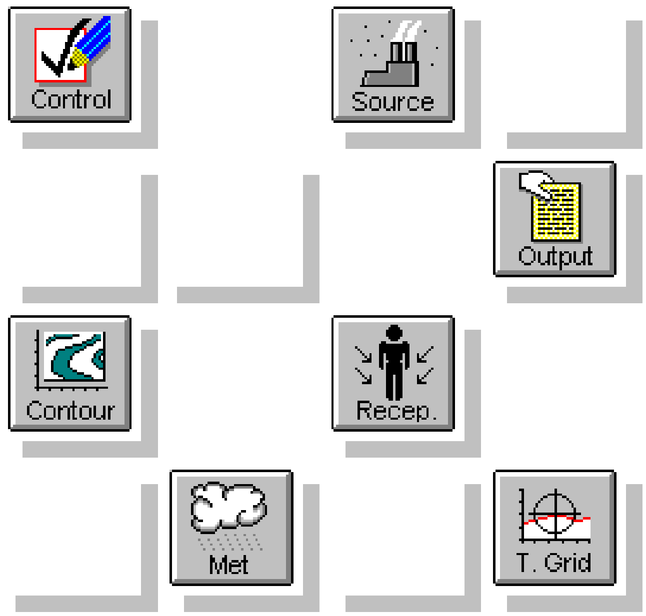
User's Guide



ISC-AERMOD View

Windows Interface for the U.S. EPA ISCST3, AERMOD,
and ISC-PRIME Air Dispersion Models

Lakes Environmental

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

ISC-AERMOD View

Published by
Lakes Environmental Software
450 Phillip Street, Suite 2
Waterloo, Ontario
N2L 5J2 Canada
Tel. : (519) 746-5995
Fax : (519) 746-0793
Web Site : http://www.lakes-environmental.com
e-mail : info@lakes-environmental.com

ISBN 0-9681806-0-4

ISCAER305UG

LAKES ENVIRONMENTAL SOFTWARE LICENSE AGREEMENT

You should carefully read the following license and warranty information. Opening this package indicates your acceptance of these terms and conditions. Lakes Environmental Software, a division of Lakes Environmental Consultants Inc., retains the ownership of this copy of the 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. The software may be moved from one computer to another, as long as there is no possibility of two or more people using it at the same time. 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.

The provided software and documentation contain trade secrets and it is agreed by the licensee that these trade secrets will not be disclosed to non-licensed persons without written consent of Lakes Environmental Software.

WARRANTY AND LIABILITY

Lakes Environmental Software warrants that, under normal use, the material of the magnetic diskettes and the 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 diskettes 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.

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 this software. We have received useful feedback from our users and for that we are truly grateful.

Some of our main contributors, in alphabetical order were:

- ✓ Russell Lee, Independent Consultant, NC
- ✓ Trevor Billington – T. Billington Eng. Consultants Ltd., UK
- ✓ Tim Morgan - Environmental Science Associates (ESA), CA
- ✓ Jeff Secrest – The Air Group, TX

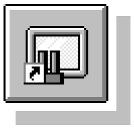
Our development effort was enormously facilitated by our team members:

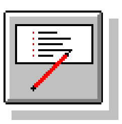
- ✓ Igor Raskin
- ✓ Oleg Shatalov
- ✓ Valeriy Smotrikov
- ✓ Irina Tchoumatcheva
- ✓ Mark Hilverda
- ✓ Sarah Thuring
- ✓ Kevin Kitsemetry
- ✓ Michael W. Davison
- ✓ Garry Seto
- ✓ Jeremy Weber
- ✓ Daniel Raithby
- ✓ Aglaia Obrekht
- ✓ Nina Obrecht
- ✓ Rafaela Thé

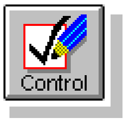
DISCLAIMER

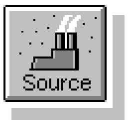
This document and accompanying software follow the U.S. EPA ISCST3, AERMOD, and ISC-PRIME models and documentation to the best of our understanding. The user is responsible for checking the input data and the results for consistency.

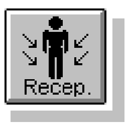
Contents

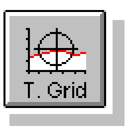


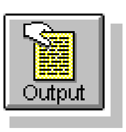


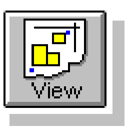

Control

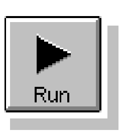

Source


Recep.


T. Grid

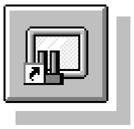

Output


View


Run

CHAPTER 1

Introduction



Welcome to ISC-AERMOD View - a Windows interface for the following U.S. Environmental Protection Agency (EPA) air dispersion modeling codes: ISCST3 (Industrial Source Complex - Short Term Model), AERMOD (AMS/EPA Regulatory Model), and ISC-PRIME (Industrial Source Complex – Plume Rise Model Enhancement). Thank you for choosing ISC-AERMOD View. Lakes Environmental has put in a lot of effort to create the easiest to use and most stable interface on the market.

This chapter gives you a brief description of the ISC-AERMOD View interface and the EPA models. It also shows you how to install ISC-AERMOD View on your computer and introduces the basic components of ISC-AERMOD View to allow a quick start into the interface.

Contents

The U.S. EPA ISCST3 Model

The Industrial Source Complex - Short Term (ISCST3) 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 (EPA), 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 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 AMS/EPA AERMOD Model

The AMS/EPA Regulatory Model (AERMOD) was specially designed to support the EPA's regulatory modeling programs. AERMOD contains basically the same options as the ISCST3 model with a few exceptions which are described below:

- Currently, the model only calculates concentration values. Dry and wet deposition algorithms were not implemented yet at the time this manual was written.
- AERMOD requires two types of meteorological data files, a file containing surface scalar parameters and a file containing vertical profiles. These two files are produced by the U.S. EPA AERMET meteorological preprocessor program.

- 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.

- 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).

- AERMOD does not support the Open Pit type source.

- 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 AERMOD model includes the option to generate an input file for an AERMOD EVENT model, but the AERMOD EVENT model has not yet been developed.

The U.S. EPA ISC-PRIME Model

The Plume Rise Model Enhancements (PRIME) model was designed to incorporate two fundamental features associated with building downwash:

1. Enhanced plume dispersion coefficients due to the turbulent wake

2. 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 ISCST3 model. This integrated model is called ISC-PRIME. The ISC-PRIME or ISC3-PRIME model uses the standard ISCST3 input file with few modifications in the Source Pathway section. These modifications include three new inputs, which are 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 run BPIP-PRIME first, which is available in BPIP View. The BPIP-PRIME building downwash output results must then be imported into the **SO-Building Downwash** window in ISC-AERMOD View.

All the remaining options for the ISC-PRIME model are the same as for the ISCST3 model. However, the enhancements of the ISCST3 model introduced in versions 98348 and 99155 are currently not being supported in ISC-PRIME. These enhancements are listed below:

ISCST3 ENHANCEMENTS INTRODUCED WITH VERSION 98348

- Post-1997 PM10 Processing
- Memory Allocation
- EVENT Processing
- INCLUDED Option
- AREAPOLY Source Type Option
- AREACIRC Source Type Option

ISCST3 ENHANCEMENTS INTRODUCED WITH VERSION 99155

- 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 (June 1999).

The ISC-AERMOD View Interface

ISC-AERMOD View is a user-friendly interface for three U.S. EPA air dispersion models: ISCST3, AERMOD, and ISC-PRIME. This interface was developed specially for Microsoft Windows and runs under Windows 95, Windows 98, and Windows NT.

The ISC-AERMOD View interface uses the five pathways that compose the ISCST3 runstream file as the basics for its functional organization. These pathways are:

1. **Control Pathway (CO):** Where you specify the modeling scenario, and the overall control of the modeling run.

2. **Source Pathway (SO):** Where you define the sources of pollutant emissions.

3. **Receptor Pathway (RE):** Where you define the receptors to determine the air quality impact at specific locations.

4. **Meteorology Pathway (ME):** Where you define the atmospheric conditions of the area being modeled, so it can be taken into account when determining the distribution of air pollution impacts for the area.

5. **Terrain Grid Pathway (TG):** Where you have the option of specifying a gridded terrain data to be used in calculating dry depletion in elevated or complex terrain.

6. **Output Pathway (OU):** Where you define which output results are necessary to meet the needs of the air quality modeling analyses.

In ISC-AERMOD View, each one these pathways is represented by a toolbar button, which will display the first input window for that pathway. From the first window of each pathway, you can have access to the other windows in the same pathway, using the buttons located on the lower right side of each window.

For easy reference, the same icons used in the toolbar buttons will be displayed in each window on the **Pathway Identification** panel. You can follow any order when inputting data in ISC-AERMOD View. Some windows however will be dependent on the inputs and options from other windows.

A good start, if you are not familiar with the inputs and options, is to use the **Next** button, which is located on the lower right side of each ISC-AERMOD View window. This button will guide you from one input window to another. Any time you want to check the missing inputs for your project you can click **Run | Details** from the menu.

ISC-AERMOD View contains all the available options for the three U.S. EPA models (ISCST3, AERMOD, and ISC-PRIME) and uses the original U.S. EPA model executables without any modifications.

Installing the ISC-AERMOD View Package

Before you install ISC-AERMOD View, make sure you have the recommended minimum requirements.

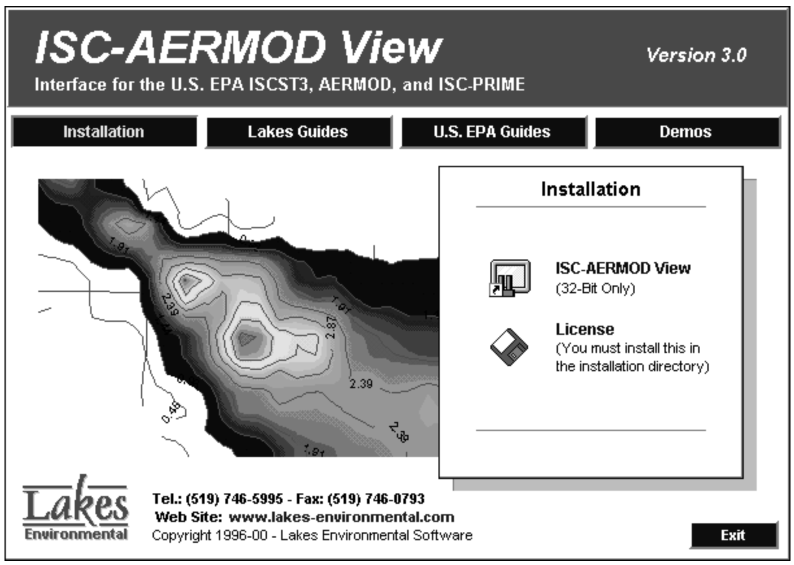
System Requirements

- An IBM or IBM-compatible machine.
- A Pentium processor or higher.
- At least 60 Megabytes of available hard disk space.
- At least 16 Megabytes of memory (RAM) (32 Megabytes recommended)
- Windows 95/98, or Windows NT.
- CD-ROM drive (for installation).

Installation Instructions

Follow the steps below to install ISC-AERMOD View on your computer:

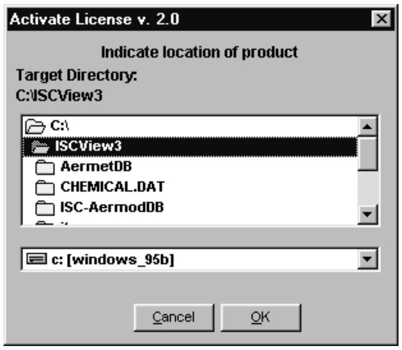
1. Insert the CD into your CD-ROM drive. The installation screen should launch automatically. If the installation screen does not launch automatically, double click on the **My Computer** icon on the Windows Desktop. This will open the **My Computer** window and list your available drives. Next, locate the CD-ROM drive icon and double click on it to show the contents of the CD. Then locate **setup.exe** and double click on it to bring up the installation screen.



Installation Screen

2. To install the ISC-AERMOD View package, click on the **Installation** button. This will display the installation options. Click on the **ISC-AERMOD View** option and follow the installation instructions.

3. After the installing is completed, you can then click on the **License** option. This will activate your user license and validate your newly installed ISC-AERMOD View software. ***ISC-AERMOD View will not operate without activating the License.*** After you click on **License**, you will be prompted to insert the License diskette in Drive A. After placing the License diskette in Drive A, you can press OK and the **Activate License** dialog box will appear.



Indicate the location of your newly installed ISC-AERMOD View software by clicking on the appropriate drive and directory (default location is c:\iscview3). Once the install directory has been selected, click OK and the user license will be activated.

4. The **Installation Instructions** option will display these installation notes in WordPad, Word, or Notepad.

5. The CD-ROM also contains the User's Guide in electronic format. The files can be read using Adobe Acrobat Reader, which can be installed from the CD-ROM if it is not already located on your computer. To install Adobe Acrobat Reader, click on the **Lakes Guides** button and then click on the **Install Acrobat Reader** button, which appears in the Lakes Guides window. To view the User's Guides files click on the **Lakes Guides** button and then select what User's Guide you wish to view.

6. Click on the **U.S. EPA Guides** button to display the U.S. EPA Guides window. From this window, you can preview some of the U.S. EPA guides using Adobe Acrobat. The guides that are not available in PDF file format, can be copied to your hard disk by clicking on the **Copy EPA Guides** option.

7. From the installation screen, you can also view demos for other products by Lakes Environmental Software. To access the demos, click on the **Demos** button and click the desired demo you wish to view.

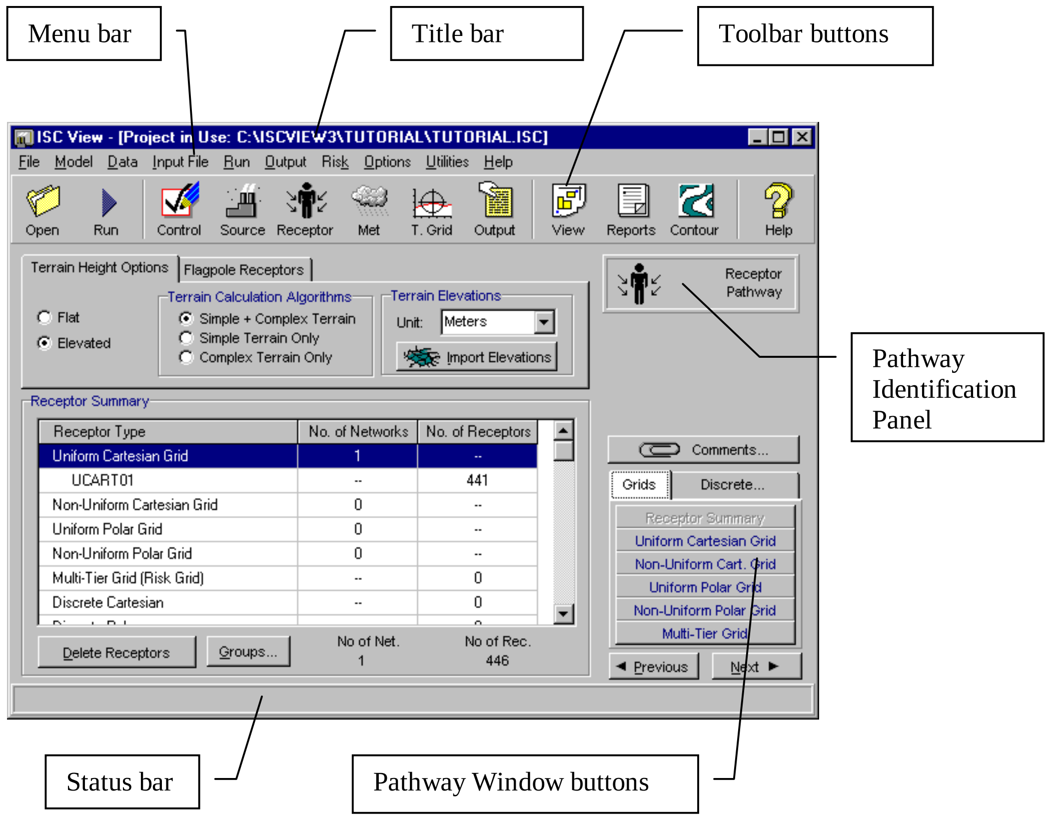
8. Clicking on the **Exit** button will exit the installation screen.

