical part, and then the remainder of the string. For example, STACK2 falls between the range 'STACK1-STACK10'.

Station Identification

An integer or character string used to uniquely identify a station or site as provided in the upper air (TD-5600 and TD-6201), mixing height (TD-9689), and surface weather (CD-144 and TD-3280) data formats available from NCDC. There are no standard station numbers for on-site data or card image/ screening data, and the user may include any integer string.

Surface Weather Observations

A collection of atmospheric data on the state of the atmosphere as observed from the earth's surface. In the U.S., the National Weather Service collects these data on a regular basis at selected locations.

TD-1440 Format

A format available from NCDC for summarizing NWS surface observations in an 80-column format; the CD-144 format is a subset of this format. This format has been superseded by the TD-3280 format.

TD-3280 Format

The current format available from NCDC for summarizing NWS surface weather observations in an elemental structure, i.e., observations of a single atmospheric variable are grouped together for a designated period of time.

TD-5600 Format

A format available from NCDC for reporting NWS upper air sounding data. This format has been superseded by the TD-6201 format.

TD-6201 Format

The current format available from NCDC for reporting NWS upper air data. The file structure is essentially the same as the TD-5600 format except that there is more quality assurance information.

TD-9689 Format

The format available from NCDC for mixing heights estimated from morning upper air temperature and pressure data and hourly surface observations of temperature.

TIFF/GeoTIFF Images

Tag **I**mage **F**ile **F**ormat (TIFF) supports black-and-white, grayscale, pseudocolor and true color images, all of which can be stored in a compressed or uncompressed format.

The following types of TIFF images can be displayed:

- Single band black and white (1 bit)
- Grayscale (4, 8, 16, 24 or 32 bits)
- Pseudocolor (4, 8, 16 bits)
- Multiband images with 8 bits per band. There is no restriction on the number of bands in the image.

The following compression schemes are supported:

- CCITT Group 3 and 4 algorithms (TIFF Compression Scheme 2, 3, 4 and 32771)
- JPEG post-6.0-style DCT algorithms (TIFF Compression Scheme 7)
- NeXT 2-bit encoding scheme (TIFF Compression Scheme 32766)
- Macintosh PackBits algorithm (TIFF Compression Scheme 32773)
- ThunderScan 4-bit run-length encoding (TIFF Compression Scheme 32809)
- SCI's compression scheme for high-resolution color images (TIFF Compression Scheme 34676 and 34677)

The following TIFF variants are supported:

- TIFF 6.0: (TIFF revision 6.0). Enhancements to image definition, RGB colorimetry. Includes JPEG compression.
- GeoTIFF: (TIFF with a Geo header)

UK DTM

United **K**ingdom Ordnance **D**igitized **T**errain **M**ap

UK NTF

United **K**ingdom Ordnance Survey **N**ational **T**ransfer **F**ormat

Unformatted File

A file written without the use of a FORTRAN FORMAT statement, sometimes referred to as a binary file.

Upper Air Data

Meteorological data obtained from balloon- borne instrumentation that provides information on pressure, temperature, humidity, and wind away from the surface of the earth.

USGS

United States Geological Survey (USGS)

Reston, VA

United States of America

Web Site: www.usgs.gov/

Vertical Potential Temperature Gradient

The change of potential temperature with height, used in modeling the plume rise through a stable layer, and indicates the strength of the stable temperature inversion. A positive value means that potential temperature increases with height above ground and indicates a stable atmosphere.

Volume Sources

Volume Sources are used to model releases from a variety of industrial sources, such as building roof monitors, multiple vents, and conveyor belts. Certain types of line sources can be handled by the models using either a string of volume sources, or as an elongated area source. The volume source algorithms are most applicable to line sources with some initial plume depth, such as conveyor belts and rail lines.

Wind Profile Exponent

The value of the exponent used to specify the profile of wind speed with height according to the power law.

WRPLOT View

WRPLOT View is a Lakes Environmental Windows program that generates wind rose statistics and plots for selected meteorological stations for user-specified date and time ranges.

Appendix: File Formats

Blanking File Format

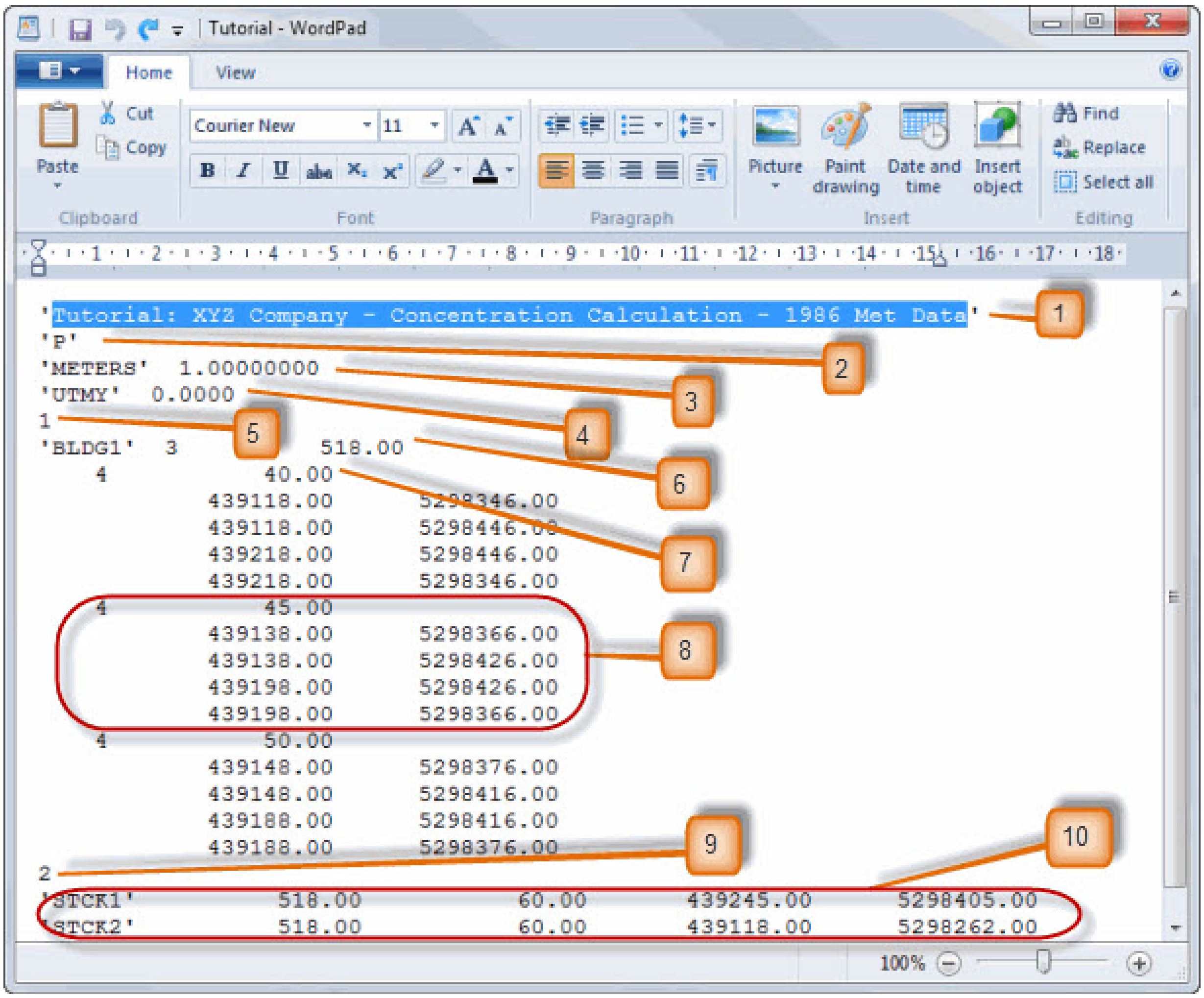
A blanking file must contain the coordinates of all the corners of the polygon that will be used to mask the contour plot lines. The first line in a blanking file must specify the number of X, Y coordinate pairs that follow (one pair per line).

```
4
438898.45   5298601.23
438898.45   5298002.30
439499.87   5298002.30
439499.87   5298601.23
```

In the above example, the first line in the file specifies the number of coordinates in the blanking file. Each subsequent line is an XY location of a point in the blanking file. If you are creating your own file, be sure to save the file with the extension *.rpb.

BPIP Input File Format

The BPIP Input File (*.bpi) contains the input data to BPIP. See below for a description of the input file parameters:



BPIP Input File

1. Contains the project title. This would be the title specified in the Dispersion Options of the Control Pathway, or if not specified then by default the project pathway is displayed. A maximum of 78 characters is permitted on this line. The title must be enclosed by single quotes.

2. A flag between single quotes controls whether BPIP will calculate output for PRIME or not. The flags are:

   - 'ST' - Output will be for an ISCST3 input file.

- 'P' - Output will be for ISC-PRIME and AERMOD.

> The 'P' flag is only found in the BPIPPRM 04274 model and newer. If you try to run an older model such as BPIP 04112, this flag will not be recognized and the project will not run. You may specify that you are using an older model in the EPA Models/Limits - BPIP screen of the Preferences dialog.

3. Identifies the input units by name followed by a units to meters conversion factor. If you input data is in Feet, for example, this input file parameter should be: 'Feet' 0.3048. Up to 10 characters are allowed for the units name. All output will be in meters.

4. Contains the UTM coordinate process flag and a plant north direction value. The flag can only be set to:

- 'UTMN' - for no UTM coordinate processing, or

- 'UTMY' - for UTM coordinate processing

Plant north is the orientation of a plant plot with respect to true north. If a plant plot showed that plant north was toward the southeast, plant north would be 135 degrees.

> AERMOD View will always use UTMY and a plant north direction value of 0.0. If you import a BPIP input file with a plant north different from true north (0 degrees), AERMOD View will automatically rotate the site.

5. Contains the number of buildings to be processed.

6. This line contains the building name, number of tiers for the building, the building's base elevation, and the optional building description. The building name can be up to up to 8 characters in length between single quotes. The number of tiers for this building is an integer and the elevation value is a real number.

7. Contains the number of tier corner coordinates and the tier height with respect to the building base elevation. The number of corners is an integer while the base elevation value is real.

8. Contains a pair of tier corner x-y coordinate values if there is a 'UTMN' flag. If there is a 'UTMY' flag, the coordinate pair are treated as UTM Easting and Northing coordinates, respectively.

9. Contains the integer value of the number of stacks to be processed.

10. Contains the stack data which consists of:

- Stack name - up to 8 characters allowed with no spaces allowed in the name. The name has to be between single quotes.

- Stack base elevation - real value

- Stack height - real value measured from the stack base.

- Stack coordinates - x-y coordinate values if there is a 'UTMN' flag, the coordinate pair are treated as UTM Easting and Northing coordinates, if there is a 'UTMY' flag.