After the installation is complete, a Program Manager group is created containing all icons for the ISC-AERMOD View Package. To start ISC-AERMOD View double-click on the ISC-AERMOD View icon.

The ISC-AERMOD View Window

ISC-AERMOD View's window components follow the standard Window features. For more information on standard Window commands, see your Windows documentation.

The components of the ISC-AERMOD View window are:



❒ **Title bar** .... Displays the interface name, ISC-AERMOD View, and between brackets the full path and name of the project in use.

❒ **Minimize button** .... Minimizes the ISC-AERMOD View window.

❒ **Maximize/Restore button** .... Maximizes the ISC-AERMOD View window, or restores to its pre-maximized size and position.

❒ **Close button** .... This button will close ISC-AERMOD View.

- **Menu bar** .... Displays menu names. To open a menu, move the mouse over the menu name and then press the left mouse button. A menu appears displaying a list of related commands.

- **Toolbar buttons** .... These are a series of buttons that provide a fast method of selecting a menu command.

- **Pathway Identification panel**.. This is the panel that appears on the top right hand side of each window, below the toolbar. The icon and label in this panel identifies in which pathway the user is currently in.

- **Pathway Window buttons** ...... These are a series of buttons that display other windows in the same pathway. In some windows, these buttons are located on tabs. If more than one tab is provided, move the mouse over the tab and then press the left button to have access to the other buttons.

- **Previous button** .... Displays the previous input window.

- **Next button** .... Displays the next input window.

- **Pathway Window** .... This is the area occupied by commands and labels particular to each pathway window. These windows will be described later on in this manual.

- **Status line** .... This area displays a description of the commands in which the mouse pointer is currently on.

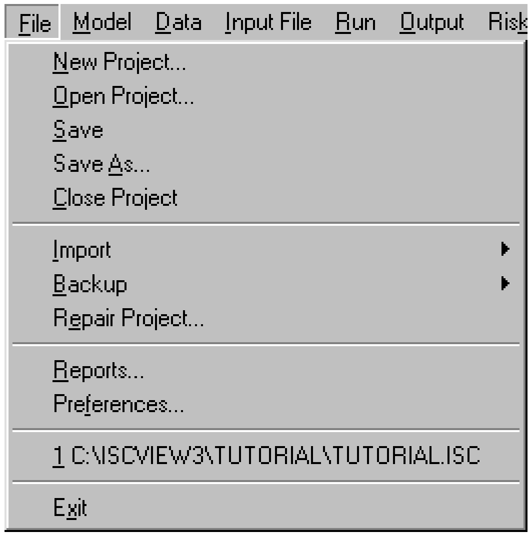
Menu Bar

The following is the description of each menu option:

ISCST3 and ISC-PRIME Menu Options

AERMOD Menu Options

File (Alt, F)



New Project .... Displays the **New Project** dialog box, where you specify the name of the new ISC-AERMOD View project file **(*.ISC).**

Open Project .... Displays the **Open Project** dialog box, where you specify an existing ISC-AERMOD View project file **(*.ISC)** to be opened.

Save .... Saves the current ISC-AERMOD View project file **(*.ISC)**.

Save As... Displays the **Save Project As** dialog box, allowing you to save the current ISC-AERMOD View project with a different name **(*.ISC)**.

Close Project.. Closes the ISC-AERMOD View interface.

Import .... Displays the following submenu options:

- **Terrain Elevations** .... Displays a dialog box where you specify the file(s) from which terrain elevations are to be imported from. The terrain elevation file type can be specified in the **Preferences** dialog box (USGS DEM, UK DTM, UK NTF, and XYZ).

- **ISCST3 Input File**............... Imports an ISCST3 Input File.

- **ISC View 2.0 File** .... Imports an ISC View Project File created with version 2.x.

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).

- ➤ **Extract from ZIP**.................Extracts from the archive file (ZIP file) your project files

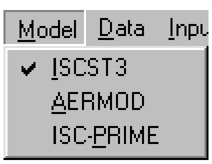
Repair Project ....Displays the **Project Repair** dialog box allowing you to repair database files for a project that may be corrupted.

Reports...Displays the **Reports** dialog box from where you can preview and print reports of all the input options for the current project.

Preferences ....Displays the **Preferences** dialog box, where you specify the U.S. EPA model executable to be used when running your project and the parameters limit for the specified model.

Exit...Closes the ISC-AERMOD View interface.

Model (Alt, M)

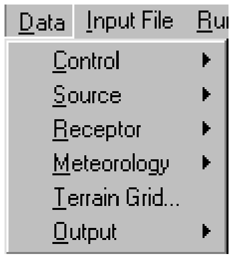


ISCST3..Select this menu option if you want to change to the ISCST3 mode. A check mark indicates which model is being used.

AERMOD..Select this menu option if you want to change to the AERMOD mode. A check mark indicates which model is being used.

ISC-PRIME...Select this menu option if you want to change to the ISC-PRIME mode. A check mark indicates which model is being used.

Data (Alt, D)



Control..Displays submenu options for each Control Pathway window.

Source..Displays submenu options for each Source Pathway window.

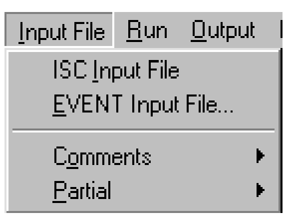
Receptor..Displays submenu options for each Receptor Pathway window.

Meteorology..................................Displays submenu options for each Meteorology Pathway window.

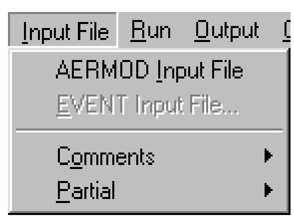
Terrain Grid...................................Displays the Terrain Grid window. This option is available only for ISCST3 and ISC-PRIME when the Elevated Terrain option is being used.

Output..Displays submenu options for each Output Pathway window.

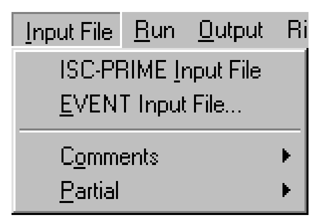
Input File (Alt, I)



ISCST3



AERMOD



ISC-PRIME

Input File.......................................Displays the Input File for the current model using Windows WordPad.

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

Comments......................................Displays submenu options allowing you to specify comments for each pathway section of the input file.

Partial ....Displays submenu options allowing you to save to a separate file the inputs written to the input file for a specific pathway.

Run (Alt, R)

Status ....Displays the **Project Status** dialog box, which displays a summary of all the options selected for the current project.

Details..Displays the **Project Details** dialog box, which lists any missing data for the current project.

Verify Run (Do Not 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 ....Runs the U.S. EPA model for the current mode. This can be the ISCST3 model, the AERMOD model, or the ISC-PRIME model.

Run EVENT ....Runs the EVENT model.

ISC-AERMOD Batcher..........................Launches the ISC-AERMOD Batcher utility, which allows you to run any one of the models outside the ISC-AERMOD View interface. ISC-AERMOD Batcher is ideal for running multiple projects in an attended mode.

Output (Alt, O)

Contour .... Shows contour plots for the current project and current model using POST View. You can only see contour plots if you have run your project. POST View is a Windows post-processor specially designed to handle ISC/AERMOD Plotfiles. POST View will generate a visual representation of the dispersal of concentration/deposition values over time and space.

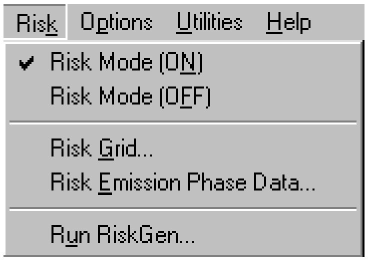
Output File .... Displays the Output File for the current model using Windows WordPad. The output file is created when you run your project.

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.

Error Listing File.................................. Displays the Error Listing File for the current model using Windows WordPad. This file only exists 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 box from where you can specify any text file to view using Windows WordPad.

Risk (Alt, K)



Risk Mode (ON)....................................Select this option if you want change to the Risk mode. The Risk mode 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 if you want to exit Risk mode.

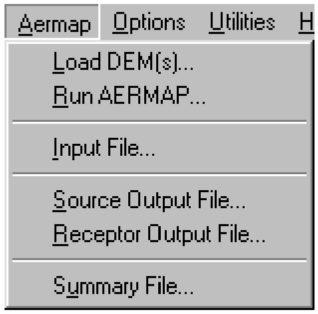
Risk Grid (Multi-Tier Grid)...................Select this option if you want to specify a grid according to the U.S. EPA Human Health and Ecological Risk Assessment Protocols.

Risk Emission Phase Data..................This utility allows you to 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 Protocol (HHRAP & SLERAP). The information specified in this dialog will not be placed in the ISCST3 input file created by ISC-AERMOD View. This information, however, is used by Lakes "ISC Risk Generator", when creating all the necessary files for your risk assessment project.

Run RiskGen..This option displays the ISC Risk Generator (RiskGen) utility. RiskGen sets up all the required ISCST3 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 (Alt, A)

This menu option is only available if you are in the AERMOD mode.



Load DEM(s)..Displays the **DEM Import** dialog box, where you can define the DEM files to be used by AERMAP.

Run ....Runs the U.S. EPA AERMAP model.

Input File ....Displays the input file that was generated to run AERMAP.

Source Output File...............................Displays the output file generated by AERMAP that contains the source data, including source elevations.

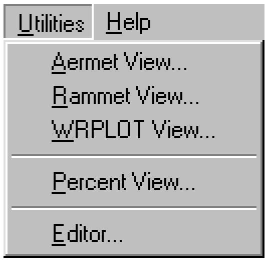
Receptor Output File ....Displays the output file generated by AERMAP that contains the receptor data, including receptor elevations and height scales.

Summary ....Displays the output message file containing an echo of the input file and a listing of any warning or error messages generated by AERMAP.

Options (Alt, O)

Move Site..This option allows you to shift all the coordinates of your sources, receptors, and buildings by a specified amount.

Utilities (Alt, U)



Aermet View .... This utility preprocesses your met data for use with the AERMOD model.

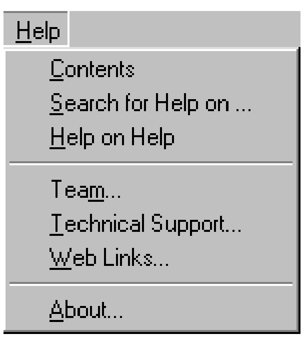
Rammet View .... This utility preprocesses your met data for use with the ISCST3 and ISC-PRIME models.

WRPLOT View .... This utility creates wind roses of your met data.

Percent View .... This utility generates percentile plots of a given averaging period contained within a Post Processing File (POSTFILE), and allows you to perform rolling averages.

Editor .... Opens Windows WordPad.

Help (Alt, H)



Contents .... Displays ISC-AERMOD View Help Contents, from which you can select topics.

Search for Help on .... Lets you search for help on a particular topic.

Help on Help .... Displays information on “How to Use Help”.

Team .... Displays information on the ISC-AERMOD View development team.

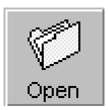
Technical Support ....Displays technical support options for Lakes Environmental software.

Web Links.. Displays Web links for product upgrades, free met data, Lakes Web site, and other related links.

About ....Displays the copyright notice and version number for the ISC-AERMOD View interface.

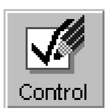
Toolbar Buttons

The toolbar buttons are shortcuts to some of the menu commands. The function of each one of these buttons is explained below:

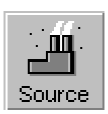


.... Lets you open an existing ISC-AERMOD View project file.

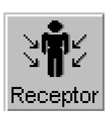
.... Displays the **Project Status** dialog box from where you can check the status of your project and run the model.



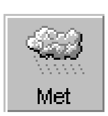
.... Displays the first Control Pathway window – the **CO-Dispersion Options** window.



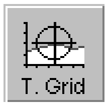
.... Displays the first Source Pathway window – the **SO-Source Inputs** window.



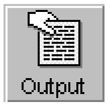
.... Displays the first Receptor Pathway window – the **RE-Receptor Summary** window.



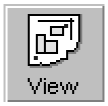
.... Displays the first Meteorology Pathway window – the **ME-Met Input Data** window.



.... Displays the **TG-Terrain Grid** window.

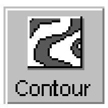


.... Displays the first Output Pathway window – the **OU-Tabular Outputs** window.



.... Displays the **Graphical Input** window from where you can view and graphically define sources and receptors for your modeling project.

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



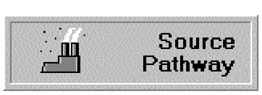
.... Displays the **POST View** window from where you can see contour plots of your results.

.... Displays the Help Contents, from where you can select topics.

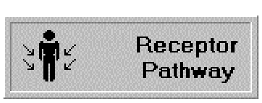
The Pathway Identification Panel

You can find the **Pathway Identification** panel in every ISC-AERMOD View pathway window with the exception of the first Control Pathway window, which is the window that first appears on your screen after you open an ISC-AERMOD View project file. The **Pathway Identification** panel identifies in which pathway you are currently working on. See below the **Pathway Identification** panel for each pathway.

Control (CO) Pathway Identification panel



Source (SO) Pathway Identification panel



Receptor (RE) Pathway Identification panel

Terrain Grid (TG) Pathway Identification panel

Meteorology (ME) Pathway Identification panel

Output (OU) Pathway Identification panel

Using Online Help

ISC-AERMOD View Help contains extensive information on the EPA model requirements, how to use ISC-AERMOD View, and related information collected from EPA's documentation and guidelines. ISC-AERMOD View Help is a very useful tool as you learn both ISC-AERMOD View and the EPA ISCST3, AERMOD, and ISC-PRIME models.

You can have access to ISC-AERMOD View online Help through the following ways:

- **Help Menu:** Two menu commands are available on the Help menu, the **Contents** and the **Search for Help on...**. See the description of these menu commands below.

  1. **Contents:** Select this option if you want to see ISC-AERMOD View Help Contents. From there, you can jump to any topic you want.

  2. **Search for Help on ...:** This option displays the Search dialog box, so you can quickly search for a particular topic. You can also get to the search dialog box by clicking the **Search** button on any Help screen.

- **Help Toolbar Button:** The **Help** button located on the menu toolbar will display the Contents for the window you are currently working on.

- **Context-Sensitive Help (F1):** Many parts of the ISC-AERMOD View interface support a context-sensitive help. This way you can get help without having to go through the Help menu. To get a context-sensitive help on a specific option or topic, the cursor focus should be over the item of interest (e.g., input field, option buttons, etc.).

- **Status Bar Quick Tips:** The status bar, located on the bottom part of every window, gives you quick tips on ISC-AERMOD View options and inputs. As you move the mouse on the screen, the status bar message changes to indicate what a particular command does.

How this User's Guide is Organized

The ISC-AERMOD View User's Guide is organized to help you learn ISC-AERMOD View quickly and on an easy way. The inputs and options for each pathway (Control, Source, Receptor, Meteorology, Terrain Grid, and Output) are described in different chapters (Chapters 3 to 7). For each one of these chapters, all the ISC-AERMOD View input windows are described in separate sections. This way, it will be easy for you to get help from this manual if you have any problems understanding the inputs and options of a specific window.

The ISC-AERMOD View User's Guide is organized into the following chapters:

Chapter 1: **Introduction -** This chapter presents basic information on the U.S. EPA models and ISC-AERMOD View interface.

Chapter 2: **Tutorial -** This chapter presents a step by step example using ISC-AERMOD View.

Chapter 3: **Control Pathway -** This chapter explains the inputs and options on the Control Pathway.

Chapter 4: **Source Pathway -** This chapter explains all the inputs and options on the Source Pathway.

Chapter 5: **Receptor Pathway -** This chapter explains all the inputs and options on the Receptor Pathway.

Chapter 6: **Meteorology Pathway -** This chapter explains all the inputs and options on the Meteorology Pathway.

Chapter 7: **Terrain Grid Pathway -** This chapter explains all the inputs and options on the Terrain Grid Pathway.

Chapter 8: **Output Pathway -** This chapter explains all the inputs and options on the Output Pathway.

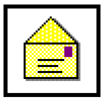
Chapter 9: **Graphical Input -** This chapter explains the options available under the Graphical Input window. The Graphical Input window allows you to graphically input sources and receptors in context with imported site base maps.

Chapter 10: **Running the U.S. EPA Model -** This chapter explains how to get information about the status of your project and how to run the U.S. EPA models (ISCST3, AERMOD, and ISC-PRIME).

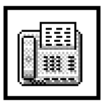
Chapter 11: **POST View -** This chapter explains the options available in POST View.

Getting Technical Support

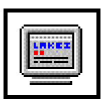
Lakes Environmental is dedicated to providing full technical support for its software. If you need any assistance please contact the Lakes Environmental technical support staff. Our technical support hours are from 9:30 a.m. to 5:00 p.m. EST, Monday through Friday. Please have your serial number and version number ready when calling us.



Lakes Environmental Software Inc.
450 Phillip Street, Suite 2
Waterloo, Ontario
N2L 5J2 Canada



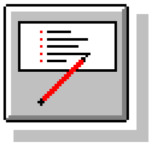
Tel.: (519) 746-5995
Fax: (519) 746-0793



e-mail: support@lakes-environmental.com
Web Site: http://www.lakes-environmental.com

CHAPTER 2

Tutorial



This tutorial will guide you through the basic steps to develop a project using ISC-AERMOD View. This tutorial assumes that you have some familiarity with air dispersion modeling.

Contents

The Problem

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. Parameters for the two sources are described in Table 8A-1 below. Please note that this is only a description of the problem. DO NOT START the tutorial project yet.

Table 2-1. Source Parameters

Parameter	STACK1	STACK2
X Coordinate	439245.00	439118.00
Y Coordinate	5298405.00	5298262.00
Base Elevation (m)	0.0	0.0
Release Height above Ground (m)	60.0	60.0
Emission Rate (g/s)	1.0	1.0
Stack Gas Exit Temperature (K)	400.0	450.0
Stack Gas Exit Velocity (m/s)	5.0	5.0

Parameter	**STACK1**	**STACK2**
Stack Inside Diameter (m)	2.0	2.0

We will model the impact from the chemical plant emissions to the atmosphere to determine if they produce a significant impact for SO_2 and to get an idea of the impacted area. We will perform the air quality impact evaluation using three U.S. EPA models: ISCST3, AERMOD, and ISC-PRIME.

This ISC-AERMOD View tutorial requires files produced by the BPIP View, Aermet View, and Rammet View tutorials. It is advisable that you complete these tutorials before starting the ISC-AERMOD View tutorial. The following files will be used by ISC-AERMOD View:

- ***TUTORIAL.MET*****:** Meteorological data file preprocessed by Rammet View.
- ***TUTORIAL.SFC and TUTORIAL.PFL*****:** Meteorological data files preprocessed by Aermet View.
- ***TUTORIAL.BPO*****:** Building downwash file output by BPIP.
- ***TUTORIAL.PRO*****:** Building downwash file output by BPIP-PRIME.

Creating the Tutorial Project

➤ **How to Create the Tutorial Project:**

Step 1: From the Windows **Start** menu, choose **Programs > Lakes Environmental > ISC-AERMOD View** or double-click on the **ISC-AERMOD View** icon if you have created a shortcut on the desktop.

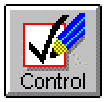
Step 2: The **About** dialog box appears on the screen. Click the **OK** button and the ISC-AERMOD View initial window is displayed.

Step 3: Select **File | New Project...** from the menu. The **New Project** dialog box is displayed.

Step 4: Select the **Directory** and **Drive** and enter the **File Name** for the tutorial project and click the **OK** button.

Note: If you only want to browse this tutorial, we have included the tutorial project file, **TUTORIAL.ISC**, in the installation directory (by default C:\ISCVIEW3\TUTORIAL).

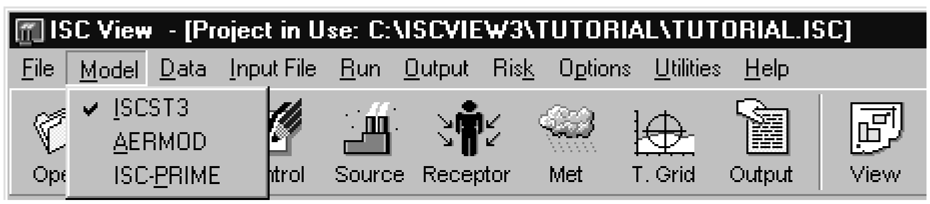
Working on the Control Pathway



After a few seconds the first ISC-AERMOD View window appears, the **CO-Dispersion Options** window. This window is part of the Control Pathway.

After we start inputting data there are some indicators to help you identify which model, window, and pathway you are in when navigating from one window to the other. The clues are the following:

- From the menu select **Model** and then select the model you want to work with (**ISCST3, AERMOD, or ISC-PRIME**).



- The button, located on the lower right hand side of the window that identifies the name of the current window will be disabled (grayed). You may find a few buttons that are also disabled in addition to the button for the current window. In such case, these buttons are disabled because they are not applicable to you current project.

- The **Pathway Identification panel** located on the upper right hand side of each window contains the icon and the name that identifies the pathway for the current window.

Pathway Identification panel

For this tutorial, we will input data into the five (5) pathways in the following sequence:

- **Control Pathway**,
- **Source Pathway**,
- **Receptor Pathway**,
- **Meteorology Pathway**, and

- **Output Pathway**.

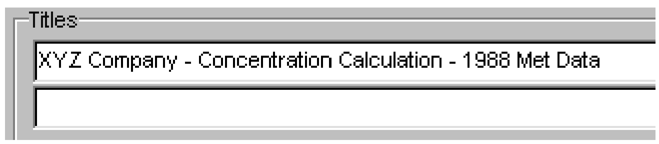
Using the information from the **Table 2-1**, we can start inputting data into our project.

CO-Dipersion Options Window

- **Titles:** The top field is for input of the first line of title and accepts up to 68 characters (including blanks). The bottom field is for an optional second line of title of the same length.

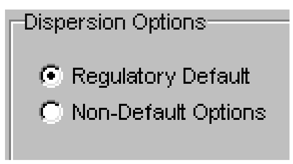
Step 5: Type in on the top field the following title:

XYZ Company - Concentration Calculation - 1988 Met Data



- **Regulatory Default:** this option is the default for regulatory applications.

Step 6: Select the **Regulatory Default** option.

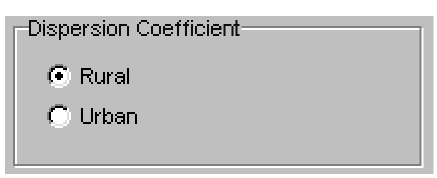


- **Output Types:** ISCST3 and ISC-PRIME can handle concentration and deposition calculations. AERMOD, at the present time, can only handle concentration calculations.

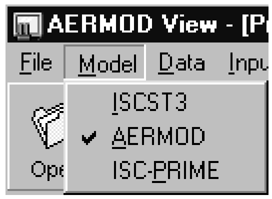
Step 7: Select **Concentration** for Output Types.

- **Dispersion Coefficient:** Either Rural or Urban dispersion coefficient can be selected, depending on the characteristics of site.

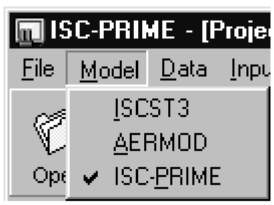
Step 8: Select **Rural** for the Dispersion Coefficient.

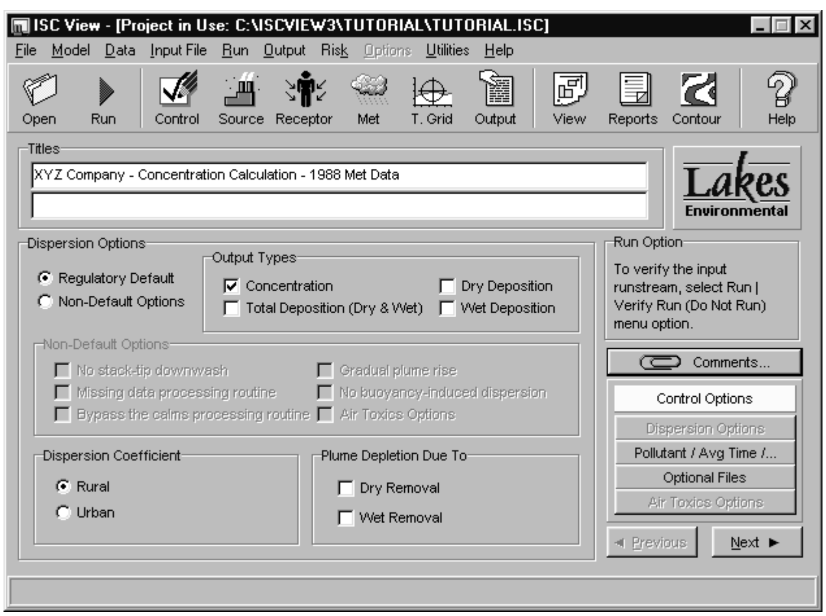


Step 9: Now, change the model by selecting **Model | AERMOD** from the menu. Note that the **CO-Dispersion Options** window changes. Note also that some of the data we input in ISC mode is also shown here. In AERMOD, for instance, rural dispersion option is used in the calculations unless user selects the **Urban Dispersion Option**.



Step 10: Verify the ISC-PRIME data for this window by selecting **Model | ISC-PRIME** from the menu. From now on, we will only change models if the data input for any of the models is different.



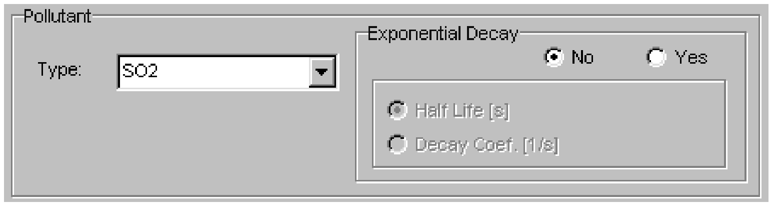


CO-Dispersion Options window

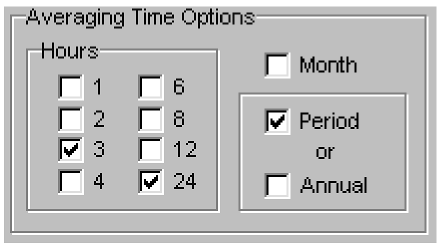
Step 11: Now that you have finished the data input in the **CO-Dispersion Options** window, press the **Next** button or the **Pollutant/Avg Time/...** button to go to the **CO-Pollutant/Avg Time/...** window.

CO-Pollutant/Avg Time/... Window

Step 12: The **Pollutant Type** list box already shows **SO_2** as the pollutant to be modeled. Leave the **Exponential Decay** option as **No** (default) since we are not going to specify half life or decay coefficient for SO_2.

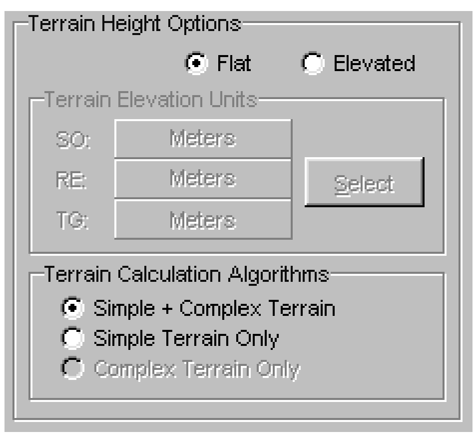


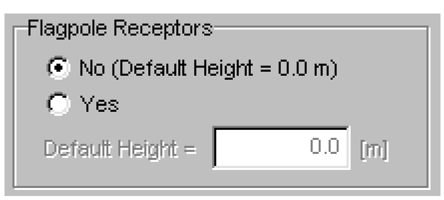
Step 13: Select the **3 Hours**, **24 Hours**, and **PERIOD** check boxes as **Averaging Time Options**.



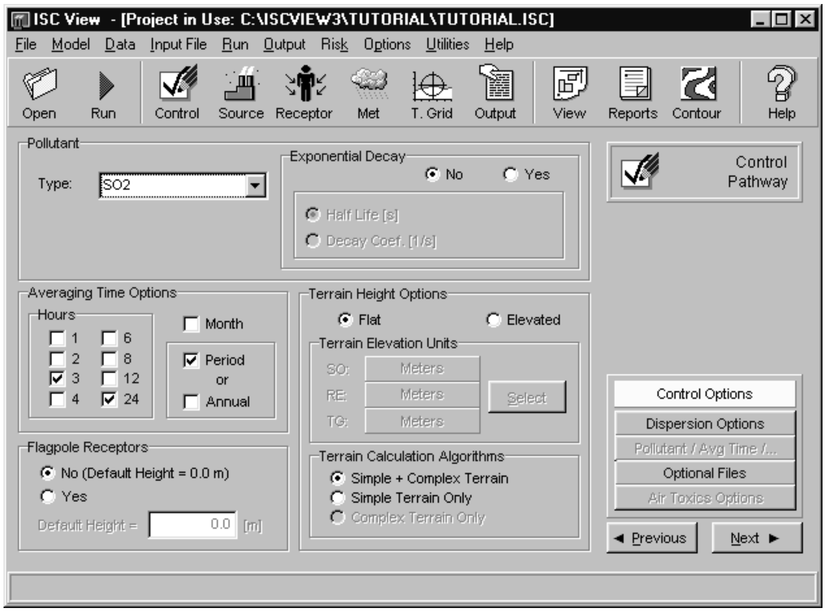
Step 14: Leave the default options:

- **Terrain Height Options:** *Flat*
- **Terrain Calculation Algorithms:** *Simple + Complex Terrain*
- **Flagpole Receptors:** *No.*





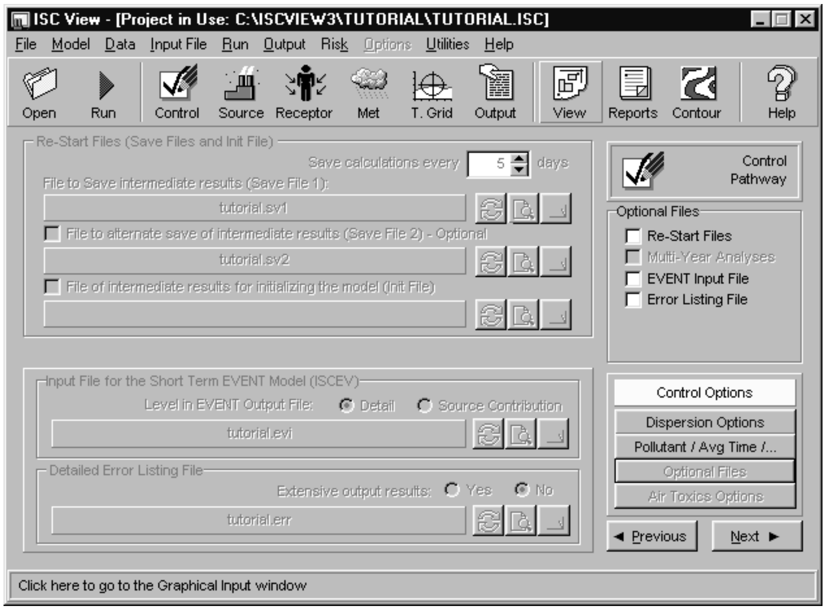
Step 15: You have finished the data input on the **CO-Pollutant/Avg Time/...** window. From here you could go to the next pathway, the **Source Pathway**. However, to make you more familiar with the windows we will just take a look into our next window in the **Control Pathway**. For this, press the **Next** button or the **Optional Files** button.