BPIP Primary Output File (*.PRO)

The BPIP Primary Output File (* .pro) contains the essential output data such as the Preliminary GEP stack height values and the BH (Building Height) and PBW (Projected Building Width) input for an ISCST3 input runstream file as well as some additional downwash information to perform enhanced downwash calculations required by the ISC-PRIME and AERMOD models. The primary difference is the inclusion of values for the additional parameters XBADJ and YBADJ in the last portion of the primary output file. AERMOD View will read this file and place all BH, PBW, PBL, XBADJ and YBADJ values for each stack.

```
Tutorial: XYZ Company - Concentration Calculation - 1986 Met Data

                             BPIP (Dated: 04274)              1
DATE :  8/ 8/2005         2
TIME : 12:41:52
Tutorial: XYZ Company - Concentration Calculation - 1986 Met Data          3

============================
BPIP PROCESSING INFORMATION:       4
============================

  The P  flag has been set for preparing downwash related data
         for a model run utilizing the PRIME algorithm.

  Inputs entered in METERS      will be converted to meters using
   a conversion factor of    1.0000.  Output will be in meters.

  The UTMP variable is set to UTMY.  The input is assumed to be in
    UTM coordinates.  BPIP will move the UTM origin to the first pair of
    UTM coordinates read.  The UTM coordinates of the new origin will
    be subtracted from all the other UTM coordinates entered to form
    this new local coordinate system.

  Plant north is set to   0.00 degrees with respect to True North.


Tutorial: XYZ Company - Concentration Calculation - 1986 Met Data          5


               PRELIMINARY* GEP STACK HEIGHT RESULTS TABLE              6
                        (Output Units: meters)

                           Stack-Building           Preliminary*
        Stack    Stack     Base Elevation   GEP**    GEP Stack
        Name     Height    Differences      EQN1     Height Value


       STCK1      60.00        0.00        125.00       125.00
       STCK2      60.00        0.00        125.00       125.00

  * Results are based on Determinants 1 & 2 on pages 1 & 2 of the GEP
    Technical Support Document.  Determinant 3 may be investigated for
    additional stack height credit.  Final values result after
    Determinant 3 has been taken into consideration.
 ** Results were derived from Equation 1 on page 6 of GEP Technical
    Support Document.  Values have been adjusted for any stack-building
    base elevation differences.
```

```
    Note:  Criteria for determining stack heights for modeling emission
    limitations for a source can be found in Table 3.1 of the
    GEP Technical Support Document.

                           BPIP (Dated: 04274)          7
 DATE :  8/ 8/2005
 TIME : 12:41:52


 Tutorial: XYZ Company - Concentration Calculation - 1986 Met Data          8

  BPIP output is in meters          9


    SO BUILDHGT STCK1     40.00   40.00   45.00   45.00   50.00   50.00
    SO BUILDHGT STCK1     50.00   50.00   45.00   50.00   50.00   50.00
    SO BUILDHGT STCK1     45.00   40.00   40.00   40.00   40.00    0.00
    SO BUILDHGT STCK1     40.00   40.00   45.00   45.00   50.00   50.00
    SO BUILDHGT STCK1     50.00   50.00   45.00   50.00   50.00   50.00
    SO BUILDHGT STCK1     45.00   40.00   40.00   40.00   40.00    0.00
    SO BUILDWID STCK1    115.85  128.17   81.96   84.53   56.35   54.64     10
    SO BUILDWID STCK1     51.27   46.34   60.00   46.34   51.27   54.64
    SO BUILDWID STCK1     84.53  140.88  136.60  128.17  115.85    0.00
    SO BUILDWID STCK1    115.85  128.17   81.96   84.53   56.35   54.64
    SO BUILDWID STCK1     51.27   46.34   60.00   46.34   51.27   54.64
    SO BUILDWID STCK1     84.53  140.88  136.60  128.17  115.85    0.00
    SO BUILDLEN STCK1    115.85  128.17   81.96   84.53   56.35   54.64
    SO BUILDLEN STCK1     51.27   46.34   60.00   46.34   51.27   54.64
    SO BUILDLEN STCK1     84.53  140.88  136.60  128.17  115.85    0.00
    SO BUILDLEN STCK1    115.85  128.17   81.96   84.53   56.35   54.64
    SO BUILDLEN STCK1     51.27   46.34   60.00   46.34   51.27   54.64
    SO BUILDLEN STCK1     84.53  140.88  136.60  128.17  115.85    0.00
    SO XBADJ    STCK1    -80.16  -98.88  -87.27  -98.65  -92.95  -98.50
    SO XBADJ    STCK1   -101.07 -100.56 -107.00  -97.44  -94.91  -89.50
    SO XBADJ    STCK1    -95.47 -113.04  -99.01  -81.96  -62.43    0.00
    SO XBADJ    STCK1    -35.69  -29.29    5.31   14.12   36.59   43.86     11
    SO XBADJ    STCK1     49.80   54.22   47.00   51.10   43.64   34.86
    SO XBADJ    STCK1     10.94  -27.84  -37.60  -46.21  -53.42    0.00
    SO YBADJ    STCK1     74.27   69.28   62.18   53.20   42.60   30.71
    SO YBADJ    STCK1     17.88    4.51   -9.00  -22.23  -34.79  -46.29
    SO YBADJ    STCK1    -56.39  -64.77  -71.18  -75.43  -77.39    0.00
    SO YBADJ    STCK1    -74.27  -69.28  -62.18  -53.20  -42.60  -30.71
    SO YBADJ    STCK1    -17.88   -4.51    9.00   22.23   34.79   46.29
    SO YBADJ    STCK1     56.39   64.77   71.18   75.43   77.39    0.00
```

PRO

Output File Parameters

1. Name of the EPA model (BPIP) and date of the model (04274).

2. Date and Time the model was run.

3. Title of your project. This title is the one you have specified in the Dispersion Options screen in the Control Pathway, or if not specified then by default the project pathway is displayed.

4. BPIP Processing Information: This section of the primary output file gives you information on the inputs and options you have selected for the current run. These inputs and options are:

- If output was selected for PRIME (P) or NO PRIME (NP);
- If the inputs were entered in Meters or in other units and the conversion factor to convert the input unit to meters.
- If the input is assumed to be in a local X-Y coordinate system (UTMN) or in a UTM coordinate system (UTMY)
- The rotation, in degrees clockwise, of Plant North from True North (set in the positive Y direction).

5. The Title of your project is repeated.

6. This section of the BPIP Primary Output File contains a table listing the following:

Column 1: Stack Name;

Column 2: Stack Height;

Column 3: Stack-Building Base Elevation Differences;

Column 4: GEP Equation 1 Stack Height Value; and

Column 5: Preliminary GEP Stack Height Values.

7. Name of the EPA model (BPIP) and date of the model (04274). Date and Time the model was run.

8. Title of your project. This title is the one you have specified in the Dispersion Options screen in the Control Pathway, or if not specified then by default the project pathway is displayed.

9. This line explains that the BPIP output results are given in Meters.

10. For each stack, BPIP outputs the values for BHs (Building Heights) and PBWs (Projected Building Widths).

11. BPIP also outputs the values for PBLs (Project Building Lengths), XBADJ (Along-Flow Distance) and YBADJ (Across-Flow Distance).

For each stack you have defined, BPIP outputs the values for BHs (Building Heights), PBWs (Projected Building Widths), XBADJs (Along-Flow Distances) and YBADJs (Across-Flow Distances) in the following way:

- The numeric output begins with sector 1 which is centered on 10 degrees north;
- Each sector is 10 degrees wide;
- Sector numbering proceeds in a clockwise direction and ends with sector 36; and
- BPIP places 6 output values per line for 6 lines for BH, PBW, XBADJ and YBADJ value sets.

BPIP Summary File (*.SUP)

The BPIP Summary File (*.sup) contains detailed output such as which tiers are affecting which stack for a particular wind flow direction. Summary files are generally very large, depending on the number of stacks and buildings being considered.

Examine the file structure below and look at the corresponding numbers on the figures for an explanation of the format of the Summary Output File. The formats are nearly identical with major differences being noted and explained.

```
                             BPIP (Dated: 04274)            1
DATE :  8/15/2005
TIME : 10:49:16           2
Tutorial: XYZ Company - Concentration Calculation - 1986 Met Data

============================
BPIP PROCESSING INFORMATION:          3
============================

  The P  flag has been set for preparing downwash related data
         for a model run utilizing the PRIME algorithm.

  Inputs entered in METERS     will be converted to meters using
   a conversion factor of    1.0000.  Output will be in meters.

  The UTMP variable is set to UTMY.  The input is assumed to be in
    UTM coordinates.  BPIP will move the UTM origin to the first pair of
    UTM coordinates read.  The UTM coordinates of the new origin will
    be subtracted from all the other UTM coordinates entered to form
    this new local coordinate system.

  The new local coordinates will be displayed in parentheses just below
    the UTM coordinates they represent.

  Plant north is set to   0.00 degrees with respect to True North.
```

```
==============
INPUT SUMMARY:
==============


Number of buildings to be processed :    1


BLDG1    has 3 tier(s) with a base elevation of  518.00 Meters
BUILDING  TIER  BLDG-TIER  TIER   NO. OF      CORNER   COORDINATES
  NAME   NUMBER   NUMBER  HEIGHT  CORNERS        X           Y

BLDG1       1         1     40.00     4
                                             565018.00  4191546.00 meters
                                         (        0.00        0.00) meters
                                             565118.00  4191546.00 meters
                                         (      100.00        0.00) meters
                                             565118.00  4191446.00 meters
                                         (      100.00     -100.00) meters
                                             565018.00  4191446.00 meters
                                         (        0.00     -100.00) meters
BLDG1       2         2     45.00     4
                                             565038.00  4191526.00 meters
                                         (       20.00      -20.00) meters
                                             565098.00  4191526.00 meters
                                         (       80.00      -20.00) meters
                                             565098.00  4191466.00 meters
                                         (       80.00      -80.00) meters
                                             565038.00  4191466.00 meters
                                         (       20.00      -80.00) meters
BLDG1       3         3     50.00     4
                                             565048.00  4191516.00 meters
                                         (       30.00      -30.00) meters
                                             565088.00  4191516.00 meters
                                         (       70.00      -30.00) meters
                                             565088.00  4191476.00 meters
                                         (       70.00      -70.00) meters
                                             565048.00  4191476.00 meters
                                         (       30.00      -70.00) meters
```

4

```
Number of stacks to be processed :    2

                     STACK            STACK    COORDINATES
 STACK NAME      BASE  HEIGHT            X            Y

 STCK1         518.00   60.00 Meters
                                    565145.00  4191505.00 meters
                                (       127.00      -41.00) meters
 STCK2         518.00   60.00 Meters
                                    565018.00  4191362.00 meters
                                (         0.00     -184.00) meters

   No stacks have been detected as being atop any structures.
```

5

```
                     Overall GEP Summary Table
                          (Units: meters)


StkNo:   1  Stk Name:STCK1    Stk Ht:  60.00 Prelim. GEP Stk.Ht:  125.00
            GEP:  BH:  50.00  PBW:   50.06             *Eqn1 Ht:  125.00
          *adjusted for a Stack-Building elevation difference of    0.00
 No. of Tiers affecting Stk:  1  Direction occurred:   72.75
  Bldg-Tier nos. contributing to GEP:   3


StkNo:   2  Stk Name:STCK2    Stk Ht:  60.00 Prelim. GEP Stk.Ht:  125.00
            GEP:  BH:  50.00  PBW:   50.06             *Eqn1 Ht:  125.00
          *adjusted for a Stack-Building elevation difference of    0.00
 No. of Tiers affecting Stk:  1  Direction occurred:  197.25
  Bldg-Tier nos. contributing to GEP:   3
```

6

```
                     Summary By Direction Table
                          (Units:  meters)

   Dominate stand alone tiers:


   Drtcn:  10.00

   StkNo:  1  Stk Name:STCK1                              Stack Ht:   60.00
                   GEP:  BH:  50.00  PBW:  50.06   *Equation 1 Ht:  125.00
       Single tier MAX:  BH:  40.00  PBW: 115.85  PBL: 115.85  *Wake Effect Ht:  100.00
       Relative Coordinates of Projected Width Mid-point: XADJ:  -80.16  YADJ:   74.27

            *adjusted for a Stack-Building elevation difference of    0.00
                  BldNo:  1  Bld Name:BLDG1     TierNo:  1
   StkNo:  2  Stk Name:STCK2                              Stack Ht:   60.00
                   GEP:  BH:  50.00  PBW:  50.06   *Equation 1 Ht:  125.00
        No single tier affects this stack for this direction.

   Drtcn:  20.00

   StkNo:  1  Stk Name:STCK1                              Stack Ht:   60.00
                   GEP:  BH:  50.00  PBW:  50.06   *Equation 1 Ht:  125.00
       Single tier MAX:  BH:  40.00  PBW: 128.17  PBL: 128.17  *Wake Effect Ht:  100.00
       Relative Coordinates of Projected Width Mid-point: XADJ:  -98.88  YADJ:   69.28

            *adjusted for a Stack-Building elevation difference of    0.00
                  BldNo:  1  Bld Name:BLDG1     TierNo:  1
   StkNo:  2  Stk Name:STCK2                              Stack Ht:   60.00
                   GEP:  BH:  50.00  PBW:  50.06   *Equation 1 Ht:  125.00
       Single tier MAX:  BH:  40.00  PBW: 128.17  PBL: 128.17  *Wake Effect Ht:  100.00
       Relative Coordinates of Projected Width Mid-point: XADJ:   78.93  YADJ:   -1.15

            *adjusted for a Stack-Building elevation difference of    0.00
                  BldNo:  1  Bld Name:BLDG1     TierNo:  1
```

```
Dominant combined buildings:

Drtcn:  10.00

StkNo:  1  Stk Name:STCK1                              Stack Ht:   60.00
                GEP:  BH:  50.00  PBW:  50.06  *Equation 1 Ht:  125.00
      No combined tiers affect this stack for this direction.
StkNo:  2  Stk Name:STCK2                              Stack Ht:   60.00
                GEP:  BH:  50.00  PBW:  50.06  *Equation 1 Ht:  125.00
      No combined tiers affect this stack for this direction.

Drtcn:  20.00

StkNo:  1  Stk Name:STCK1                              Stack Ht:   60.00
                GEP:  BH:  50.00  PBW:  50.06  *Equation 1 Ht:  125.00
      No combined tiers affect this stack for this direction.
StkNo:  2  Stk Name:STCK2                              Stack Ht:   60.00
                GEP:  BH:  50.00  PBW:  50.06  *Equation 1 Ht:  125.00
      No combined tiers affect this stack for this direction.
```

8

Contour Plot File Format (PLOTFILE)

The contour plot file consists of several header records, each identified by an asterisk (*) in column one, followed by data records. See below the contents of the file:

Header

- Model Name + Model Version + First Title
- List of modeling option keywords applicable to the results
- Averaging Period + High Value + Source Group included in the file
- The total number of receptors included
- The Fortran format used for writing the data records
- Column headers for the variables included in the file

Data Record

The variables provided on each data record include:

- **X:** X Coordinate of the receptor location
- **Y:** Y Coordinate of the receptor location
- **AVERAGE...:** Concentration or deposition value for that location
- **ZELEV:** Receptor terrain elevation
- **AVE:** Averaging period
- **GRP:** Source group ID
- **HIVAL:** The high value included for short-term averages or the number of hours in the period for PERIOD averages.
- **NET ID:** Receptor ID for receptor networks and NA for other receptor types.

Header

```
* ISCST3 (02035): C:\ISCView4\Tutorial\tutorial.isc
* MODELING OPTIONS USED:
*  CONC                        RURAL ELEV         DFAULT
*         PLOT FILE OF  HIGH 1ST  HIGH  3-HR VALUES FOR SOURCE GROUP: ALL
*         FOR A TOTAL OF   441 RECEPTORS.
*         FORMAT: (3(1X,F13.5),1X,F8.2,3X,A5,2X,A8,2X,A4,6X,A8)
*        X             Y      AVERAGE CONC    ZELEV     AVE     GRP       HIVAL     NET ID
*  ____________ ____________ ____________  ______  ______  ________  ________  ________
  438200.00000 5297300.00000      5.06653    0.00    3-HR  ALL       1ST       UCART1
  438300.00000 5297300.00000      4.92133    0.00    3-HR  ALL       1ST       UCART1
  438400.00000 5297300.00000      6.06474    0.00    3-HR  ALL       1ST       UCART1
  438500.00000 5297300.00000      3.08760    0.00    3-HR  ALL       1ST       UCART1
  438600.00000 5297300.00000      3.51729    0.00    3-HR  ALL       1ST       UCART1
  438700.00000 5297300.00000      9.93065    0.00    3-HR  ALL       1ST       UCART1
  438800.00000 5297300.00000      8.51766    0.00    3-HR  ALL       1ST       UCART1
  438900.00000 5297300.00000      8.95620    0.00    3-HR  ALL       1ST       UCART1
  439000.00000 5297300.00000      9.52749    0.00    3-HR  ALL       1ST       UCART1
  439100.00000 5297300.00000      4.61453    0.00    3-HR  ALL       1ST       UCART1
  439200.00000 5297300.00000      7.78353    0.00    3-HR  ALL       1ST       UCART1
```

Data Record

The contour plot file will include the selected high value (1st high, 2nd high, etc.) concentration or deposition at each receptor location for the specified averaging period and source group. For example, if a contour plot file is being generated for the high first high 3-hr averaging period for a total of 441 receptors, then the generated file will display the highest concentration or deposition value that was found for each receptor location at the end of the averaging period (for the data period being processed). For this example, a total of 441 data records will be produced (one for each receptor).

Default ASCII File Format

This is the type of output for the **ASCII** *.met file format.

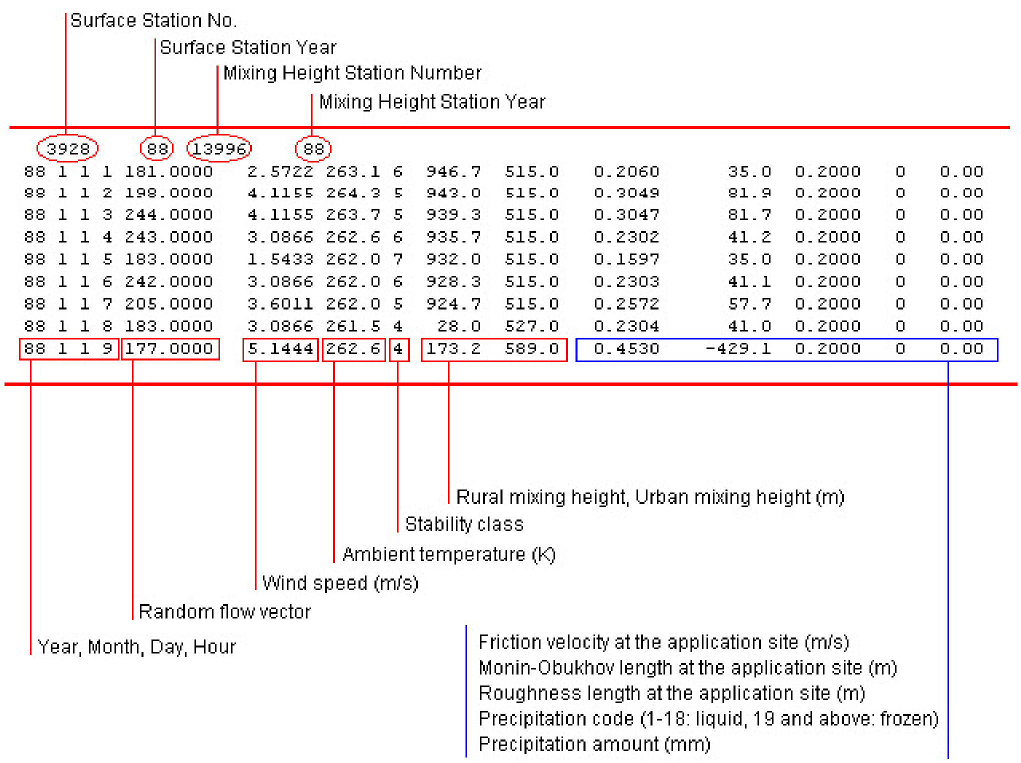
HEADER RECORD

The first record of the ASCII file consists of the following four variables:

Field	Description
001	Surface Station Number
002	Surface Station Year
003	Mixing Height Station Number
004	Mixing Height Station Year

These variables are written with the format:

(4(I6, 1X))



Default ASCII File Format

DATA RECORDS (ONE PER HOUR)

For **CONCENTRATION** estimates, with no deposition, the ASCII file consists of the following variables, one record for each hour of the period.

Field	Description
001	Year (2 digits)
002	Month
003	Day
004	Hour
005	Random flow vector
006	Wind speed (m/s)
007	Ambient temperature (K)
008	Stability category
009	Rural mixing height (m)
010	Urban mixing height (m)

These variables are written with the format:

(4I2, 2F9.4, F6.1, I2, 2F7.1)

For **DRY DEPOSITION** estimates the following three variables are added to the 10 above:

Field	Description
011	Friction velocity at the application site (m/s)
012	Monin-Obukhov length at the application site (m)
013	Roughness length at the application site (m)

The 13 variables are written with the format:

(4I2, 2F9.4, F6.1, I2, 2F7.1, F9.4, F10.1, F8.4)

For **WET DEPOSITION** estimates the following three variables are added to the 13 above:

Field	Description
014	Precipitation code (1-18: liquid, 19 and above: frozen)
015	Precipitation amount (mm)

The 15 variables are written with the format:

(4I2, 2F9.4, F6.1, I2, 2F7.1, F9.4, F10.1, F8.4, I4, F7.2)

Evaluation File Format (EVALFILE)

For each hour of meteorological data processed and for each selected source, the Evaluation file outputs five records containing the following parameters:

Record 1

- Source ID (eight characters)
- Date variable (YYMMDDHH)
- Arc ID (eight characters)
- Arc maximum X/Q
- Emission rate for arc maximum (including unit conversions)
- Crosswind integrated concentration based on true centerline concentration
- Normalized non-dimensional crosswind integrated concentration

Record 2

- Downwind distance corresponding to arc maximum (m)
- Effective wind speed corresponding to arc maximum (m/s)
- Effective sv corresponding to arc maximum (m/s)
- Effective sw corresponding to arc maximum (m/s)
- sy corresponding to arc maximum (m)
- Effective plume height corresponding to arc maximum (m)

Record 3

- Monin-Obukhov length for current hour (m)
- Mixing height for current hour (m)
- Surface friction velocity for current hour (m/s)

- Convective velocity scale for current hour if unstable (m/s), or sz for current hour if stable (m)
- Buoyancy flux for current hour (m**4/s**3)
- Momentum flux for current hour (m**4/s**2)

Record 4

- Bowen ratio for current hour
- Plume penetration factor for current hour
- Centerline X/Q for direct plume
- Centerline X/Q for indirect plume
- Centerline X/Q for penetrated plume
- Nondimensional downwind distance

Record 5

- Plume height/mixing height ratio
- Non-dimensional buoyancy flux
- Source release height (m)
- Arc centerline X/Q
- Developmental option settings place holder (string of 10 zeroes)
- Flow vector for current hour (degrees)
- Effective height for stable plume reflections (m)

```
STCK1     88010101 ARCREC   0.776221E-24  50.0000     0.333701E-22 0.362301E-18         Record 1
              156.3         10.33         1.097       0.7472        17.15        73.81     Record 2
              534.9         1051.         0.5930       8.234        32.36        67.00     Record 3
              2.000         0.000         0.000        0.000        0.000       -999.0     Record 4
           0.7023E-01    -999.0          60.00      0.7762E-24 0000000000     81.00         1051.     Record 5
 STCK1     88010102 ARCREC    0.242445E-23  50.0000     0.105720E-21 0.136210E-17
              156.3         10.97         1.187       0.8095        17.40        72.66
              619.9         1175.         0.6400       8.226        32.51        66.85
              2.000         0.000         0.000        0.000        0.000       -999.0
           0.6184E-01    -999.0          60.00      0.2424E-23 0000000000     78.00         1175.
 STCK1     88010103 ARCREC    0.136589E-22  50.0000     0.607031E-21 0.106824E-16
              156.3         12.25         1.361       0.9299        17.73        70.71
              809.3         1436.         0.7310       8.195        32.64        66.72
              2.000         0.000         0.000        0.000        0.000       -999.0
           0.4924E-01    -999.0          60.00      0.1366E-22 0000000000     74.00         1436.
```

GDEP.DAT File

In order to facilitate review and testing of the deposition algorithms in the AERMOD model, outputs of the main resistance terms and deposition velocities for gaseous sources are made available. The GDEP.DAT file is produced during an AERMOD-DEP run and contains gas deposition data. This file includes the values of Ra, Rb, Rc and Vdepg for each source and for each hour modeled. A header record is found at the top of the file to easily identify each columns. The model overwrites the GDEP.DAT files each time the model is executed.

```
YYMMDDHH   ISRC    Ra            Rb            Rc            Vdepg
96010101      1  0.100000E+04  0.399170E+02  0.118356E+06  0.837552E-05
96010102      1  0.100000E+04  0.399170E+02  0.118356E+06  0.837552E-05
96010103      1  0.100000E+04  0.399170E+02  0.118447E+06  0.836913E-05
96010104      1  0.100000E+04  0.253212E+02  0.117839E+06  0.841295E-05
96010105      1  0.100000E+04  0.309063E+02  0.117794E+06  0.841576E-05
96010106      1  0.100000E+04  0.252815E+02  0.117551E+06  0.843338E-05
96010107      1  0.100000E+04  0.309949E+02  0.115760E+06  0.856228E-05
96010108      1  0.137447E+02  0.216001E+02  0.115609E+06  0.864718E-05
96010109      1  0.130826E+02  0.208893E+02  0.178761E+05  0.558346E-04
96010111      1  0.144490E+02  0.238126E+02  0.171832E+05  0.580670E-04
96010112      1  0.142556E+02  0.237122E+02  0.171497E+05  0.581814E-04
```

AERMOD View renames the GDEP.DAT file to projectname.gdp. This prevents the file from being overwritten if you have more than one project in your folder.