CO-Pollutant/Avg Time/... window

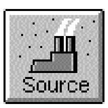
CO-Optional Files Window

The **CO-Optional Files** window has the flexibility of storing information for use in later runs. By simply checking or unchecking the boxes in the **Optional Files** frame, you define if the file already specified for these options will be generated in the current run. Click one or more of these check boxes to see what happens with these frames. You can input data in any one of these frames, but for this run, do not forget to clear all checked boxes in the **Optional Files** frame. Now it is time for the next step, the **Source Pathway**. Press the **Next** button or the **Source** button on the menu toolbar.

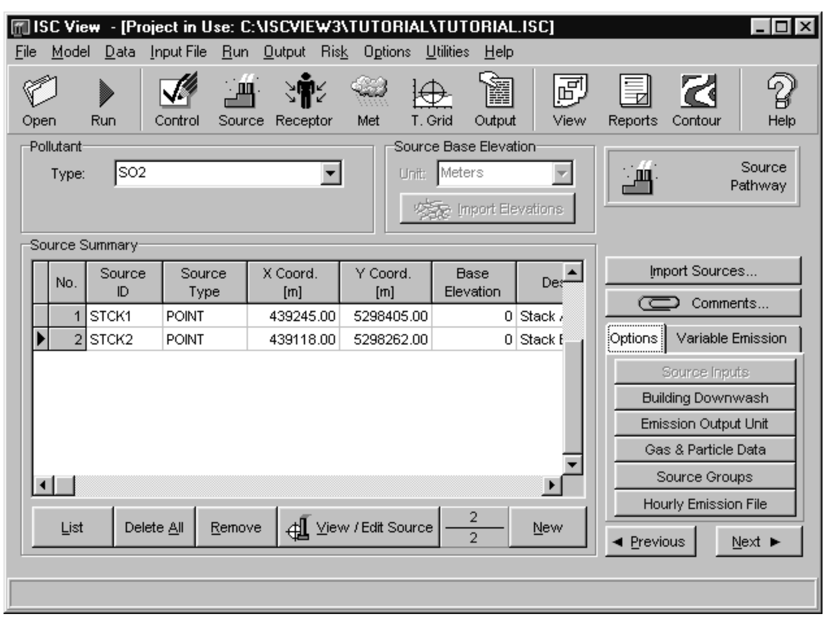


CO-Optional Files window

Working on the Source Pathway



The first window that appears after you click the **Source** toolbar button is the **SO-Source Inputs** window. In this window, you define the source or sources being modeled. Using the information from **Table 2-1**, follow the steps below:



SO-Sources Summary window

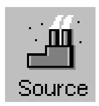
SO-Source Inputs Window or Graphical Input Window

You can input the source information in the following ways:

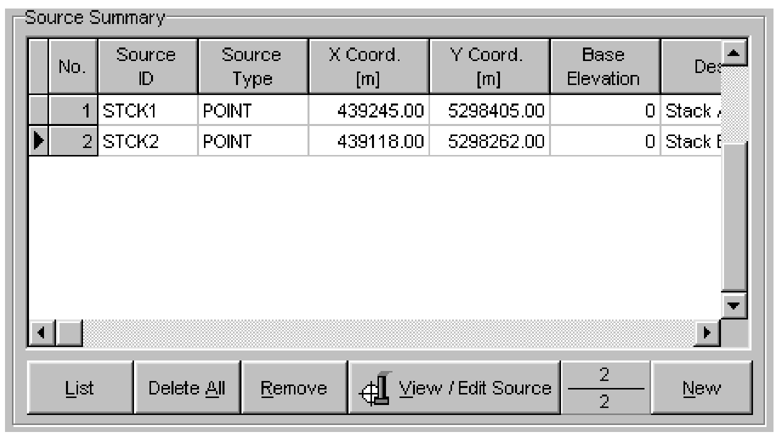
- In the **Source Inputs** dialog box (text mode),
- In the **Graphical Input** window (graphical mode)
- Importing from your BPIP View project (importing mode)

If you followed the BPIP View tutorial, and your BPIP View tutorial project has the same name and is located in the same directory as your ISC-AERMOD View project, then you will see that all the stacks you have defined in BPIP View are available in ISC-AERMOD View.

Step 16: Press the Source button to go to the Source Pathway. The **SO - Source Inputs** window is displayed. Note that the two stacks you defined in BPIP View also appear here.

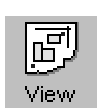


➤ *Source button*



Step 17: Press the **View/Edit Source** button to display the **Source Inputs** dialog box. From this dialog, you can check all the parameters for each source.

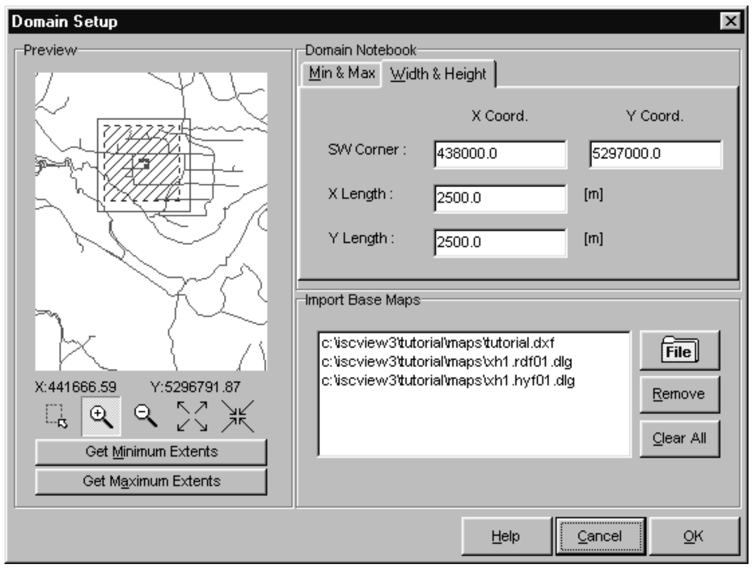
Step 18: To see your sources in graphical mode, press the **View** menu toolbar button. Note that the site maps and the stacks you defined in BPIP View are displayed on the drawing area of the **Graphical Input** window.



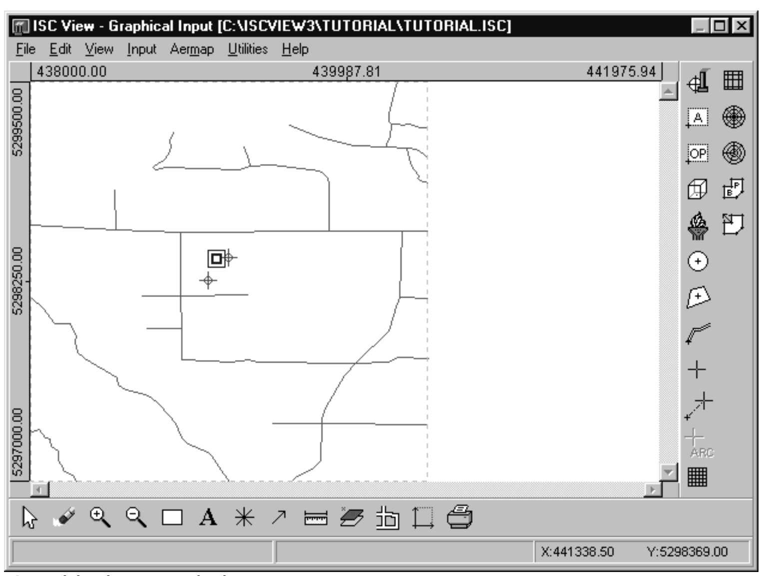
➤ *View button*

Step 19: You may need to adjust the domain size. Press the Domain tool ( ) to display the **Domain Setup** dialog box. Zoom in the area around the red box. The red box marks the current domain area. Click on the **Choose Extents** tool ( ) and draw a rectangle where you want the new domain to be. The selected domain extents are written into the Domain Notebook. See below the approximate coordinates and dimensions for the selected domain:

SW Corner: 438000.0 and 5297000.0 (for X and Y UTM Coordinates)
X Length: 2500 meters
Y Length: 2500 meters

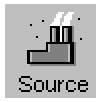


Domain Setup dialog box



Graphical Input window

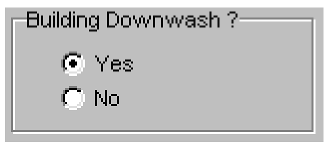
Step 20: Select **Input | Sources** from the menu to view the sources you already input in BPIP View. Close the **Graphical Input** window. If you are not in the Source Pathway, then press the **Source** menu toolbar button.



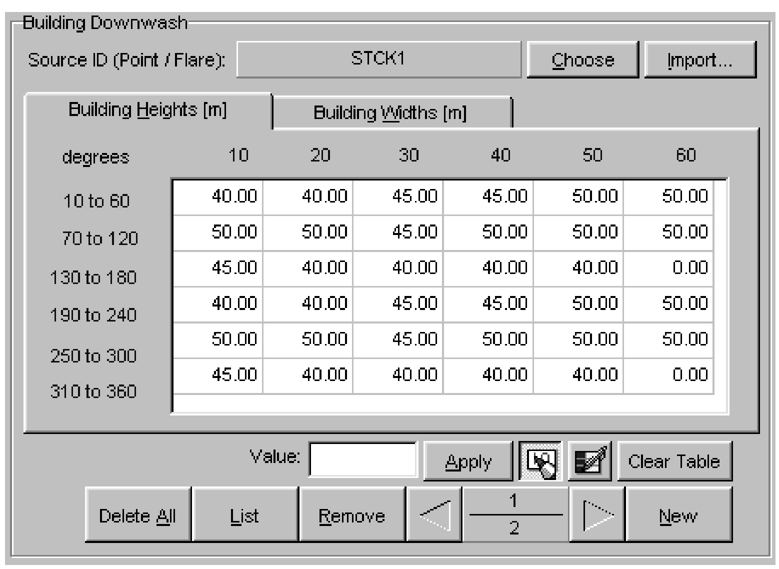
➤ *Source button*

Step 21: You should be now on the **Sources Inputs** window. From there click on the **Building Downwash** button. The **SO-Building Downwash** window is displayed.

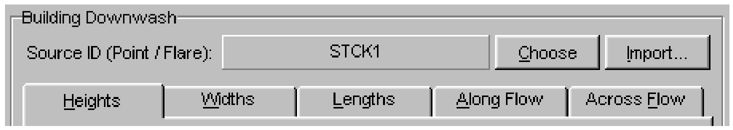
Step 22: Select **Yes** for the **Building Downwash** option, since we have to verify the building influence on the plume from the three buildings located nearby.



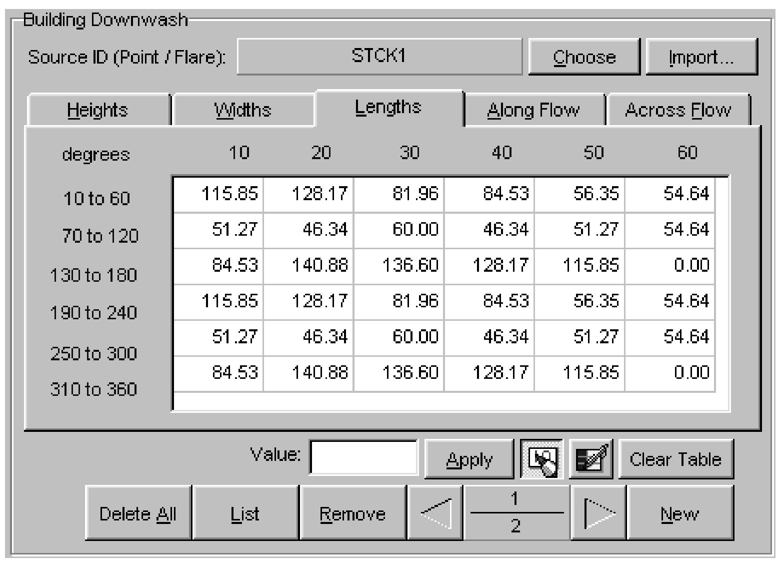
Step 23: Note that if your BPIP View project was created in the same directory as your ISC-AERMOD View project and with same name, then ISC-AERMOD View automatically places all the information contained in the BPIP Output file into the tables (Building Heights and Building Widths). You should have two building downwash records created, one record for STCK1 and one record for STCK2. If the building data was not placed automatically, then you should press the **Import** button and select the BPIP output file that was generated for your BPIP Tutorial project. The BPIP Output file extension is BPO.



Step 24: Now is time to change models again. AERMOD uses the same type of information as ISCST3. However, ISC-PRIME requires three additional parameters: the projected building length, the along-flow distance, and the across-flow distance. Lets change to the ISC-PRIME model by selecting **Model | ISC-PRIME** from the menu. Press on the **Building Downwash** button. Note that three more tabs were added to the **Building Downwash** window.



Step 25: Note that if your BPIP View project was created in the same directory as your ISC-AERMOD View project and with same name, then ISC-AERMOD View places all the information contained in the BPIP-PRIME Output file into the tables (Building Heights and Building Widths, Building Lengths, Along Flow Distance, and Across Flow Distance). You should have two building downwash records created, one record for STCK1 and one record for STCK2. If the building data was not placed automatically, then you should press the **Import** button and select the BPIP-PRIME output file that was generated for your BPIP Tutorial project. The BPIP-PRIME Output file extension is PRO.



degrees	10	20	30	40	50	60
10 to 60	115.85	128.17	81.96	84.53	56.35	54.64
70 to 120	51.27	46.34	60.00	46.34	51.27	54.64
130 to 180	84.53	140.88	136.60	128.17	115.85	0.00
190 to 240	115.85	128.17	81.96	84.53	56.35	54.64
250 to 300	51.27	46.34	60.00	46.34	51.27	54.64
310 to 360	84.53	140.88	136.60	128.17	115.85	0.00

Step 26: Go back to the ISCST3 Mode. Note that additional windows are available in the Source Pathway. The following options will not be used in this tutorial:

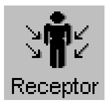
- **Emission Output Units:** We will use the default Concentration output units of MICROGRAMS/M^3.

- **Gas & Particle Data:** We will not specify particle information for the sources, since we are only modeling for Concentration values.

- **Source Groups:** For this tutorial, we are going to use only one source group, called **ALL**, which includes all sources being modeled. ISC-AERMOD View, by default, automatically creates the **Source Group 'ALL'** for you.

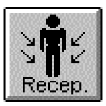
- **Variable Emission:** For this tutorial we are assuming that all the emissions are constant throughout the year (1 gram/second). Therefore, none of the variable emission windows will be used.

Step 27: We finished the input of data in the Source Pathway. The next pathway is the Receptor. Press the **Receptor** menu toolbar button.

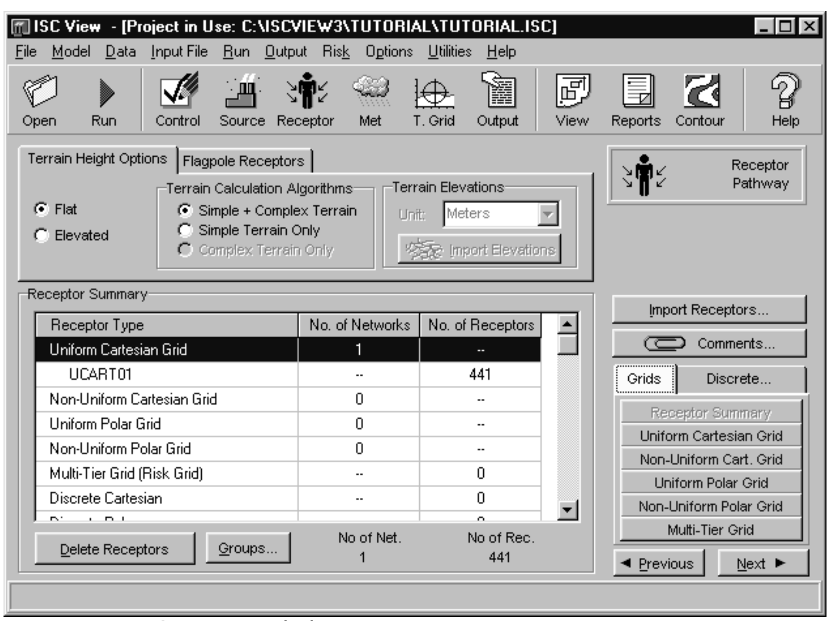


➤ *Receptor button*

Working on the Receptor Pathway



The first window on the **Receptor Pathway** is the **RE-Receptor Summary** window. This window contains summary information on the number of receptors defined for each receptor type, terrain height options, and flagpole receptors option. For this tutorial, we are going to define a uniform Cartesian grid of 2 km by 2 km centered around the stacks. To define this grid, double click on the **Receptor Summary** table on the row that displays *Uniform Cartesian Grid* as the receptor type. You can also press **the Uniform Cartesian Grid** button to jump to the **RE-Uniform Cartesian Grid** window.

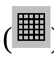


RE-Receptor Summary window

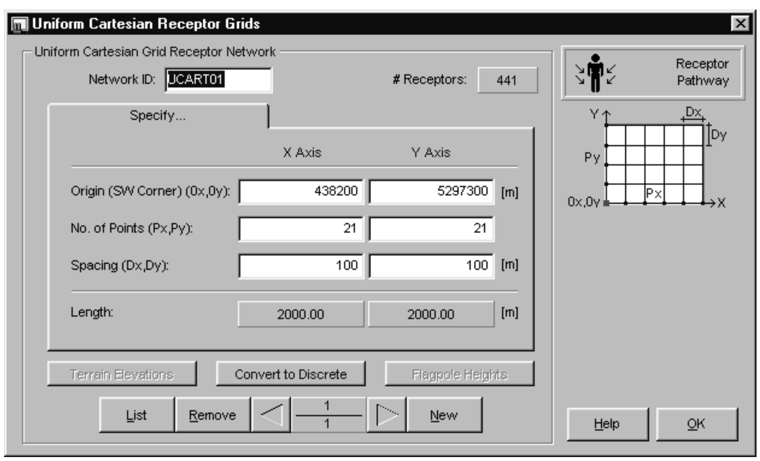
RE-Uniform Cartesian Grid Window

Using the information below, you can define the uniform Cartesian grid in the **RE-Uniform Cartesian Grid** window (text mode). However, you can also define your uniform Cartesian grid graphically in the **Graphical Input** window.

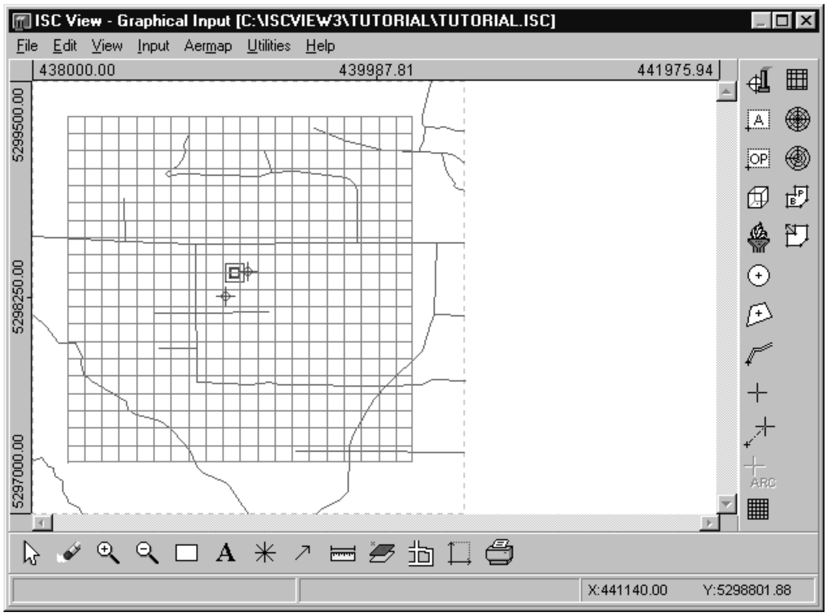
- **Receptor Network ID:** *UCART01*
- **Grid Origin (Ox, Oy):** *438200 and 5297300*
- **No. of X and Y Axis Receptors (Px, Py):** *21 by 21*
- **Spacing Between Receptors (Dx, Dy):** *100 meters, 100 meters*
- **Length:** *2000 by 2000 meters*

Step 28: To define your uniform Cartesian Grid graphically, go to the **Graphical Input** window by clicking on the **View** menu toolbar button. Select the **Uniform Cartesian Grid** tool () located on the toolbar. With the mouse pointer click on the drawing area on 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.

Step 29: The **Uniform Cartesian Receptor Grid** dialog box is displayed to allow you to adjust, if necessary, the coordinates for the southwest corner, the number of points, and the spacing. If you defined the grid graphically, edit the information to match that outlined above. Check the total number of receptors. It should have 441 receptors.

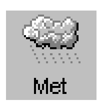


Uniform Cartesian Receptor Grid dialog box



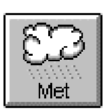
Graphical Input window

Step 30: When you have finished setting up your uniform Cartesian grid, move on to the Meteorology Pathway. Press the **Met** button on the menu toolbar. The **ME-Met Input Data** window is displayed.



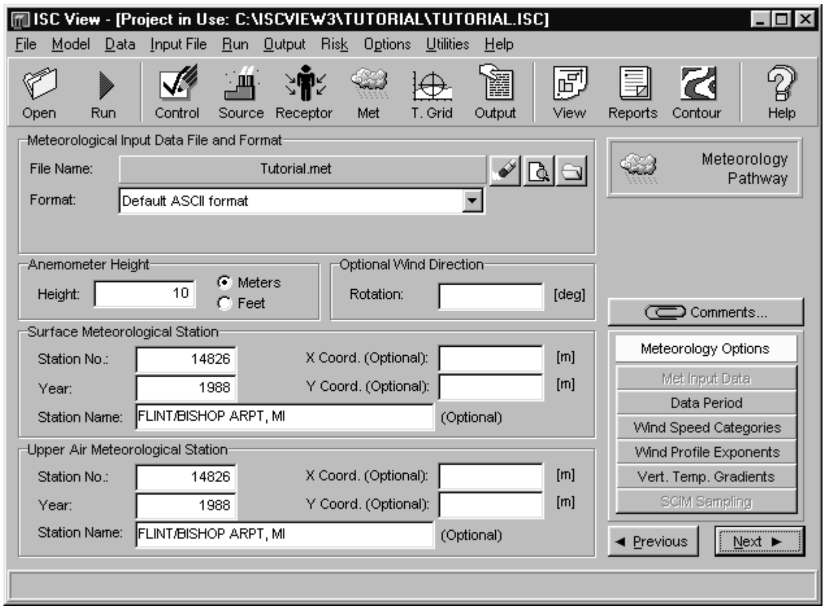
➤ *Met button*

Working on the Meteorology Pathway



The Meteorology Pathway is where we provide information regarding meteorology to the ISCST3, AERMOD, and ISC-PRIME models. ISCST3 and ISC-PRIME request the same information and both use the same meteorological data file. The met data file used by the ISCST3 and ISC-PRIME models is the one that you preprocessed using Rammet View.

AERMOD uses hourly meteorological data from separate surface and profile data files. These met data files are the ones you preprocessed using Aermet View.



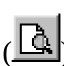
ME-Met Input Data window (ISCST3 and ISC-PRIME)

ME - Met Input Data Window

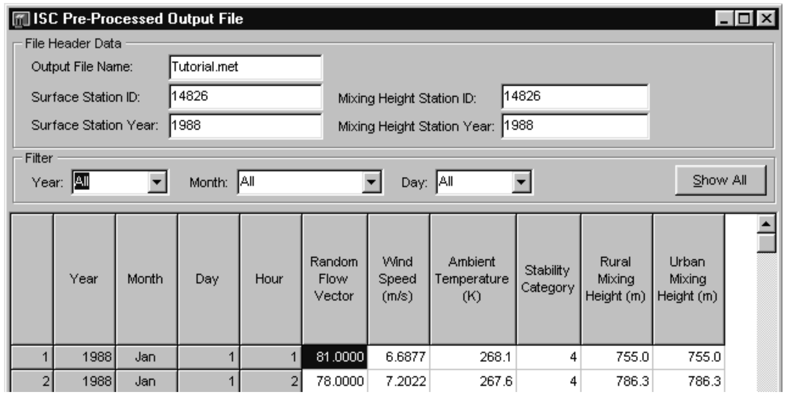
The ME – Met Input Data window is the main Meteorology Pathway window. In this window you specify the met data file, stations, and year of met data.

Follow Steps 31 to 36 to input data for the ISCST3 and ISC-PRIME models:

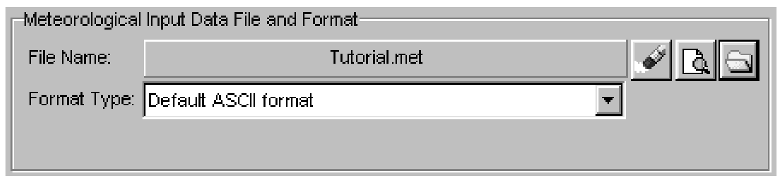
Step 31: Make sure you are in the ISCST3 or ISC-PRIME Mode.

Step 32: **Meteorological Input Data File** – Click on the file button () and select the met file that you have processed using Rammet View (TUTORIAL.MET). After specifying the meteorological input data file, press on the **Preview** button () and then select **Grid** from the pop up menu (

). The selected met data is then displayed in a grid format. A short description of the data contained in the met file is provided for easy reference.



Step 33: Format Type - Select Default ASCII format.



Step 34: Anemometer Height – Specify 10 meters.

Step 35: Surface Meteorological Station – ISC-AERMOD View reads the Station No. and Year from the specified met file and places theses values on the appropriate fields.

Station No. = 14826
Year = 1988
Station Name = FLINT/BISHOP ARPT, MI (this is an optional field that should be specified by the user)

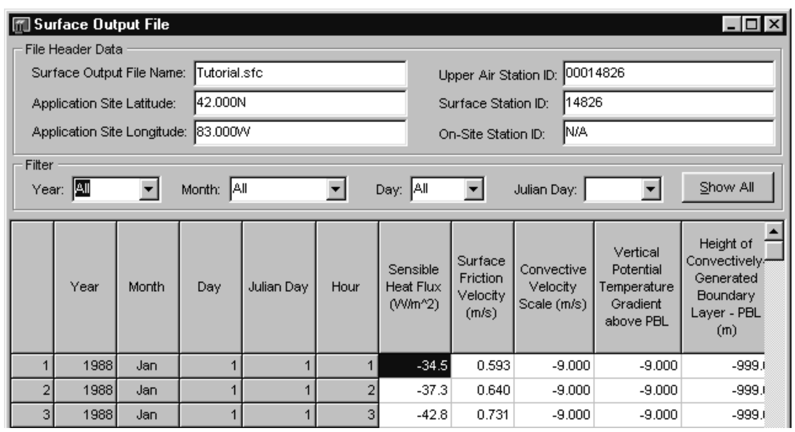
Step 36: Upper Air Meteorological Station – ISC-AERMOD View reads the Station No. and Year from the specified met file and places theses values on the appropriate fields.

Station No. = 14826
Year = 1988
Station Name = FLINT/BISHOP ARPT, MI (this is an optional field that should be specified by the user)

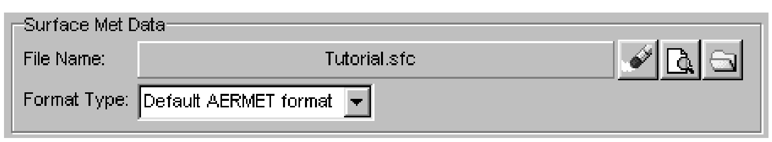
Follow Steps 37 to 45 to input data for the AERMOD model:

Step 37: Make sure you are in the AERMOD Mode by clicking on **Model | AERMOD** from the menu bar.

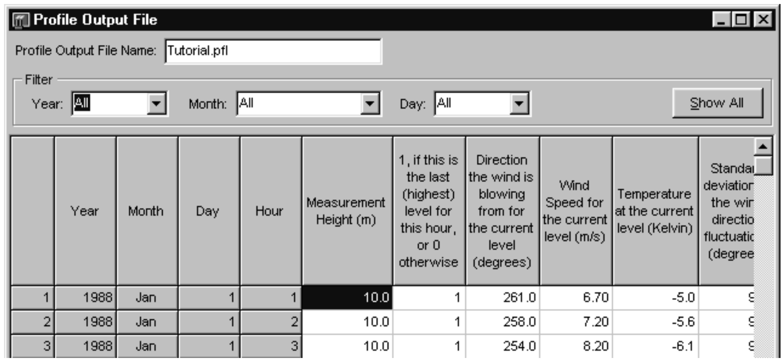
Step 38: **Surface Met Data** – Click on the file button () and select the surface met data file that you have processed using Aermet View (TUTORIAL.SFC). After specifying the surface met data file, press on the **Preview** button () and then select **Grid** from the pop up menu (Text... Grid...). The selected surface met data is then displayed in a grid format. A short description of the data contained in the met file is provided for easy reference.



Step 39: **Format Type** - Select Default AERMET format.



Step 40: **Profile Met Data** – Click on the file button () and select the profile met data file that you have processed using Aermet View (TUTORIAL.PFL). After specifying the profile met data file, press on the **Preview** button () and then select **Grid** from the pop up menu (Text... Grid...). The selected profile met data is then displayed in a grid format. A short description of the data contained in the met file is provided for easy reference.

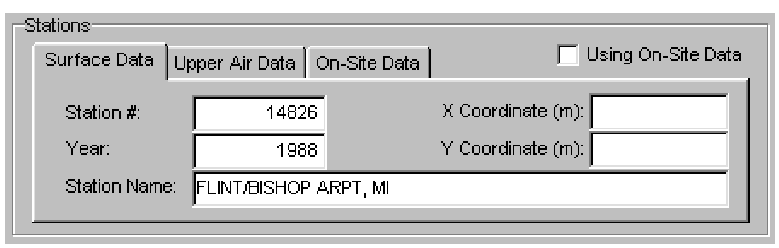


Step 41: **Format Type -** Select Default AERMET format.

Step 42: **Potential Temperature Profile** – Specify 0 as the base elevation above sea mean level for the primary met tower.