Lakes Format Template

When importing or exporting data into/from your project, the Lakes Format Template can be used. When sources are exported to Excel from your project they are automatically saved in the Lakes format. Templates are found in the folder **C:\Lakes\AERMOD View\Templates**. If you need to import data, you can use these templates.

Source Parameter Template

The **Source Parameter Template** allows you to import parameters for point, volume, area, line volume, and line area sources. Fill in the information in the template, save under a different name, then import it into AERMOD View.

AERMOD View - Source Parameters
MS Excel - Lakes Format: Version 3.0
Supported Source Types: Point, Rectangular Area, Circular Area, Polygon Area, Volume, Open Pit, Line Volume, Line Area

Parameters	Units	Description
Type		POINT, AREA, AREA_CIRC, AREA_POLY, VOLUME, OPEN_PIT, LINE, LINE_VOLUME, LINE_AREA, BUOYLINE
ID		Source ID up to 12 characters
Desc		Optional description
SourceID_Prefix		Text prefix up to 4 characters long for generated LINE_VOLUME and LINE_AREA sources
Base_Elev	[m]	Source base elevation above mean sea level
Height	[m]	Release height above ground
Diam	[m]	Inner stack diameter (POINT) or circular area radius (AREA_CIRC)
Exit_Vel	[m/s]	Exit velocity (POINT only)
Exit_Temp	[K]	Exit temperature (POINT only)
Release Type		VERTICAL, HORIZONTAL, CAPPED (POINT only) - HORIZONTAL and CAPPED are non-default beta options
SigmaY	[m]	Initial sigma Y (VOLUME only)
SigmaZ	[m]	Initial sigma Z (AREA, AREA_CIRC, AREA_POLY, VOLUME, LINE, and LINE_AREA only; optional for AREA, AREA_CIRC, AREA_POLY, and LINE)
Length_X	[m]	X side length (AREA, VOLUME, OPEN PIT, and LINE_AREA only; optional for VOLUME, will be used to calculate SigmaY)
Length_Y	[m]	Y side length (AREA and OPEN PIT only); width for LINE sources
Rotation_Angle	[degrees]	Clockwise rotation from North of Y side (AREA and OPEN PIT only)
Pit_Volume	[m^3]	Volume of the open pit (OPEN PIT only)
Emission_Rate	[g/s or g/s/m2]	Emission rate (g/s for POINT, VOLUME, and LINE_VOLUME; g/s/m2 for AREA, AREA_CIRC, AREA_POLY, OPENPIT, LINE, and LINE_AREA)
Configuration		LINE_VOLUME configuration: Separated, Adjacent or Separated2W
LineVolumeHeight	[m]	Plume Height or Building Height for LINE_VOLUME source
PlumeWidth	[m]	Plume width for LINE_VOLUME source
LineVolumeType		LINE_VOLUME type: None, Surface-Based, Elevated, Elevated Building
LineArea_Ratio1		Ratio 1 for LINE_AREA sources
Line_FPRIMEL =	[m**4/s**3]	Average buoyance parameter (BUOYLINE source only)
Line_L =	[m]	Building Length (BUOYLINE source only)
Line_HB =	[m]	Building Height (BUOYLINE source only)
Line_WB =	[m]	Building Width (BUOYLINE source only)
Line_WM =	[m]	Line Source Width (BUOYLINE source only)
Line_DX =	[m]	Separation between buildings (BUOYLINE source only)
Num_Coords		Number of coordinate pairs (POINT, AREA, AREA_CIRC, VOLUME, OPENPIT = 1; AREA_POLY >= 3; LINE = 2; LINE_AREA, LINE_VOLUME >=2)
X1	[m]	X coordinate of source location [m]
Y1	[m]	Y coordinate of source location [m]
X2	[m]	Secondary X coordinate of source location [m] (AREA_POLY, LINE, LINE_VOLUME, LINE_AREA, BUOYLINE sources only)
Y2	[m]	Secondary Y coordinate of source location [m] (AREA_POLY, LINE, LINE_VOLUME, LINE_AREA, BUOYLINE sources only)
X3	[m]	Additional X coordinate of source location [m] (AREA_POLY, LINE_VOLUME, LINE_AREA only)
Y3	[m]	Additional Y coordinate of source location [m] (AREA_POLY, LINE_VOLUME, LINE_AREA only)
X4	[m]	Additional X coordinate of source location [m] (AREA_POLY, LINE_VOLUME, LINE_AREA only)
Y4	[m]	Additional Y coordinate of source location [m] (AREA_POLY, LINE_VOLUME, LINE_AREA only)
Base_Elev_m	[m]	Base Elevation for LINE_VOLUME, LINE_AREA Nodes
Rel_Height_m	[m]	Release height for LINE_VOLUME, LINE_AREA Nodes

NOTE: you may keep adding additional coordinate pairs for an AREA_POLY or LINE_VOLUME sources, be sure to add the headers as well (eg. X5, Y5, etc)

Type	ID	Desc	SourceID_Prefix	Base_Elev	Height	Diam	Exit_Vel	Exit_Temp	Release_Type	SigmaY	SigmaZ	Length_X	Length_Y	Rotation_Angle
				[m]	[m]	[m]	[m/s]	[K]		[m]	[m]	[m]	[m]	[deg]
POINT	STCK1			1430	30	3	10	350	VERTICAL					
POINT	STCK2			1429	50	5	15	400	VERTICAL					

Source Parameters - Lakes Format Template

Importing Line Volume sources and Line Area Sources

To import line area and line volume sources use the **Source Parameter Template**.

Type	ID	Num_Coords	X1	Y1	X2	Y2	X3	Y3	X4	Y4	X5	Y5
			[m]	[m]	[m]	[m]	[m]	[m]	[m]	[m]		
LINE_VOLUME	SLINE1	5	X - start node	Y - start node	X - node 2	Y - node 2	X - node 3	Y - node 3	X - node 4	Y - node 4	X - node 5	Y - node 5
LINE_VOLUME	SLINE1	**Rel_Height_m**	rel_height - start node	rel_height - node 2	rel_height - node 3	rel_height - node 4	rel_height - node 5					
LINE_VOLUME	SLINE1	**Base_Elev_m**	base_elev - start node	base_elev - node 2	base_elev - node 3	base_elev - node 4	base_elev - node 5					

Line Volume Source - Lakes Format Template (excerpt)

When importing **Line Volume Sources** or **Line Area Sources**, in addition to the emissions data you need to import the coordinates, base elevation, and release height for each line node.

To configure the Excel file containing line volume source data for import, each line volume or line area source must have 3 lines of data:

- **Line 1:** Enter the following necessary data for line volume or line area source:
  - **Volume**
    - **SourceID_Prefix**
    - **Configuration**
    - **LineVolumeHeight**
    - **PlumeWidth**
    - **LineVolumeType**
  - **Area**
    - **SourceID_Prefix**
    - **SigmaZ**
    - **Length_X**
    - **Length_Y**
    - **LineArea_Ratio1**
- **Line 2:** Enter the **Release Height** of each node in order, starting in the **X1** column and continuing on with one **Release Value** per cell.

- **Line 3:** Enter the **Base Elevation** of each node in order, starting in the **X1** column and continuing on with one **Base Elevation** per cell.

> All three lines must have the same **ID**.

Gas and Particle Data Template

The **Gas and Particle Data Template** allows you to import for deposition calculations. Fill in the information in the template, save under a different name, then import it into AERMOD View.

AERMOD View - Gas & Particle Data
MS Excel - Lakes Format: Version 1.0

Parameters	Units	Applicability	Description
Type =			GAS, PARTM1 (Particle Method 1), PARTM2 (Particle Method 2)
Source_ID =			Source ID up to 8 characters
Particle_Diameter	[microns]	PARTM1 and PARTM2	
Mass_Fraction	-	PARTM1 and PARTM2	
Particle_Density	[g/cm^3]	PARTM1	
Diffusivity_Air	[cm^2/s]	GAS	
Diffusivity_Water	[cm^2/s]	GAS	
Cuticular_Resistance	[s/cm]	GAS	
Henry's Law Constant	[Pa-m^3/mol]	GAS	

		PARTM1 and PARTM2	PARTM1 and PARTM2	PARTM1	GAS	GAS	GAS	GAS
Type	**Source_ID**	**Particle_Diameter [microns]**	**Mass Fraction [0 to 1]**	**Particle_Density [g/cm^3]**	**Diffusivity_Air [cm^2/s]**	**Diffusivity_Water [cm^2/s]**	**Cuticular_Resistanc [s/cm]**	**Henry's Law Const [Pa-m^3/mol]**
PARTM1	STCK2	1	0.75	1				
PARTM1	STCK2	2	0.25	1				

Gas & Particle Data - Lakes Format Template

Plant Boundary Template

The **Plant Boundary Template** allows you to import primary plant boundary receptors, including base elevations, flagpole heights, and hill heights. Fill in the information in the template, save under a different name, then import it into AERMOD View.

AERMOD View - Plant Boundary
MS Excel - Lakes Format

Created :
User : Machine :

Parameters	Description	Units	Notes
ID_Boundary=	Name up to 8 characters with no spaces or "-"		Note: The ID must be identical and repeated for each line of a single plant boundary
X=	X coordinate of plant boundary nodes	[m]	
Y=	Y coordinate plant boundary nodes	[m]	
Base_Elevation=	Receptor base elevation above mean sea level	[m]	Note: Optional - If blank, base elevation will be auto-calculated
Flagpole_Height=	Receptor height above ground (Flagpole)	[m]	Note: Optional - If blank, height = 0
Hill_Height=	Receptor hill height	[m]	Note: Optional - If blank, height = 0

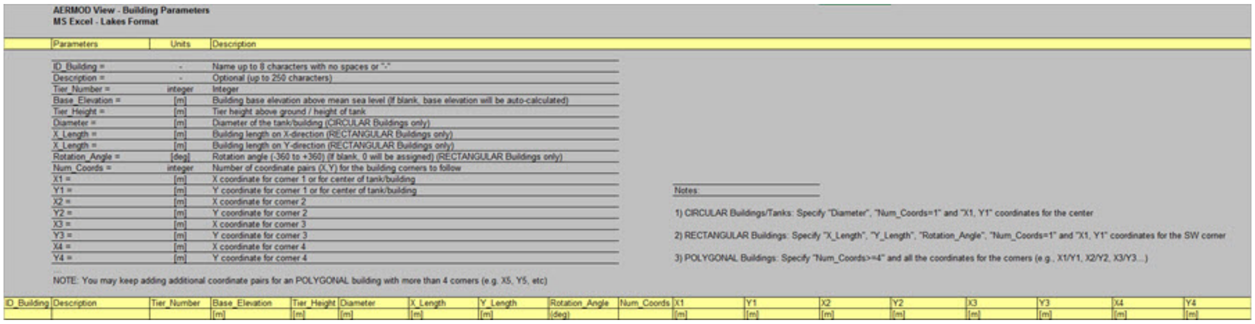
ID_Boundary	X	Y	Base_Elevation	Flagpole_Height	Hill_Height
	[m]	[m]	[m]	[m]	[m]
PLBN1	441466.8	5300887.69	0		
PLBN1	441235.78	5299963.64	0		
PLBN1	442529.46	5299594.02	0		
PLBN1	444700.99	5300102.25	0		
PLBN1	443638.33	5302597.19	0		
PLBN1	442021.23	5301580.74	0		

Plant Boundary - Lakes Format Template

You can import multiple polygons by specifying proper **ID_Boundary** parameter.