Step 43: **Stations – Surface Data** – ISC-AERMOD View reads the Station No. and Year from the specified surface met data file and places theses values on the appropriate fields.

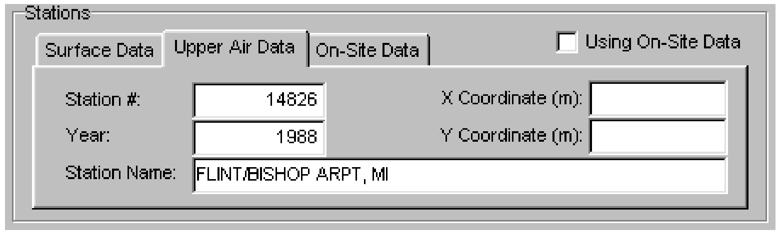
Station No. = 14826
Year = 1988
Station Name = FLINT/BISHOP ARPT, MI (this is an optional field that should be specified by the user)



Step 44: **Stations – Upper Air Data** – ISC-AERMOD View reads the Station No. and Year from the specified surface met data file and places these values on the appropriate fields.

Station No. = 14826

Year = 1988
Station Name = FLINT/BISHOP ARPT, MI (this is an optional field that should be specified by the user)



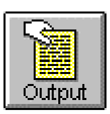
Step 45: Since we did not use On-Site data when preprocessing the met data with Aermet View, the on **On-Site Data** tab cannot be open.

ME – Data Period Window

Now that we finished inputting data on the **ME-Met Input Data** for all the models, lets check the **ME-Data Period** window by pressing the **Data Period** button. In this window, you can specify the data period to read from the met data file(s). By default, the model will read the entire met data file(s).

No more inputs are needed on the **Meteorology Pathway**. Let's go to the next Pathway.

Working on the Output Pathway

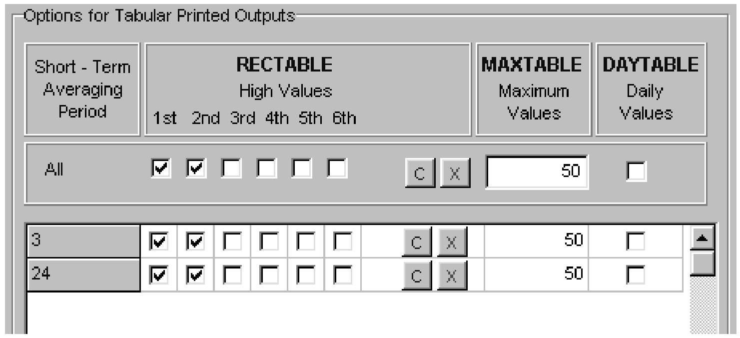


The **Output Pathway** is where you indicate what type of results are required for the present study. You go to the Output Pathway by pressing the **Output** menu toolbar button. The first window in the Output Pathway is the **OU-Tabular Outputs** window.

OU-Tabular Outputs Window

Step 46: **RECTABLE Option (High Values) -** Select the First (**1st**) and Second (**2nd**) highest values by receptor for all short term averages (**3** hrs and **24** hrs). Note that if you check the 1st and 2nd check boxes for ALL then all check boxes for that specific high value are checked for all short-term averages.

Step 47: MAXTABLE Option (Maximum Values) – Specify 50 maximum values for all short-term averages. Input 50 on the MAXTABLE option for ALL.

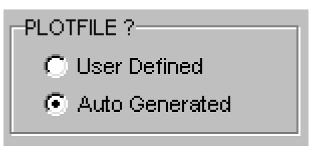


OU-Tabular Outputs

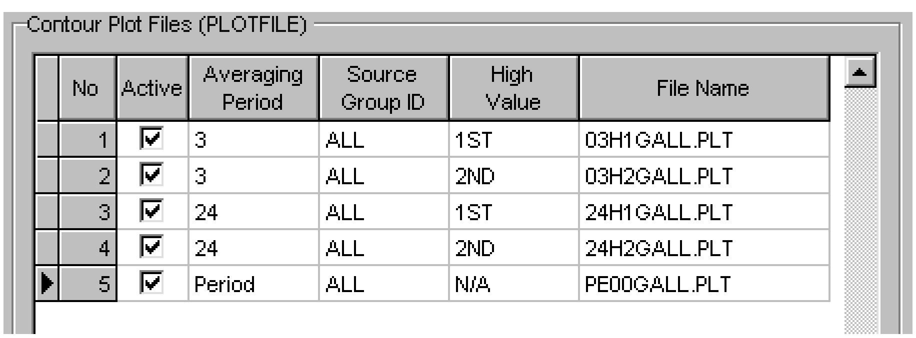
In order to have the model results displayed as contour plots using POST View, we need to specify to the models that Contour Plotfiles are to be generated. You do this in the **OU-Contour Plot Files** window.

OU-Contour Plot Files Window

ISC-AERMOD View automatically generates all possible combinations of plot files that can be setup for the current run (**Auto Generated** option).



You can discard one or more of these plot files by unchecking the **Active** field. For the tutorial project we will use all the auto-generated plots files. Theses plot files will later be converted into contour plots by POST View.


Contour Plot Files (PLOTFILE)

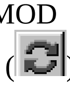
No	Active	Averaging Period	Source Group ID	High Value	File Name
1	☑	3	ALL	1ST	03H1GALL.PLT
2	☑	3	ALL	2ND	03H2GALL.PLT
3	☑	24	ALL	1ST	24H1GALL.PLT
4	☑	24	ALL	2ND	24H2GALL.PLT
5	☑	Period	ALL	N/A	PE00GALL.PLT

Note that you can specify the location where all the plot files are to be placed. The names of the auto-generated plotfiles are the same for ISCST3, AERMOD, and ISC-PRIME. Therefore, make sure you specify different paths for the plot files on each model. By default, ISC-AERMOD View specifies that these files should be placed on the project directory in the following folders:

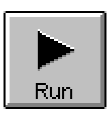
- ***ProjectName.IS*** for the Plotfiles generated by the ISCST3 model
- ***ProjectName.AE*** for the Plotfiles generated by the AERMOD model
- ***ProjectName.PR*** for the Plotfiles generated by the ISC-PRIME model

NOTE: Note that the panel that displays the selected path only shows, for example, **TUTORIAL.IS** instead of **C:\ISCVIEW3\TUTORIAL\TUTORIAL.IS**.

If you select a folder that is not within the project folder, then the full path is displayed.

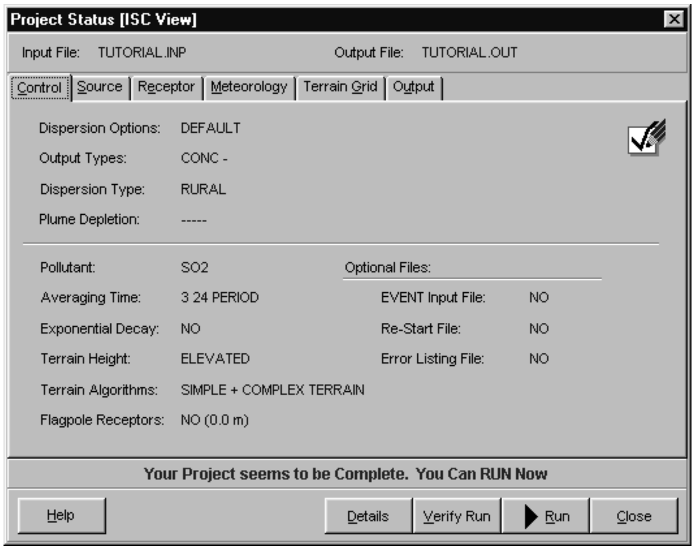
With the filename path scheme described above, you can easily copy your project to another directory structure, and ISC-AERMOD view will automatically adjust the file paths. Use the default button () to get the default path for the plot files.

Running the Three U.S. EPA Models



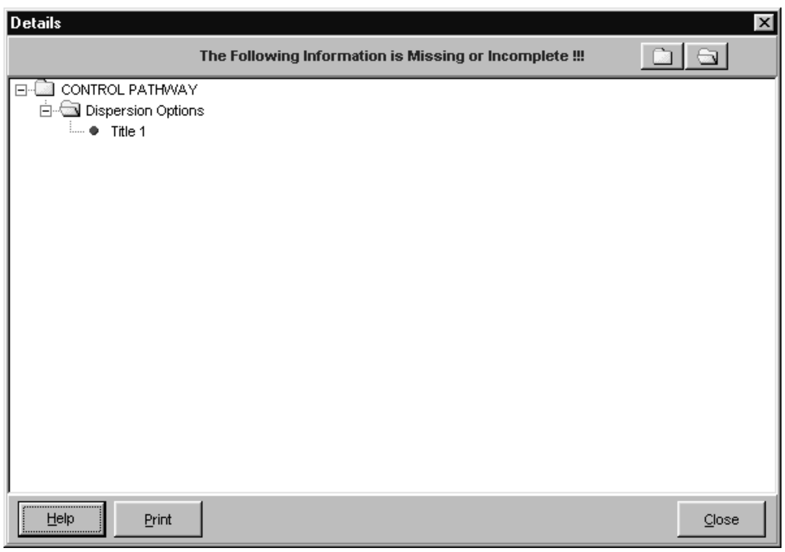
Your project should be complete now. Before running your project, we suggest that you follow these steps:

Step 48: Check the Project Status to make sure your options are correct. Select **Run | Status**.



Project Status dialog box

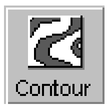
Step 49: Check the details of your project. If any crucial piece of information is missing, it will be displayed on the **Details** dialog box. You can get to the **Details** dialog box by selecting **Run | Details** or pressing the **Run** toolbar button to display the **Project Status** dialog box and then clicking on the **Details** button, at the bottom of the Status dialog box, as shown in the figure below.



Details dialog box

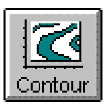
Step 50: When all the necessary information is supplied, you can click on the **Run** button and ISC-AERMOD View will translate all the given data into an input file and will run this input file into the selected model. Make sure you change modes (ISCST3, AERMOD, and ISC-PRIME) to run each model.

Step 51: If your run finishes successfully, then a message box is displayed asking you what you want to see. You can either choose to see the produced output file by pressing on the **Output file** button or you can view contours plots for the generated AERMOD Plotfiles by pressing on the **Contours** button. For now, you should press the **Contours** button. If you pressed another button, you should be able to see your contours by pressing on the **Contour** menu toolbar button.



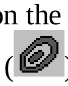
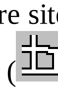
Contour menu toolbar button

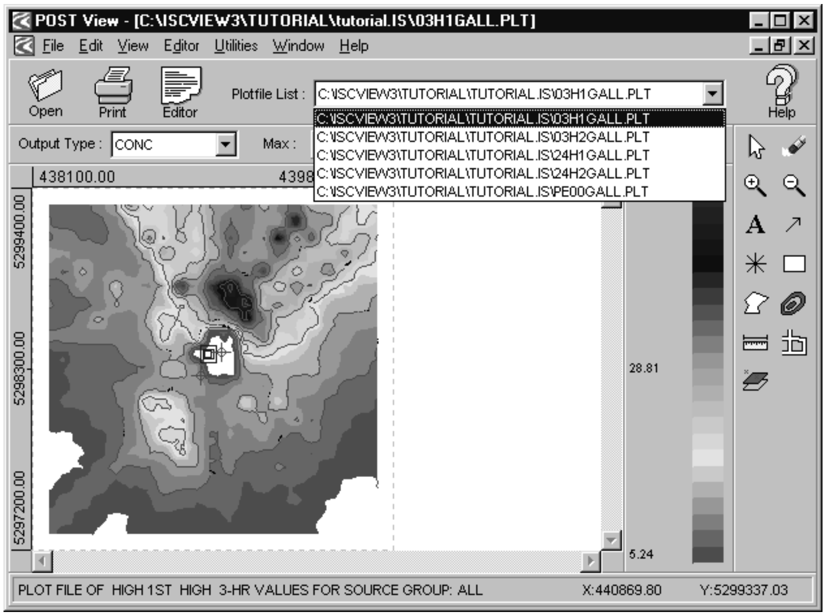
Post-Processing of Results with POST View



POST View will perform a lot of the post-processing for you in the background. This way, the tasks of gridding the data for plotting and displaying the isopleths are automatically accomplished.

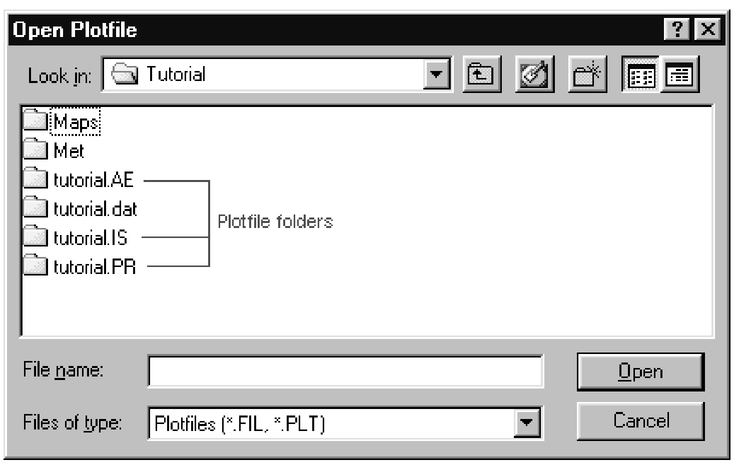
POST View automatically opens the plot files generated for the current model when you press the **Contour** menu toolbar button in ISC-AERMOD View. The first plot file that you have defined on the **OU-Contour Plot Files** window will be automatically loaded and contours (isopleths) will be generated. To open the other plot files, just click on the down arrow of the **Plotfile List** drop-down list box.

In POST View you can modify the default options for contour levels, labeling, contour lines, grid size, overlay options, etc. on the **Contour Options** dialog box. To display this dialog box click on the **Contour** tool (). You can also display one or more site maps as overlays. You can import base maps by clicking the **Import Base Map** tool ().

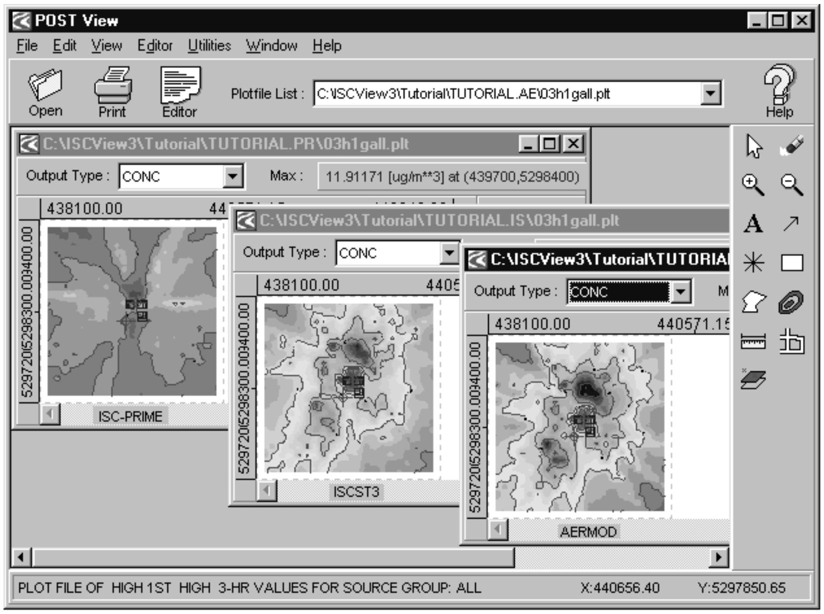


POST View – ISCST3 Plotfile for high 1st high 3-hour and source group ALL

You can easily compare results for all the three models by opening the plotfiles for the same high value, averaging period and source group. Remember that by default, the plotfiles produced by the ISCST3 model are placed in the project directory, in the ***ProjectName.IS*** folder. The AERMOD plot files are in the ***ProjectName.AE*** folder and the ISC-PRIME plot files are in the ***ProjectName.PR*** folder.