Building Parameter Template

The **Building Parameter Template** allows you to import building information, including base elevations, tier heights and rotation angles. Fill in the information in the template, save under a different name, then import it into AERMOD View.



AERMOD View - Building Parameters
MS Excel - Lakes Format

Parameters	Units	Description
ID_Building =	-	Name up to 8 characters with no spaces or "-"
Description =	-	Optional (up to 250 characters)
Tier_Number =	integer	Integer
Base_Elevation =	[m]	Building base elevation above mean sea level (If blank, base elevation will be auto-calculated)
Tier_Height =	[m]	Tier height above ground / height of tank
Diameter =	[m]	Diameter of the tank/building (CIRCULAR Buildings only)
X_Length =	[m]	Building length on X-direction (RECTANGULAR Buildings only)
X_Length =	[m]	Building length on Y-direction (RECTANGULAR Buildings only)
Rotation_Angle =	[deg]	Rotation angle (-360 to +360) (If blank, 0 will be assigned) (RECTANGULAR Buildings only)
Num_Coords =	integer	Number of coordinate pairs (X,Y) for the building corners to follow
X1 =	[m]	X coordinate for corner 1 or for center of tank/building
Y1 =	[m]	Y coordinate for corner 1 or for center of tank/building
X2 =	[m]	X coordinate for corner 2
Y2 =	[m]	Y coordinate for corner 2
X3 =	[m]	X coordinate for corner 3
Y3 =	[m]	Y coordinate for corner 3
X4 =	[m]	X coordinate for corner 4
Y4 =	[m]	Y coordinate for corner 4

...

NOTE: You may keep adding additional coordinate pairs for an POLYGONAL building with more than 4 corners (e.g. X5, Y5, etc)

Notes:

1) CIRCULAR Buildings/Tanks: Specify "Diameter", "Num_Coords=1" and "X1, Y1" coordinates for the center

2) RECTANGULAR Buildings: Specify "X_Length", "Y_Length", "Rotation_Angle", "Num_Coords=1" and "X1, Y1" coordinates for the SW corner

3) POLYGONAL Buildings: Specify "Num_Coords>=4" and all the coordinates for the corners (e.g., X1/Y1, X2/Y2, X3/Y3...)

ID_Building	Description	Tier_Number	Base_Elevation	Tier_Height	Diameter	X_Length	Y_Length	Rotation_Angle	Num_Coords	X1	Y1	X2	Y2	X3	Y3	X4	Y4
			[m]	[m]	[m]	[m]	[m]	(deg)		[m]	[m]	[m]	[m]	[m]	[m]	[m]	[m]

Building Parameters - Lakes Format Template

LEADPOST User-Created Text Files Format

LEADPOST (LEAD Post-Processor) can also read user-created simple text files with variables in the order, units, and type (integer, real, character) shown in table below:

Order	Variable	Unit	Type
1	Receptor x-coordinate	meters	Real
2	Receptor y-coordinate	meters	Real
3	Monthly concentration	ug/m3 (1)	Real
4	Receptor terrain elevation	meters	Real
5	Receptor hill-height scale	meters	Real
6	Receptor height above ground	meters	Real
7	Source Group		Character (max 8)
8	2-digit year		Integer
9	Month		Integer

(1) If concentrations are in units other than micrograms per cubic meter, results from LEADPOST should be converted to micrograms per cubic meter to compare against the lead NAAQS standard.

See LEAD Post-Processor Utility

MPRM file format with CARD option

The order of the meteorological variables for the formatted ASCII files and the default ASCII format are dependent on whether the CARD option is used. See below the format of the MPRM file with the CARD option:

> If the MPRM file format with CARD option is used, then the met file format must be set to **CARD Image** when you are specifying your met data file in the Met Input Data window.

Variable	FORTRAN Format	Columns
Year (last 2 digits)	I2	1-2
Month	I2	3-4
Day	I2	5-6
Hour	I2	7-8
Flow Vector (deg.)	F9.4	9-17
Wind Speed (m/s)	F9.4	18-26
Ambient Temperature (K)	F6.1	27-32
Stability Class (A=1, B=2, … F=6)	I2	33-34
Rural Mixing Height (m)	F7.1	35-41
Urban Mixing Height (m)	F7.1	42-48
Wind Profile Exponent (CARD Only)	F8.4	49-56
Vertical Potential Temperature Gradient (K/m) (CARD Only)	F8.4	57-65
Friction Velocity (m/s) (Dry or Wet Deposition Only)	F9.4	66-74

Variable	FORTRAN Format	Columns
Monin-Obukhov Length (m) (Dry or Wet Deposition Only)	F10.1	75-84
Surface Roughness Length (m) (Dry or Wet Deposition Only)	F8.4	85-92
Incoming Short-wave Radiation (W/m2) (Gas Dry Deposition Only)	F8.1	93-100
Leaf Area Index (Gas Dry Deposition Only)	F8.3	101-108
Precipitation Code (00-45) (Wet Deposition Only)	I4	109-112 (93-96 without Gas Dry Deposition)
Precipitation Rate (mm/hr) (Wet Deposition Only)	F7.2	113-119 (97-103 without Gas Dry Deposition)

MPRM file format without CARD option

The order of the meteorological variables for the formatted ASCII files and the default ASCII format are dependent on whether the CARD option is used. See below the format of the MPRM file without the CARD option:

Variable	FORTRAN Format	Columns
Year (last 2 digits)	I2	1-2
Month	I2	3-4
Day	I2	5-6
Hour	I2	7-8
Flow Vector (deg.)	F9.4	9-17
Wind Speed (m/s)	F9.4	18-26
Ambient Temperature (K)	F6.1	27-32
Stability Class (A=1, B=2, … F=6)	I2	33-34
Rural Mixing Height (m)	F7.1	35-41
Urban Mixing Height (m)	F7.1	42-48
Friction Velocity (m/s) (Dry or Wet Deposition Only)	F9.4	49-57
Monin-Obukhov Length (m) (Dry or Wet Deposition Only)	F10.1	58-67
Surface Roughness Length (m) (Dry or Wet Deposition Only)	F8.4	68-75
Incoming Short-wave Radiation (W/m2) (Gas Dry Deposition Only)	F8.1	76-83
Leaf Area Index (Gas Dry Deposition Only)	F8.3	84-91

Variable	FORTRAN Format	Columns
Precipitation Code (00-45) (Wet Deposition Only)	I4	92-95 (76-79 without Gas Dry Deposition)
Precipitation Rate (mm/hr) (Wet Deposition Only)	F7.2	96-102 (80-86 without Gas Dry Deposition)

Multi-Chemical File Import Format (*.csv)

The **Multi-Chemical Run** utility allows you to easily import your source chemical and emission rate data through a CSV file. You may do this by clicking on the  button in the Chemical Setup tab.

It is essential your file have the following format to be imported successfully:

- The file must be a CSV (comma separated values) file. These are easily created in applications like Microsoft® Excel® spreadsheet software.
- The file must follow the format: Source ID, Chemical Name, Emission Rate.
- Each row must have a unique Source ID and Chemical Name to be imported. Any lines that repeat this information will be ignored.

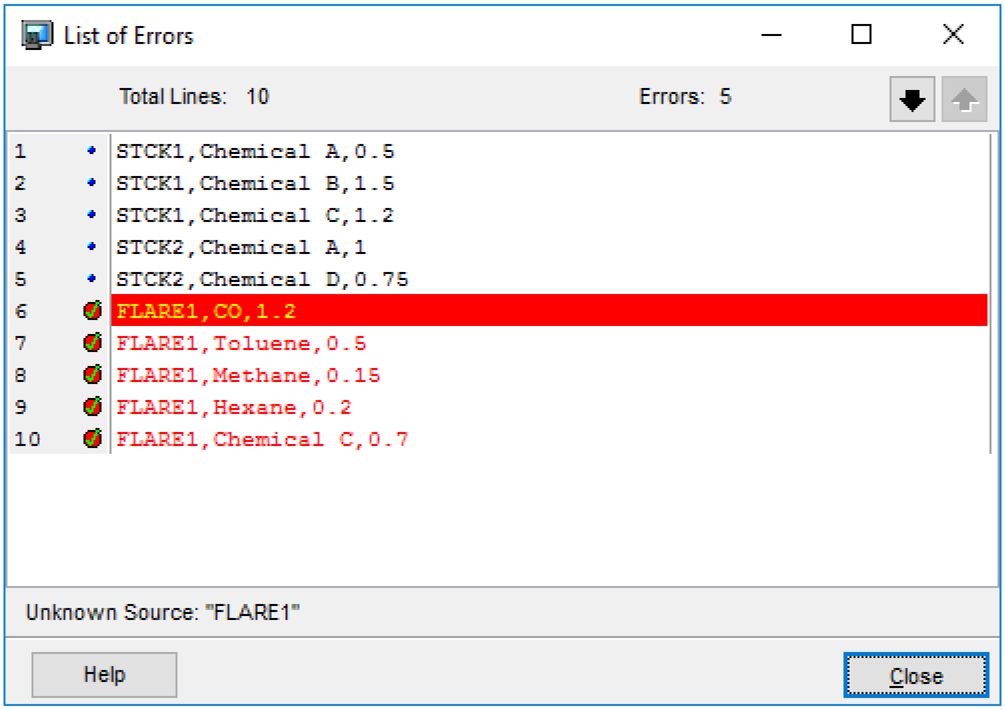
> If any Source IDs that are not defined in the AERMOD View project are found in the *.csv file, the List of Errors dialog will be displayed. The lines containing the unknown Source IDs will be ignored during the import.

Below is a sample portion of a CSV file:

```
Stack1,Benzene,1.2
Stack1,Butane,0.8
Stack1,CO,0.033
Stack1,Diethyl Ether,0.06
Stack1,Hexanol,5
Stack1,Methane,1.2
Stack1,Toluene,1.2
Point2,Benzene,1.2
Point2,Butane,0.03
Point2,CO,0.044
Area1,Benzene,0.8
Area2,Diethyl Ether,0.1
Area3,CO,0.2
Area4,Hexane,1.2
```

List of Errors

The **List of Errors** dialog is displayed during the import of a *.csv file containing source chemical and emission rate data if any Source IDs that are not defined in the AERMOD View project are found in the file.



List of Errors dialog

The unknown Source IDs are displayed in red. These lines will be ignored during the import.

PDEP.DAT File

In order to facilitate review and testing of the deposition algorithms in the AERMOD model, outputs of the main resistance terms and deposition velocities for particle sources are made available. The PDEP.DAT file is produced during an AERMOD-DEP run and contains particle deposition data. This file includes the values of Ra, Rp, Vg and Vd for each source and for each hour modeled. The particle outputs are labeled as being either Method 1 or Method 2. For Method 1, results are output for each size category for the particles. The model overwrites the PDEP.DAT files each time the model is executed.

```
YYMMDDHH    ISRC  ICAT   Method No.   Ra            Rp            Vg(i)         Vdep(i)
96010101       1     -   METHOD_2     0.262839E+02  0.247525E+04  0.000000E+00  0.129086E-02
96010102       1     -   METHOD_2     0.262839E+02  0.247525E+04  0.000000E+00  0.129086E-02
96010103       1     -   METHOD_2     0.262839E+02  0.247525E+04  0.000000E+00  0.129086E-02
96010104       1     -   METHOD_2     0.162346E+02  0.156740E+04  0.000000E+00  0.152261E-02
96010105       1     -   METHOD_2     0.200043E+02  0.191571E+04  0.000000E+00  0.140775E-02
96010106       1     -   METHOD_2     0.162346E+02  0.156740E+04  0.000000E+00  0.152261E-02
96010107       1     -   METHOD_2     0.200030E+02  0.191571E+04  0.000000E+00  0.140775E-02
96010108       1     -   METHOD_2     0.137447E+02  0.133333E+04  0.000000E+00  0.163350E-02
96010109       1     -   METHOD_2     0.130826E+02  0.123592E+04  0.000000E+00  0.169155E-02
96010111       1     -   METHOD_2     0.144490E+02  0.541018E+03  0.000000E+00  0.267811E-02
96010112       1     -   METHOD_2     0.142556E+02  0.467511E+03  0.000000E+00  0.295055E-02
```

AERMOD View renames the PDEP.DAT file to projectname.pdp. This prevents the file from being overwritten if you have more than one project in your folder.