Open Plotfile dialog box



Comparing results between the three models

Terrain Processing

The ISC-AERMOD View Tutorial was processed using the FLAT terrain option. In this section, we will see how the model results change if the ELEVATED terrain option is introduced.

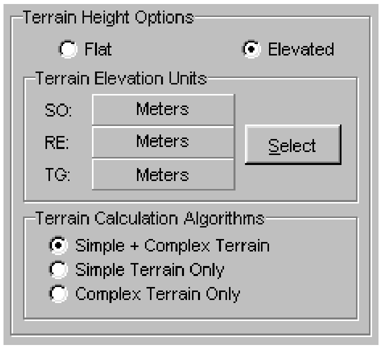
Changing Terrain Height Options

Two terrain height options are available in ISCST3, AERMOD, and ISC-PRIME: FLAT and ELEVATED. Follow the steps below on how to change the terrain height from FLAT to ELEVATED:

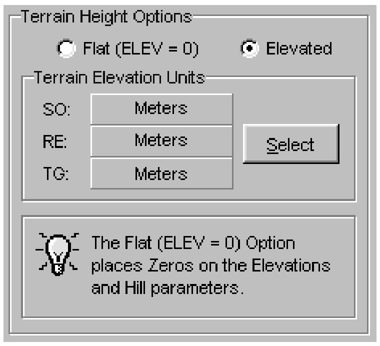
Step 1: Start the ISC-AERMOD View application and open the Tutorial file that you created in the previous sections.

Step 2: The first window displayed is the **CO-Dispersion Options** window. From there click on the **Pollutant / Avg Time /…** button.

Step 3: If you are in the **ISCST3** or **ISC-PRIME** mode, then click on the **Elevated** option. For the **Terrain Elevation Units**, we will use the default, **Meters**. For the **Terrain Calculation Algorithms** option we will also use the default**,** **Simple + Complex Terrain**.

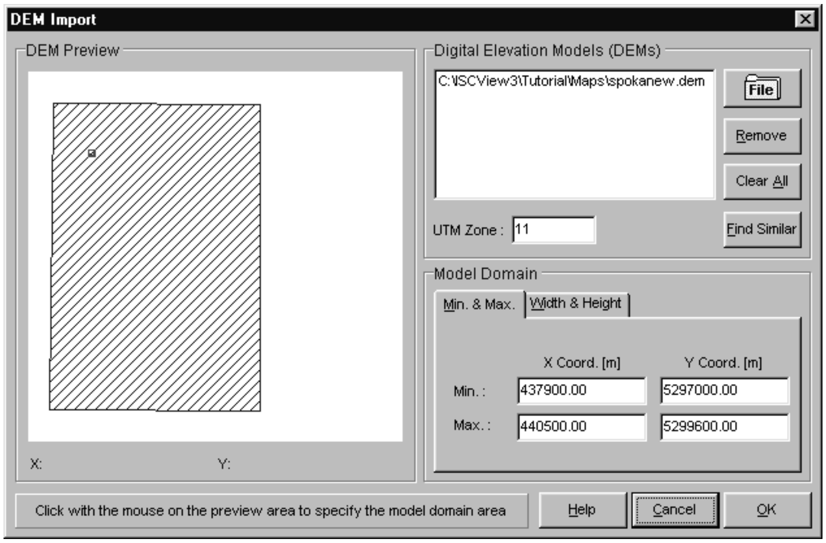


Step 4: If you change to AERMOD, then you should see the **Terrain Height Options** as displayed on figure below.



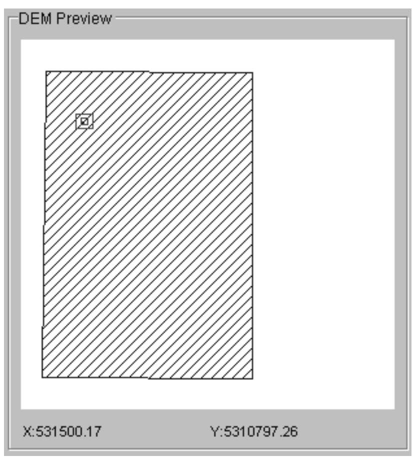
Step 5: Now we need to setup the DEM file from which ISCST3 and ISC-PRIME terrain elevations should be imported. Switch back to ISCST3 or ISC-PRIME under Models. Select **File | Import | Terrain Elevations | USGS DEM** from the menu. The **DEM Import** dialog box is displayed.

Step 6: Press the **File** button and select the DEM file that is located on your installation directory under **\tutorial\maps** folder. In the **Preview** area, you can visualize the area covered by the imported DEM file (blue box), and the current project domain area (green box).

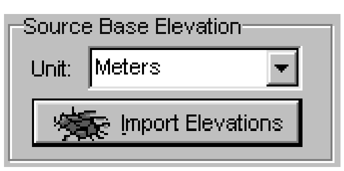


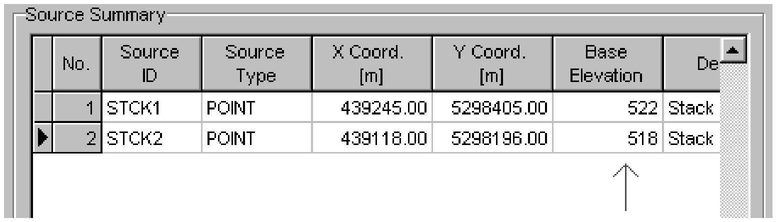
DEM Import dialog box

Step 7: Click with the mouse on the **Preview** area to specify the extents of the DEM to be processed. Draw a rectangle around the green box (current model domain). The selected domain area (marked in red) will define the limits of the area to be imported. Press the **OK** button. ISC-AERMOD View will extract from the DEM file all the necessary information and will store it in an intermediate file.



Step 8: Now go to the **Source Pathway** by pressing the **Source** menu toolbar button. In the **SO-Source Inputs** window, click on the **Import Elevations** button. ISC-AERMOD View will then import all the base elevations for all the sources and automatically place them in the **Base Elevation** field.



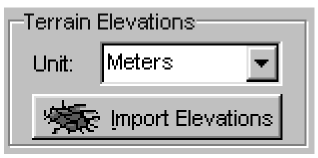

Source Summary

	No.	Source ID	Source Type	X Coord. [m]	Y Coord. [m]	Base Elevation	De
	1	STCK1	POINT	439245.00	5298405.00	522	Stack
▶	2	STCK2	POINT	439118.00	5298196.00	518	Stack

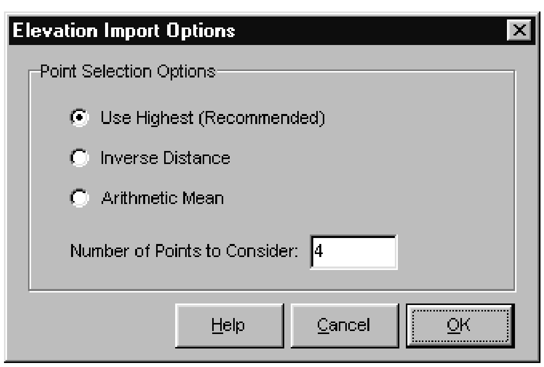
Step 9: Go to the **Receptor Pathway** by pressing the **Receptor** menu toolbar button.

Importing Receptor Elevations for ISC and PRIME

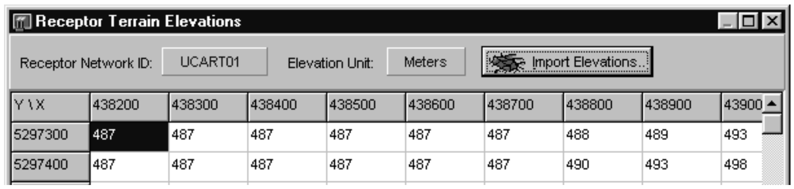
Step 10: On the **RE-Receptor Summary** window, note that the **Import Elevations** button became enabled. Click this button. A message box is displayed informing that existing elevations will be overwritten and that the elevations will be imported in Meters. Press the **OK** button. The **Elevation Import Options** dialog box is displayed.



Step 11: On the **Elevation Import Options** dialog box leave the default option selected (Use Highest) and press the **OK** button. ISC-AERMOD View will then import all the terrain elevations for all the grid node points and place them in the table.



Step 12: Check the imported terrain elevations by going to **the RE-Uniform Cartesian Grid** window and pressing the **Terrain Elevations** button.



Step 13: We are done with the import of terrain elevations from DEM for the ISCST3 and ISC-PRIME mode. Lets run ISCST3 and ISC-PRIME. Make sure you are in the ISCST3 mode. Select **Run | Run ISCST3** from the menu. The DOS window is displayed and your run starts. When the run is complete, close the DOS window. A message is displayed asking you what you want to see. Select **Contours**. This should open the first generated ISCST3 plot file in POST View.

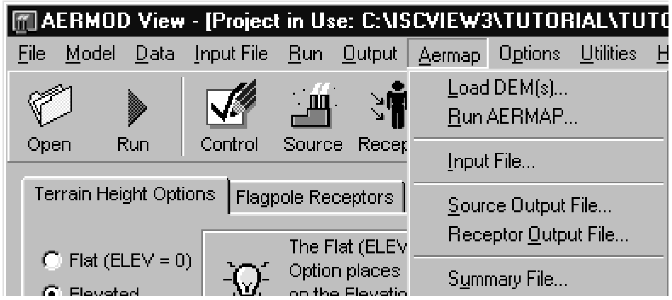
Step 14: You can modify the default options for contour levels, labeling, contour lines, grid size, overlay options, etc. by pressing the **Contour** tool (). You can also display site maps as overlays.

Step 15: Change to the ISC-PRIME mode. Follow Steps 13 to 14 to run the ISC-Prime model.

Importing Receptor Elevations for AERMOD

Step 16: Change to **AERMOD** and select **Aermap | Load DEM(s)…** from the menu. The **DEM Import** dialog is displayed. Make sure the DEM file that you specified for ISC is also specified here. Press the **OK** button.

Step 17: Now select **Aermap | Run…** from the menu. The U.S. EPA AERMAP model starts the run. When the AERMAP run is complete, close the DOS window and select **Aermap | View | Receptor Output**. This is the file created by AERMAP that is going to be included in the AERMOD input file. The **Receptor Output** file contains AERMAP calculated values for Terrain Elevations (ELEV) and Hill values (HILL) for all receptors defined for the current run.



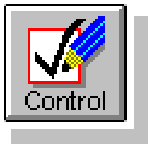
Step 18: Let's run AERMOD now. Select **Run | Run AERMOD** from the menu. The DOS window is displayed and your run starts. When the run is complete, close the DOS window. A message is displayed asking what you want to see. Select **Contours**. This should open the generated AERMOD plot file in POST View.

Step 19: You can modify the default options for contour levels, labeling, contour lines, grid size, overlay options, etc. by pressing the **Contour** tool (). You can also display site maps as overlays.

Step 20: Now you can compare the results for the three model runs for FLAT and ELEVATED terrain.

CHAPTER 3

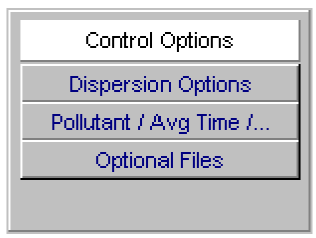
Control Pathway



This chapter explains ISC-AERMOD View inputs and options in the Control Pathway (CO). The Control Pathway provides the overall control of the model run. In ISC-AERMOD View, the Control Pathway inputs and options are available to you in four windows. The contents of these four windows will be explained in detail in the sections that follow.

Contents

ISCST3



ISC-PRIME &AERMOD

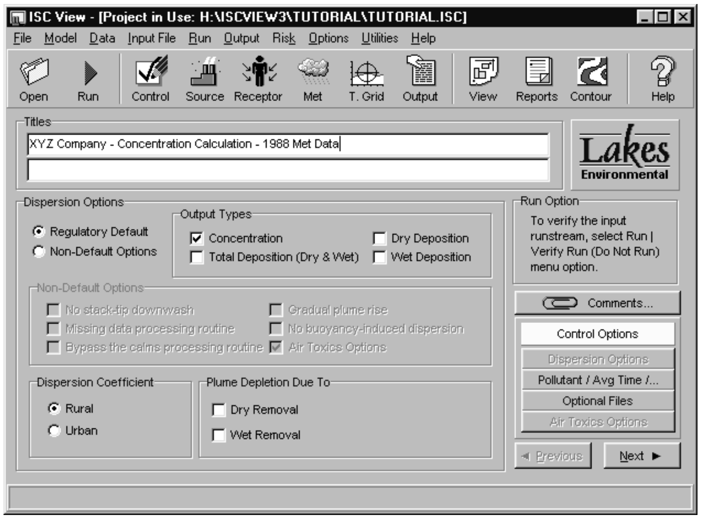
CO - Dispersion Options Window

The **CO-Dispersion Options** window is the first window displayed on your screen after you have pressed the **OK** button on the **About** dialog box.

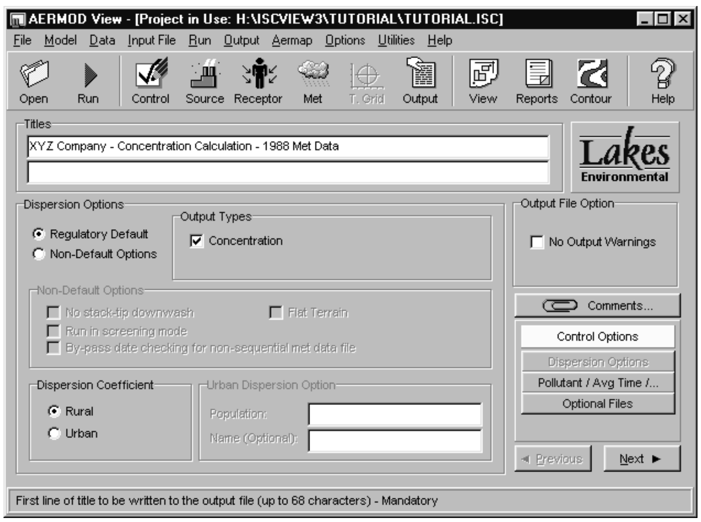
You have access to the **CO-Dispersion Options** window by pressing the **Control** button located on the menu toolbar. If you are in another Control Pathway window, then press the **Dispersion Options** button located on the lower right side of any Control Pathway window. From the menu select **Data | Control | Dispersion Options**.

or

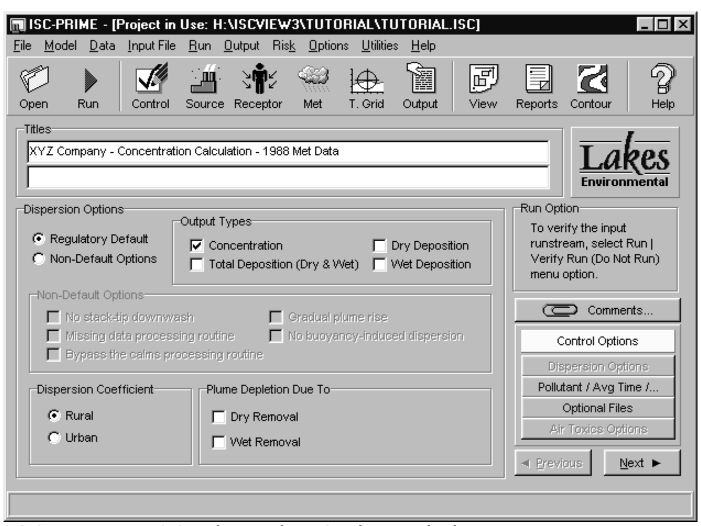

= Displays the **OU-Dispersion Options** window



ISCST3 - *CO-Dispersion Options window*



AERMOD - *CO-Dispersion Options window*



ISC-PRIME - *CO-Dispersion Options window*

Specifying Dispersion Options

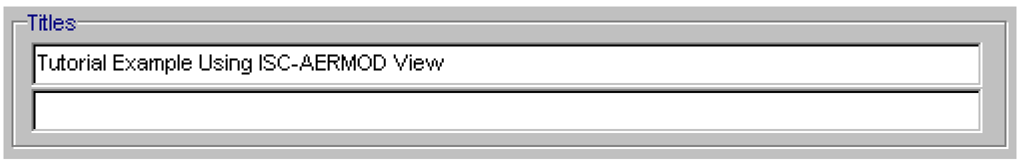
You should start entering data in your project on the **CO-Dispersion Options** window, since most of the information requested here will be needed to complete other windows. In this window, you define the titles, dispersion options, and the run option. See the description of each one of these options below.