Post-Processing File Format (POSTFILE)

The formatted plot file option (PLOT) of the post-processing file consists of several header records, each identified by an asterisk (*) in column one, followed by data records. See below the contents of the file:

Header

- Model Name + Model Version + First Title
- List of modeling option keywords applicable to the results
- Averaging Period + Source Group included in the file
- The total number of receptors included
- The Fortran format used for writing the data records
- Column headers for the variables included in the file

Data Record

The variables provided on each data record include:

- **X:** X Coordinate of the receptor location
- **Y:** Y Coordinate of the receptor location
- **AVERAGE...:** Concentration or deposition value for that location
- **ZELEV:** Receptor terrain elevation
- **AVE:** Averaging period
- **GRP:** Source group ID
- **DATE or NUM HRS:** Date (YYMMDDHH) for the end of the averaging period for short-term averages or the number of hours in the period for PERIOD averages.
- **NET ID:** Receptor ID for receptor networks and NA for other receptor types.

```
* AERMOD (04300): C:\ISCView4\Tutorial\tutorial.isc
* MODELING OPTIONS USED:
*  CONC                          DFAULT ELEV                                          DRYDPL WETDPL
*         POST/PLOT FILE OF CONCURRENT  3-HR VALUES FOR SOURCE GROUP: ALL
*         FOR A TOTAL OF    441 RECEPTORS.
*         FORMAT: (3(1X,F13.5),3(1X,F8.2),2X,A6,2X,A8,2X,I8.8,2X,A8)
*        X             Y      AVERAGE CONC   ZELEV    ZHILL    ZFLAG    AVE     GRP       DATE     NET ID
* ____________ ____________ _____________ ______ ______ ______ ______ ________ ________ ________
  438538.40625 5297572.50000      0.00013    0.00     0.00     0.00  3-HR  ALL       96010103  UCART1
  438608.18750 5297572.50000      0.00012    0.00     0.00     0.00  3-HR  ALL       96010103  UCART1
  438678.00000 5297572.50000      0.00009    0.00     0.00     0.00  3-HR  ALL       96010103  UCART1
  438747.78125 5297572.50000      0.00005    0.00     0.00     0.00  3-HR  ALL       96010103  UCART1
  438817.56250 5297572.50000      0.00002    0.00     0.00     0.00  3-HR  ALL       96010103  UCART1
  438887.34375 5297572.50000      0.00001    0.00     0.00     0.00  3-HR  ALL       96010103  UCART1
  438957.15625 5297572.50000      0.00000    0.00     0.00     0.00  3-HR  ALL       96010103  UCART1
  439026.93750 5297572.50000      0.00000    0.00     0.00     0.00  3-HR  ALL       96010103  UCART1
  439096.71875 5297572.50000      0.00000    0.00     0.00     0.00  3-HR  ALL       96010103  UCART1
  439166.53125 5297572.50000      0.00000    0.00     0.00     0.00  3-HR  ALL       96010103  UCART1
```

Header

Data Record

The post-processing file will include the concentration or deposition values at each receptor location at the end of the averaging period for the data period being processed. For example, if a post-processing file is being generated for the 3-hr averaging period from Jan 1 to Jan 31 (31 days) for a total of 441 receptors, then the following calculates the number of data records in the file:

$$\text{\# Data Records} = \frac{(\text{\# of Hrs/Day})}{(\text{\# Avg})} \text{ \# Days (\# of Rec)}$$

$$= \frac{24}{3}(31*441) = \text{109,368 data records}$$

where:

# of Hrs/Day = Number of hours per day (24 hrs)

 # Avg = Number of hours per average period

# of Rec = Number of receptors

 # Days = Number of days being processed

Profile Met Data File Format

The **Profile** met data file contains the observations made at each level of an on-site tower, or the one level observations taken from the hourly surface data and upper air data, one record per hour. The profile met data file consists of one or more records for each hour of data. and is distinguished by its extension (*.PFL). There are no processing headers in the profile and this file does not have a record at the beginning of the file that identifies the sites that were used in the processing. The data is delimited by at least one space between each element and may be read as Fortran free format.

> You can find more information on the profile data file format in Appendix D of the User's Guide for the AMS/EPA Regulatory Model – AERMOD (U.S. EPA, 1992).

The **Profile** file has the following format:

DATA RECORDS (ONE PER HOUR)

The following variables are found in each data record:

Field	Description
001	Year
002	Month
003	Day
004	Hour
005	Measurement Height (m)
006	1, if this is the last (highest) level for this hour, or 0 otherwise
007	Wind Direction at the Current Level (degrees from north)
008	Wind Speed at the Current Level (m/s)
009	Temperature at the Current Level (°C)

Field	Description
010	Standard Deviation of the Wind Direction fluctuations (degrees)
011	Standard Deviation of the Vertical Wind Speed Fluctuations (m/s)

Receptor Import/Export Format

Many of the Receptor dialogs allow you to import from, and export receptor locations and parameters to, a file. The following formats are supported:

- **CSV Format (Comma Separated Value)** - CSV format (*.csv) is the default format used by AERMOD View to export and import receptor data within each receptor dialog.

X, Y, ELEV, HILL, FLAG
0, 0, 0, 5, 10
5, 5, 0, 6, 10
100, 100, 0, 6, 10

In the example above, you can see the required parameters for the file. Exported files will contain headers as follows:

X = x coordinate (required)

Y = y coordinate (required)

ELEV = terrain elevation (optional)

HILL = hill heights (AERMOD only)

FLAG = flagpole heights (optional)

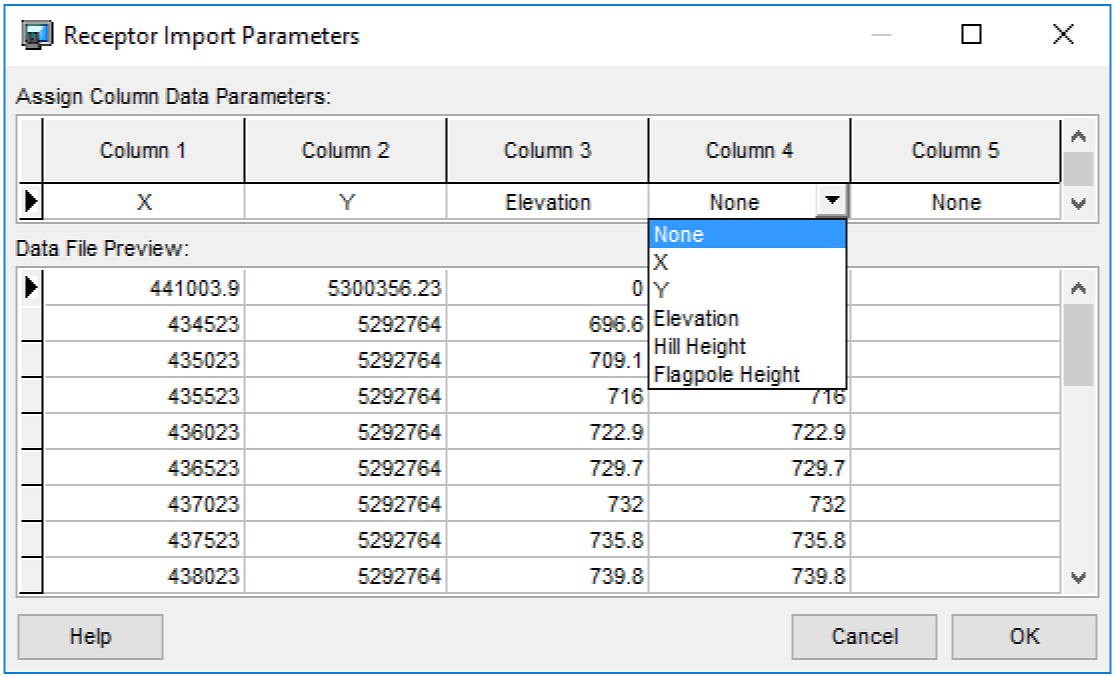
- ***.XYZ format** - AERMOD View can import, but not export, comma or tab delimited text files in *.XYZ format containing the x coordinate, y coordinate, terrain elevations, hill heights and/or flagpole heights. See below a sample xyz file:

```
439300 5297300 496 10
439400 5297300 501 10
439500 5297300 501 10
439600 5297300 501 10
439700 5297300 502 10
```

How to Import Receptor Files:

1. Click the Actions ▼ button in the appropriate receptor dialog. From the drop-down list, choose **Import from File**; the system file browser displays. Select a file and click **Open**.

2. If the file you are importing does not contain the appropriate headers (X, Y, ELEV, HILL and/or FLAG), the **Receptor Import Parameters** dialog displays. On the top of the dialog, you can specify the parameter for each column of data within the receptor file. On the bottom of the dialog, you can see a data preview of the contents of the receptor file.



Receptor Import Parameters dialog

3. For each column header, choose the appropriate parameter from the drop-down list (e.g. under **Column 4**, select **Hill Height**). After all the headers are specified, click **OK**; the new receptors will display in the **Receptor** dialog.

Season by Hour File Format (SEASNHR)

The season by hour-of-day output file consists of several header records, each identified by an asterisk (*) in column one, followed by data records. See below the contents of the file:

Header

- Model Name + Model Version + First Title
- List of modeling option keywords applicable to the results
- Source Group included in the file
- The total number of receptors included
- The Fortran format used for writing the data records
- Column headers for the variables included in the file

Data Record

The variables provided on each data record include:

- **X:** X Coordinate of the receptor location
- **Y:** Y Coordinate of the receptor location
- **AVERAGE...:** Concentration or deposition value for that location
- **ZELEV:** Receptor terrain elevation
- **GRP:** Source group ID
- **NHRS:** Contains the number of non-calm and non-missing hours used to calculate the season-by-hour-of-day averages.
- **SEAS:** Contains the season index, and is 1 for winter, 2 for spring, 3 for summer and 4 for fall.
- **HOUR:** Indicates the hour-of-day (1 to 24).
- **NET ID:** Receptor ID for receptor networks and NA for other receptor types.

The records loop through hour-of-day first and then through the seasons.

```
* ISCST3 (02035): C:\ISCView4\Tutorial\tutorial.isc
* MODELING OPTIONS USED:
*  CONC                     RURAL FLAT        TOXICS
*        FILE OF SEASON/HOUR VALUES FOR SOURCE GROUP: ALL
*        FOR A TOTAL OF   441 RECEPTORS.
*        FORMAT: (2(1X,F13.5),1X,F13.8,F8.2,2X,A8,2X,I4,2X,I4,2X,I4,2X,A8)
*       X             Y      AVERAGE CONC  ZELEV     GRP      NHRS  SEAS  HOUR   NET ID
*  ____________  ____________  ____________  ______  ________  ____  ____  ____  ________
  438538.40600 5297572.50000   0.00000236    0.00  ALL         90     1     1  UCART1
  438608.18800 5297572.50000   0.00000420    0.00  ALL         90     1     1  UCART1
  438678.00000 5297572.50000   0.00000377    0.00  ALL         90     1     1  UCART1
  438747.78100 5297572.50000   0.00000218    0.00  ALL         90     1     1  UCART1
  438817.56300 5297572.50000   0.00000298    0.00  ALL         90     1     1  UCART1
  438887.34400 5297572.50000   0.00000431    0.00  ALL         90     1     1  UCART1
  438957.15600 5297572.50000   0.00000499    0.00  ALL         90     1     1  UCART1
  439026.93800 5297572.50000   0.00000438    0.00  ALL         90     1     1  UCART1
  439096.71900 5297572.50000   0.00000258    0.00  ALL         90     1     1  UCART1
  439166.53100 5297572.50000   0.00000148    0.00  ALL         90     1     1  UCART1
  439236.31300 5297572.50000   0.00000129    0.00  ALL         90     1     1  UCART1
```

Header

Data Record

Space, Tab, or Comma Delimited Format

For the Space, Tab, or Comma Delimited Format, each data record (row) must contain all the parameters for one source. For the Space Delimited Format, each source parameter is separated by one or more spaces. For the Tab Delimited Format, each source parameter is separated by a tab. For the Comma Separated Values (CSV) Format, each source parameter is separated by comma. The order these parameters are written in the file will depend on the type of source. The description and the order these parameters are expected in the file are provided below:

POINT Sources

```
Srcid Srctyp Xs Ys (Zs) Ptemis Stkhgt Stktmp Stkvel Stkdia
```

- **Srcid:** This is the Source ID (up to 8 characters). The Source ID should not contain any spaces or invalid characters (e.g., -)
- **Srctyp:** This is the source type which is identified by the POINT keyword.
- **Xs:** This is the X coordinate of the source location in meters
- **Ys:** This is the Y coordinate of the source location in meters
- **Zs:** This is the source elevation location in meters or feet. This parameter is optional and can be omitted. If this parameter is omitted, then you need to uncheck the **Base Elevation** (Zs) box in the **Sources Import** dialog box.
- **Ptemis:** This is the point emission rate in g/s.
- **Stkhgt:** This is the release height above ground in meters.
- **Stktmp:** This is the stack gas exit temperature in degrees K.
- **Stkvel:** This is the stack gas exit velocity in m/s.
- **Stkdia:** This is the stack inside diameter in meters.

VOLUME Sources

```
Srcid Srctyp Xs Ys (Zs) Vlemis Relhgt Syinit Szinit
```

- **Srcid:** This is the Source ID (up to 8 characters). The Source ID should not contain any spaces or invalid characters (e.g., -)
- **Srctyp:** This is the source type which is identified by the VOLUME keyword.
- **Xs:** This is the X coordinate of the source location in meters

- **Ys:** This is the Y coordinate of the source location in meters
- **Zs:** This is the source elevation location in meters or feet. This parameter is optional and can be omitted. If this parameter is omitted, then you need to uncheck the **Base Elevation** (Zs) box in the **Sources Import** dialog box.
- **Vlemis:** This is the volume emission rate in g/s.
- **Relhgt:** This is the release height (center of volume) above ground in meters.
- **Syinit:** This is the initial lateral dimension in meters.
- **Szinit:** This is the initial vertical dimension in meters.

AREA Sources

```
Srcid Srctyp Xs Ys (Zs) Aremis Relhgt Xinit (Yinit) (Angle) (Szinit)
```

- **Srcid:** This is the Source ID (up to 8 characters). The Source ID should not contain any spaces or invalid characters (e.g., -)
- **Srctyp:** This is the source type which is identified by the AREA keyword.
- **Xs:** This is the X coordinate of the source location in meters
- **Ys:** This is the Y coordinate of the source location in meters
- **Zs:** This is the source elevation location in meters or feet. This parameter is optional and can be omitted. If this parameter is omitted, then you need to uncheck the **Base Elevation** (Zs) box in the **Sources Import** dialog box.
- **Aremis:** This is the area emission rate in g/(s-m^2).
- **Relhgt:** This is the release height above ground in meters.
- **Xinit:** This is the length of X side of the area (in the east-west direction if Angle is o degrees) in meters.
- **Yinit:** This is the length of Y side of the area (in the north-south direction if Angle is o degrees) in meters (optional).
- **Angle:** This is the orientation angle for the rectangular area in degrees from North, measured positive in the clockwise direction (optional).
- **Szinit:** This is the initial vertical dimension of the area source plume in meters (optional).

> The **Yinit** can only be omitted if the **Angle** and the **Szinit** are omitted. The **Angle** can only be omitted if **Szinit** is omitted.

OPENPIT Sources

```
Srcid Srctyp Xs Ys (Zs) Opemis Relhgt Xinit Yinit Pitvol (Angle)
```

- **Srcid:** This is the Source ID (up to 8 characters). The Source ID should not contain any spaces or invalid characters (e.g., -)
- **Srctyp:** This is the source type which is identified by the OPENPIT keyword.
- **Xs:** This is the X coordinate of the source location in meters
- **Ys:** This is the Y coordinate of the source location in meters
- **Zs:** This is the source elevation location in meters or feet. This parameter is optional and can be omitted. If this parameter is omitted, then you need to uncheck the **Base Elevation** (Zs) box in the **Sources Import** dialog box.
- **Opemis:** This is the open pit emission rate in g/(s-m^2).
- **Relhgt:** This is the average release height above the base of the pit in meters.
- **Xinit:** This is the length of X side of the open pit (in the east-west direction if Angle is o degrees) in meters.
- **Yinit:** This is the length of Y side of the open pit (in the north-south direction if Angle is o degrees) in meters.
- **Pitvol:** This is the volume of open pit in cubic meters.
- **Angle:** This is the orientation angle for the rectangular open pit in degrees from North, measured positive in the clockwise direction (optional). If this parameter is omitted, then 0.0 degrees is assumed.

AREAPOLY Sources

```
Srcid Srctyp Xs Ys (Zs) Aremis Relhgt Nverts (Szinit) Xv(1) Yv(1) Xv(2) Yv(2) ... Xv(Nverts) Yv(Nverts)
```

- **Srcid:** This is the Source ID (up to 8 characters). The Source ID should not contain any spaces or invalid characters (e.g., -)

- **Srctyp:** This is the source type which is identified by the AREAPOLY keyword.
- **Xs:** This is the X coordinate of the source location in meters
- **Ys:** This is the Y coordinate of the source location in meters
- **Zs:** This is the source elevation location in meters or feet. This parameter is optional and can be omitted. If this parameter is omitted, then you need to uncheck the **Base Elevation** (Zs) box in the **Sources Import** dialog box.
- **Aremis:** This is the area emission rate in g/(s-m^2).
- **Relhgt:** This is the release height above ground in meters.
- **Nverts:** This is the number of vertices (or sides) of the area source polygon.
- **Szinit:** This is the initial vertical dimension of the area source plume in meters (optional).
- **Xv(Nverts):** This is the x-coordinate of the vertices of the area polygon.
- **Yv(Nverts):** This is the y-coordinate of the vertices of the area polygon

> The first vertex, **Xv(1)** and **Yv(1)**, must also match the coordinates given for the source location Xs and Ys.

AREACIRC Sources

```
Srcid Srctyp Xs Ys (Zs) Aremis Relhgt Radius (Nverts) (Szinit)
```

- **Srcid:** This is the Source ID (up to 8 characters). The Source ID should not contain any spaces or invalid characters (e.g., -)
- **Srctyp:** This is the source type which is identified by the AREACIRC keyword.
- **Xs:** This is the X coordinate of the source location in meters
- **Ys:** This is the Y coordinate of the source location in meters
- **Zs:** This is the source elevation location in meters or feet. This parameter is optional and can be omitted. If this parameter is omitted, then you need to uncheck the **Base Elevation** (Zs) box in the **Sources Import** dialog box.
- **Aremis:** This is the area emission rate in g/(s-m^2).
- **Relhgt:** This is the release height above ground in meters.
- **Radius:** This is the radius of the circular area in meters.

- **Nverts:** This is the number of vertices (or sides) of the area source polygon (optional, 20 sides will be used if omitted).
- **Szinit:** This is the initial vertical dimension of the area source plume in meters (optional).

Nverts can only be omitted if **Szinit** is omitted.

Summary File Parameters

1. Name of the EPA model (BPIP) and date of the model (04274).

2. Date and Time the model was run and the title of your project. This title is the one you have specified in the Dispersion Options screen in the Control Pathway, or if not specified then by default the project pathway is displayed.

3. **BPIP Processing Information:** This section of the BPIP Summary Output File is the same as the one presented for the BPIP Primary Output File. This section gives you information on the inputs and options you have selected for the current run. These inputs and options are:

   - If output was selected for PRIME (P) or NO PRIME (NP);
   - If the inputs were entered in Meters or in other units and the conversion factor to convert the input unit to meters.
   - If the input is assumed to be in a local X-Y coordinate system (UTMN) or in a UTM coordinate system (UTMY)
   - The rotation, in degrees clockwise, of Plant North from True North (set in the positive Y direction).

4. **Input Summary (Buildings):** This section gives a summary of the original input data as read by BPIP. The first line indicates the number of buildings to be processed. This is followed by a summary table for each building which contents is explained below. The heading for each building table gives the name of the building, no. of tiers, and building elevation. This is followed by the following-

   - **Column 1:** Building Name
   - **Column 2:** Tier Number
   - **Column 3:** Building-Tier Number
   - **Column 4:** Tier Height
   - **Column 5:** No. of Corners
   - **Column 6:** Corner X Coordinate
   - **Column 7:** Corner Y Coordinate

5. **Input Summary (Stacks):** After the information for all buildings is printed, the number of stacks to be processed is given. For each stack the following information is given-

   - **Column 1:** Stack Name
   - **Column 2:** Stack Base

- **Column 3:** Stack Height
- **Column 4:** X Coordinate
- **Column 5:** Y Coordinate

For the X and Y coordinates, two lines may follow. One line may contain the conversion of the coordinates from UTM to a local X-Y coordinate system. The other line may contain rotated plant coordinates from plant north to True North. The values printed in both of these lines are between parentheses.

6. **Overall GEP Summary Table:** In this section the following information is given for each stack:

- **Line 1:** Stack No., Stack Name, Stack Height, Preliminary GEP Stack Height.
- **Line 2:** GEP: BH (Building Height), PBW (Projected Building Width), Equation 1 Height.
- **Line 3:** Specifies the adjusted Stack-Building elevation difference that has been subtracted from the GEP Eqn1 values.
- **Line 4:** Specifies the No. of Tiers affecting the Stack, Wind Flow Direction for which the maximum GEP Eqn1 occurs.
- **Line 5:** Specifies the Building-Tier numbers contributing to GEP.

7. **Summary By Direction Table:** The first part of this section gives the dominant stand alone tiers for 36 wind directions beginning with 10 degrees and ending at 360 degrees with increments of 10 degrees. For each stack the following information is given:

- **Line 1:** Stack No., Stack Name, and Stack Height.
- **Line 2:** GEP: BH (Building Height), PBW (Projected Building Width), *Equation 1 Height.
- **Line 3:** Single tier MAX: BH (Building Height), PBW (Projected Building Width), PBL (Projected Building Length), Wake Effect Height. MAX refers to the sector values producing the maximum wake effect height with the minimum amount of width for the stack in question.
- **Line 4:** Specifies the relative coordinates of project width midpoint.
- **Line 5:** Specifies the adjusted Stack-Building elevation difference that has been subtracted from the GEP Eqn1 values.
- **Line 6:** Specifies Building Number, Building Name, and Tier Number affecting this stack. The building-tier numbers can be found in the Input Summary section of the Summary File.

8. **Summary By Direction Table:** The second part of this section gives the dominant combined buildings for 36 wind directions beginning with 10 degrees and ending at 360 degrees with increments of 10 degrees. A different project's summary file containing all information was used for explaining this section. For each stack the following information is given:

- **Line 1:** Stack No., Stack Name, Stack Height.
- **Line 2:** GEP: BH (Building Height), PBW (Projected Building Width), *Equation 1 Height.
- **Line 3:** Combined tier MAX: BH (Building Height), PBW (Projected Building Width), PBL (Projected Building Length), Wake Effect Height. MAX refers to the sector values producing the maximum wake effect height with the minimum amount of width for the stack in question.
- **Line 4:** Specifies the relative coordinates of projected width midpoint.
- **Line 5:** Specifies the adjusted Stack-Building elevation difference that has been subtracted from the GEP Eqn1 values.
- **Line 6:** Specifies the number of tiers affecting the stack in question.
- **Line 7:** Specifies the Building-Tier numbers that are contributing to MAX. The building-tier numbers can be found in the Input Summary section of the Summary File.

Surface Met Data File Format

The **Surface** met data file contains observed and calculated surface variables, one record per hour, of hourly boundary layer parameters estimates. The surface file is distinguished by its extension (*.SFC). There is a single record at the beginning of the file with the boundary layer parameters that identifies the sites that were used in the processing. The data is delimited by at least one space between each element (the data

> You can find more information on the surface data file format in Appendix D of the User's Guide for the AMS/EPA Regulatory Model – AERMOD (U.S. EPA, 1992).

The format of the **Surface** file is as follows:

HEADER RECORD

The first record of the surface file consists of the following six variables:

Field	Description
001	Latitude
002	Longitude
003	UA identifier station identifier for upper air data
004	SF identifier station identifier for hourly surface observations
005	OS identifier on-site identifier
006	Version date AERMET version date; this date appears in the banner on each screen of the summary reports

DATA RECORDS (ONE PER HOUR)

The following variables are found in each data record:

Field	Description
001	Year
002	Month
003	Day
004	Julian Day
005	Hour
006	Sensible Heat Flux (W/m2)
007	Surface Friction Velocity (m/s)
008	Convective Velocity Scale (m/s)
009	Vertical Potential Temperature Gradient above PBL
010	Height of Convectively-Generated Boundary Layer - PBL (m)
011	Height of Mechanically-Generated Boundary Layer - SBL (m)
012	Monin-Obukhov Length (m)
013	Surface Roughness Length (m)
014	Bowen Ratio
015	Albedo
016	Wind Speed (m/s)
017	Wind Direction (degrees)
018	Reference Height for Ws and Wd (m)
019	Temperature (K)
020	Reference Height for Temp (m)

Field	Description
021	Precipitation Code (0-45)
022	Precipitation Rate (mm/hr)
023	Relative Humidity (%)
024	Surface Pressure (mb)
025	Cloud Cover (tenths)

The AERMET meteorological processor (dated 03273) has been modified to output additional meteorological parameters needed for the deposition algorithms in AERMOD. The additional variables include the precipitation code, precipitation rate, relative humidity, surface pressure, and cloud cover.

Threshold Violation File Format (MAXIFILE)

The threshold violation file consists of several header records, each identified by an asterisk (*) in column one, followed by data records. See below the contents of the threshold violation file:

Header

- Model Name + Model Version + First Title
- List of modeling option keywords applicable to the results
- Averaging Period + Threshold + Source Group included in the file
- The Fortran format used for writing the data records
- Column headers for the variables included in the file

Data Record

The variables provided on each data record include:

- **AVE:** Averaging period
- **GRP:** Source group ID
- **DATE:** Date (YYMMDDHH) for the end of the averaging period
- **X:** X Coordinate of the receptor location
- **Y:** Y Coordinate of the receptor location
- **ELEV:** Receptor terrain elevation
- **FLAG:** Flagpole receptor height
- **AVERAGE...:** Concentration or deposition value that violated the threshold

```
* AERMOD (04300): C:\ISCView4\Tutorial\tutorial.isc
* MODELING OPTIONS USED:
*  CONC                          DFAULT ELEV                                                 DRYDPL WETDPL
*         MAXI-FILE FOR  3-HR VALUES >= A THRESHOLD OF    0.5000E-04
*         FOR SOURCE GROUP: ALL
*         FORMAT: (1X,I3,1X,A8,1X,I8.8,2(1X,F13.5),3(1X,F7.2),1X,F13.5)
*AVE   GRP      DATE           X              Y          ZELEV    ZHILL    ZFLAG  AVERAGE CONC
*___ ________ ________ _____________ _____________ _______ _______ _______ _____________
   3 ALL      96010103  438538.40625 5297572.50000    0.00    0.00    0.00       0.00013
   3 ALL      96010103  438608.18750 5297572.50000    0.00    0.00    0.00       0.00012
   3 ALL      96010103  438678.00000 5297572.50000    0.00    0.00    0.00       0.00009
   3 ALL      96010103  438538.40625 5297639.00000    0.00    0.00    0.00       0.00013
   3 ALL      96010103  438608.18750 5297639.00000    0.00    0.00    0.00       0.00014
   3 ALL      96010103  438678.00000 5297639.00000    0.00    0.00    0.00       0.00012
   3 ALL      96010103  438747.78125 5297639.00000    0.00    0.00    0.00       0.00008
   3 ALL      96010103  438538.40625 5297705.50000    0.00    0.00    0.00       0.00012
   3 ALL      96010103  438608.18750 5297705.50000    0.00    0.00    0.00       0.00014
```

Header

Data Record

Unformatted (Binary) File Format

This is the **Unformatted (Binary) File** format:

HEADER RECORD

Data type	Number	Description
Integer	1	Surface Station Number
Integer	1	Surface Station Year
Integer	1	Height Station Number
Integer	1	Mixing Height Station Year

DATA RECORDS (ONE PER DAY)

Data type	Number	Description
Integer	1	Year
Integer	1	Month
Real	1	Julian Day
Integer	24	Hourly Values of Stability Class
Real	24	Hourly Values of Wind Speed (m/s)
Real	24	Hourly Values of Temperature (K)
Real	24	Hourly Values of Flow Vector Values (degrees)
Real	48	Array dimensioned 2,24 containing: 24 rural mixing height values (1), and 24 urban mixing height values (2) (m)

XYZ Terrain Elevations Data File Format

The XYZ terrain elevations data file is one of the file formats used by AERMOD View to import terrain elevations. The XYZ data file should be organized in Column and Row format. The first column must contain the X coordinate of the point, the second column must contain the Y Coordinate of the point, and the third column must contain the terrain elevation (Z) for the specified location. The columns can be separated by one or more blank spaces, a tab or a comma.

The X and Y coordinates must be either in meters or kilometers and the elevation must always be in meters or kilometers. See below a sample XYZ file:

```
439069.88 5298369.18 518
439165.15 5298326.22 502
439217.45 5298298.2 528
439249.21 5298357.97 518
439292.17 5298443.9 501
439266.02 5298475.65 500
439196.91 5298503.67 513
439116.58 5298494.33 518
439084.83 5298458.84 520
439043.73 5298389.73 521
439082.96 5298313.14 519
439068.02 5298288.86 518
```

Features Available with Current Maintenance

The following software **features** are **only available** to users/licenses with **current maintenance**:

- Import of Tile Maps

> The **Lakes Satellite** option is only available to users with **paid** current maintenance.

- Download of NED Terrain Data (NED 1/3 - USA: ~10 m | NED 1- USA, Canada, Mexico: ~30 m) in Terrain Processor
- Download of SRTM1 Terrain Data (Global: ~30 m - Version 3) in Terrain Processor
- Download of CORINE Land Use Data (Europe: 100 m and 250 m) in AERMET View's Land Use Creator
- Download of EOSD Land Use Data (Canada: 25 m) in AERMET View's Land Use Creator

The following software **services** are **only available** to users/licenses with **current maintenance**:

- Access to the Online Knowledgebase
- Software Updates
- Technical Support (support@weblakes.com)

EOSD Land Use Data Files

EOSD - Earth Observation for Sustainable Development of Forests

Resolution: 25 m
Coverage: Canada

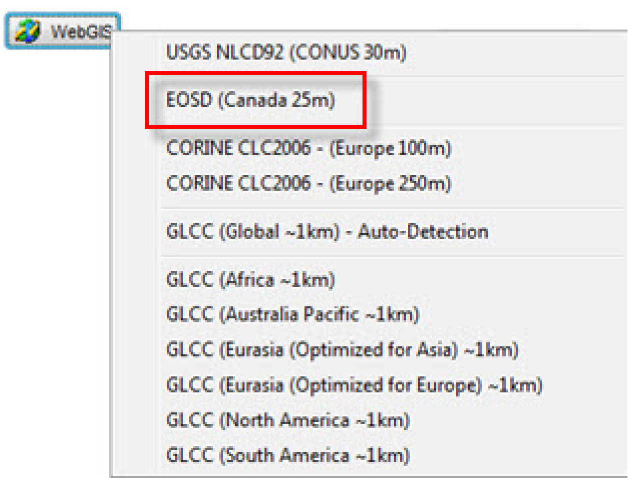
EOSD was developed by the Canadian Forest Service (CFS) in partnership with the Canadian Space Agency and other organizations to use spaced-based earth observation technologies to develop a land cover map of the forested area of Canada.

> The EOSD Land Cover data does not cover portions of Northern Quebec, Southern Ontario, Southern Saskatchewan, Southeastern Alberta or Nunavut.

> The automatic import of **EOSD** land use data feature is ONLY available to users with current maintenance.

How to Download EOSD Data Files

You can download EOSD land use data directly from AERMET View's Land Use Creator window. Press the  button. A sub-menu will be displayed. Choose the EOSD land use data type from the pop-up menu. The land use file corresponding to the modeling area will be automatically downloaded from WebGIS.



You can also download the EOSD land use data files from the CFS web site at https://pfc.cfsnet.nfis.org/mapserver/eosd_portal/htdocs/eosd-cfsnet.phtml.

CORINE Land Use Data Files

CORINE - Coordination of Information on the Environment

Resolution: 100 m or 250 m

Coverage: Europe

The CORINE (Coordination of Information on the Environment) program was initiated in the European Union. It was a prototype project working on many different environmental issues. The CORINE databases and several of its programs have been taken over by the EEA (European Environment Agency). This database is operationally available for most areas of Europe.

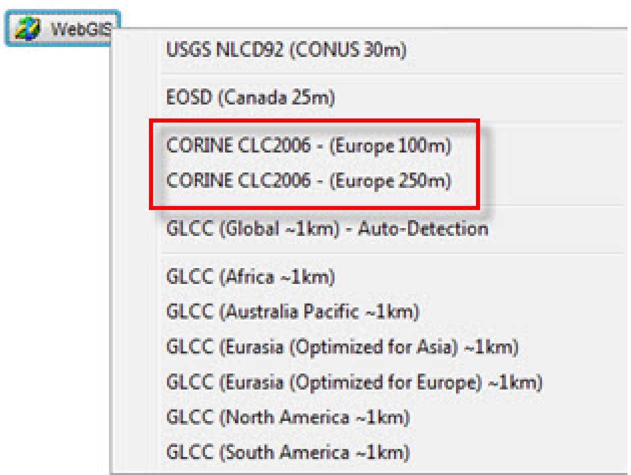
> The automatic import of **CORINE** land use data feature is ONLY available to users with current maintenance.

How to Download CORINE Data Files

You can download CORINE land use data directly from AERMET View's **Land Use Creator** window.

Press the 

button. A sub-menu will be displayed. Choose the CORINE land use data type from the pop-up menu. The land use file corresponding to the modeling area will be automatically downloaded from WebGIS.



You can also download the CORINE land use data files from the EEA web site at http://www.eea.europa.eu/data-and-maps/data/corine-land-cover-2006-raster-3.

Met Data Services

Lakes Environmental provides **worldwide meteorological data** for use in the AERMOD and CALPUFF air dispersion models. Choose from the Weather Research and Forecasting model (WRF) or the NCAR 5th-Generation Mesoscale Model (MM5). These models are used to compute accurate windfields anywhere in the world.

To **request a quote** for these services, please use the **online quotation request form** at https://www.webLakes.com/services/met_order.html.

Lakes employs a wide variety of output formats and processing options. Types of data available through our service include:

- CALMET-Ready from WRF or MM5
- CALPUFF-Ready from WRF
- AERMET-Ready from WRF or MM5
- AERMOD-Ready Station Data (USA only)
- And other products available

Standard WRF data orders are available in 1-km, 4-km, and 12-km resolutions, and MM5 data orders in 4-km and 12-km. Our in-house certified consulting meteorologists can process WRF data based on any custom parameters you require such as:

- Domain size

- Grid resolution
- Physics options
- Output formats

Lakes Environmental Software

Tel.: +1.519.746.5995
Fax: +1.519.746.0793
info@webLakes.com
www.webLakes.com