Titles

The user can specify up to two lines of title information. These two lines of title will be printed on each page of the main output file. Each line of the title accepts up to 68 characters.

- **Title 1:** The first line of the title is mandatory.

- **Title 2:** The second line of the title is optional.



Regulatory Default Options

The **Regulatory Default** option specifies that the following regulatory default options will be used for each model:

➤ **ISCST3 & ISC PRIME Models**

- Use stack-tip downwash (except for Schulman-Scire downwash)
- Use buoyancy-induced dispersion (except for Schulman-Scire downwash)
- Do not use gradual plume rise (except for building downwash)
- Use the calms processing routines
- Use upper-bound concentration estimates for sources influenced by building downwash from super-squat buildings
- Use default wind speed profile exponents
- Use default vertical potential temperature gradients

➤ **AERMOD Model**

The regulatory default option in AERMOD includes the following:

- Use stack-tip downwash (except for Schulman-Scire downwash)
- Incorporate the effects of elevated terrain
- Use the calms processing routines
- Use missing data processing routine

- Use upper-bound concentration estimates for sources influenced by building downwash from super-squat buildings
- Use a 4-hour half life for exponential decay of SO_2 for urban sources
- Does not allow for exponential decay for other applications.

The default values for wind profile exponents and vertical potential temperature gradients are listed on Tables 3-1 and 3-2 below.

Table 3-1: 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

Table 3-2: Default Vertical Potential Temperature Gradients (K/m)

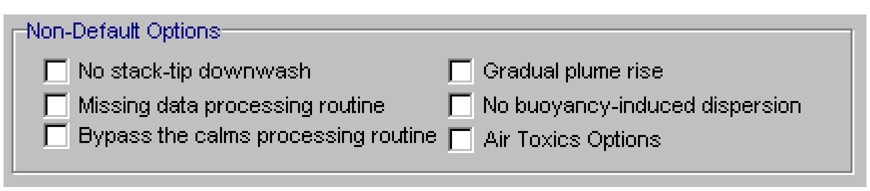
Pasquill Stability Category	Rural	Urban
A	0.000	0.000
B	0.000	0.000
C	0.000	0.000
D	0.000	0.000
E	0.020	0.020
F	0.035	0.035

Non-Regulatory Default Options

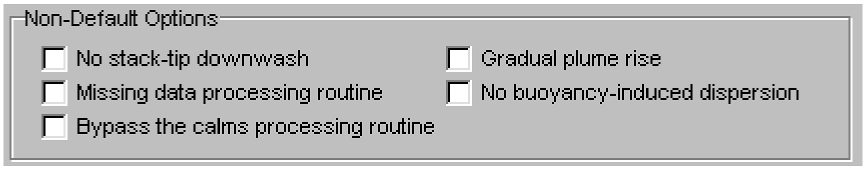
If the **Non-Default Options** option is selected then at least one non-default option should be selected from the **Non-Default Options** frame. The following are the **Non-Default Options** available:

➤ **ISCST3 & ISC PRIME Models**

- No stack-tip downwash (NOSTD)
- Missing data processing routine (MSGPRO)
- Bypass the calms processing routine (NOCALM)
- Gradual plume rise (GRDRIS)
- No buoyancy-induced dispersion (NOBID)
- Air Toxics Options (TOXICS) **(ISCST3 Only)**



ISCST3 - Non-Default Options Frame

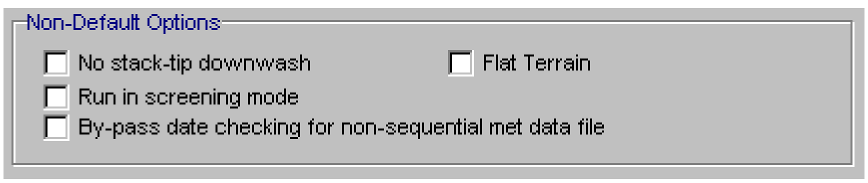


ISC-PRIME - Non-Default Options Frame

If the **Air Toxics Options** is selected, then the **Air Toxics Options** window will became enabled for use. For the options available in this window, please refer to the **Air Toxics Options** window topic in this chapter.

➤ **AERMOD Model**

- No stack-tip downwash (NOSTD)
- Run in screening mode (SCREEN)
- By-pass date checking for non-sequential met data file
- Flat terrain (FLAT)



AERMOD - Non-Default Options Frame

Note: If you selected the **Non-Default Options**, you must select at least one non-default option from the **Non-Default Options** frame.

Output Types

If you are in the ISCST3 or ISC-PRIME mode, then you may select any or all of the output types, **Concentration**, **Total Deposition (Dry and Wet)**, **Dry Deposition**, and/or **Wet Deposition**, in a single run. At the present date, deposition algorithms have not been implemented in AERMOD yet.

ISCST3 & ISC-PRIME - Output Types Options

AERMOD – Output Types Options

- **Concentration:** This is the default option and specifies that concentration values will be calculated. If you do not select any one of the **Output Types** check boxes, then ISC-AERMOD View will automatically check this option as the default.

- **Total Deposition (Dry and Wet):** This option specifies that total deposition flux values (dry and wet) will be calculated.

- **Dry Deposition:** This option specifies that dry deposition flux values only will be calculated.

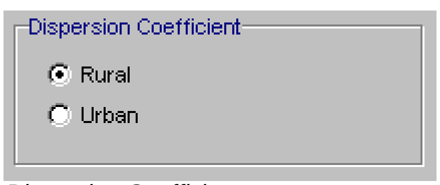
- **Wet Deposition:** This option specifies that wet deposition flux values only will be calculated.

Note: If you select the **Total Deposition (Dry and Wet)** option or the **Wet Deposition** option, then you MUST define scavenging coefficients (for particulate sources or gaseous sources) on the **SO-Gas & Particle Data** window.

Note: When modeling with the **Total Deposition (Dry and Wet)** option or the **Dry Deposition** option, you MUST include particle information for each source on the **SO-Gas & Particle Data** window.

Dispersion Coefficient (ISCST3 & ISC PRIME Only)

Either **Rural** or **Urban** dispersion coefficient can be selected, depending on the characteristics of the source location.



Dispersion Coefficient

The classification of a site as urban or rural should be based on one of the procedures described on the EPA document “Guideline on Air Quality Models. These procedures are described below:

➤ **How to Classify a Site as Rural or Urban:**

The selection of either urban or rural dispersion coefficients should be based upon 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,

  (a) if land use types I1, I2, C1, R2, and R3 account for 50 % or more of ***A***o, select the **Urban** option,

(b) otherwise, use the **Rural** option.

- **Population Density Procedure:** Compute the average population density, $\boldsymbol{p}$, per square kilometer with $\boldsymbol{A}$o as defined above,

  (a) If $\boldsymbol{p} > 750$ people/km^2, select the **Urban** option,
  (b) If $\boldsymbol{p} <= 750$ people/km^2, select the **Rural** option.

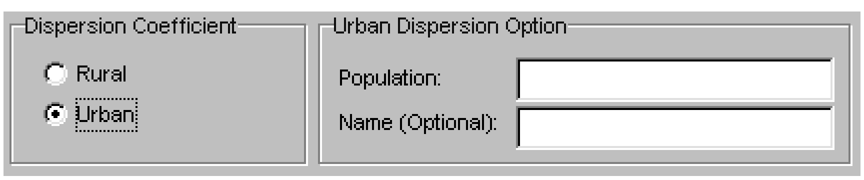
Of the two methods above, the **Land Use Procedure** is considered a more definitive criterion. The population density procedure should be used with caution and should not be applied to highly industrialized areas where the population density may be low and thus a rural classification would be indicated, 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.

Urban Dispersion Option (AERMOD Only)

AERMOD allows you to incorporate the effects of increased surface heating from an urban area on pollutant dispersion under stable atmospheric conditions by using the **Urban Dispersion Option**. The following parameters are needed if the **Urban** dispersion coefficient option was selected:

- **Population:** This is the population of the urban area.
- **Name (Optional):** This may be used to identify the name of the urban area.

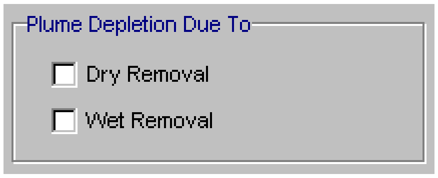
If the **Urban** dispersion option is selected, then you also need to identify which sources are to be modeled with urban effects. These sources are specified in the Source Pathway on the **SO-Urban Sources** window.



Urban Dispersion Option

Plume Depletion Due To (ISCST3 and ISC-PRIME Only)

Two depletion options are available, **Wet Removal** and **Dry Removal**. You can select either one, both, or none of these options. The **Wet Removal** and **Dry Removal** options may be used with the **Concentration**, the **Total Deposition (Dry and Wet)**, the **Dry Deposition**, or the **Wet Deposition** options.



- **Dry Removal:** Specifies that plume depletion due to dry removal mechanisms will be included in the calculations.

- **Wet Removal:** Specifies that plume depletion due to wet removal mechanisms will be included in the calculations.

Note: If the **Wet Removal** option is selected, than you MUST specify scavenging coefficients in the **SO – Gas & Particle Data** window.

Note: If the **Dry Removal** option is selected, than you MUST specify particle information in the **SO – Gas & Particle Data** window.

Run Option

The following options are available:

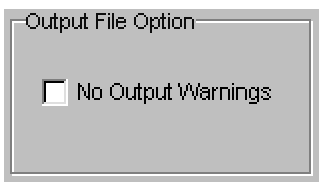
- **Run:** This option indicates that the model will read through all of the inputs in the runstream file (input file) regardless of any errors or warnings that may be encountered. If a fatal error occurs in the processing of the runstream information, then further model calculations will be aborted. To run the model, select **Run | Status** from the menu and press the **Run** button from the **Project Status** dialog box. You can also select the **Run | Run (ISCST3, AERMOD, or ISC-PRIME)** menu option.

- **Do Not Run:** This option indicates that the model will not run and will only process the input runstream data and summarize the setup information. Fatal error messages and warning messages will be issued if any errors are found in the input file. You should always check your input file by using this option. The **Do Not Run** option is available by selecting **Run | Status** from the menu and pressing the **Verify Run** button from the **Project Status** dialog box. You can also select the **Run | Verify Run (Do Not Run)** menu option.

Note: The **Do Not Run** option is used to overcome the potential for wasted resources if a large run is performed with some incorrect input data. With the ISC-AERMOD View interface, the number of possible incorrect input data in the input file is greatly reduced, since ISC-AERMOD View checks most of all the input data before you run the models. Any missing data is reported by ISC-AERMOD View in the **Details** dialog box.

Output File Option (AERMOD Only)

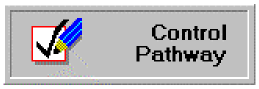
- **No Output Warning:** This option suppresses the detailed listing of warning messages in the main output file. The number of warning messages is still reported and warning messages are still included in the error listing file.



Comments (Optional)

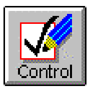
This option allows you to add any extra comments you feel are necessary to the Control Pathway portion of the input file.

CO - Pollutant/Avg Time/... Window



In the **CO - Pollutant/Avg Time/...** window you define the pollutant being modeled, the pollutant's exponential decay, the averaging time options, the terrain height options, the terrain elevation units, and the flagpole receptor options.

You have access to the **CO- Pollutant/Avg Time/...** window by pressing the **Control** button located on the menu toolbar and then pressing the **Pollutant/Avg Time/...** button located on the lower right side of any Control Pathway window. From the menu select **Data | Control | Pollutant/Avg Time/...**.



+


= Displays the **CO- Pollutant/Avg Time/...** window

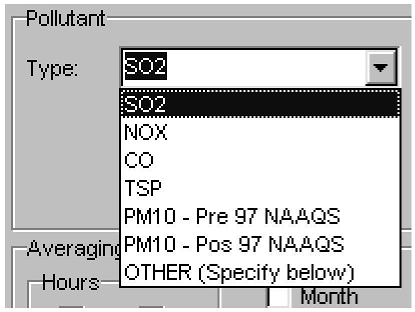
Specifying Pollutant, Averaging Time, and Terrain Options

The CO-Pollutant/Avg Time/…window contains options for specifying pollutant type, pollutant exponential decay, averaging time options, terrain options, and flagpole options. Each one of these options is described in the sections that follow.

Specifying the Pollutant Type

You must identify the type of pollutant being modeled for a particular run. The **Type** drop-down list box contains the names of the following pollutants:

- **SO_2:** Sulfur Dioxide
- **NOx:** Nitrogen Oxides
- **CO:** Carbon Monoxide
- **TSP:** Total Suspended Particulates
- **PM10-Pre 97 NAAQS:** Particle Matter – 10 Microns – Pre 97 National Ambient Air Quality Standard
- **PM10-Post 97 NAAQS (ISCST3 Only):** Particle Matter – 10 Microns – Post 97 National Ambient Air Quality Standard
- **OTHER:** This option allows you to specify a different name for the pollutant being modeled. In this case, the pollutant name, up to eight (8) characters long, can be input in the **Specify** text box. This text box is not available for use if **OTHER** is not selected from the drop-down list box.



The only pollutant choices that currently have any impact on the results are:

1. The selection of **SO_2** in conjunction with **Urban** dispersion and the **Regulatory Default** option. In this case, the model uses a **Half Life** of 4 hours for exponential decay.

2. The selection of **PM10-Pre 97 NAAQS** with the **Multiple Year Analyses** option for generating the high-sixth-high in five years.

3. The selection of **PM10-Pos 97 NAAQS** with the **24-Hour** and **Annual** averages and the **4th** highest value option for the new Post-1997 PM_{10} NAAQS analysis.

PM10 – Pos 97 NAAQS Processing

The revised ISCST3 Model (dated 98226) incorporates changes to the processing of multiyear averages for the new PM_{10} National Ambient Air Quality Standard (NAAQS), promulgated by EPA in July 1997. This change allows averaging of high-fourth-high values across a multiple year meteorological data file.

For the new Post 97 PM10 option (**PM10-Pos 97 NAAQS**), the model will compute an average of the 4^{th} highest concentrations at each receptor across the number of years of meteorological data being processed. If you are using a single year of met data, then the model will report the 4^{th} highest concentration at each receptor. If you are using a five-year period met data, the model will report the average of the five 4^{th} highest values at each receptor.

In ISC-AERMOD View, you can process the 24-hour and annual averages for PM_{10} according to the new NAAQS if the pollutant ID selected from the Pollutant **Type** drop-down list box was **PM10-Pos 97 NAAQS**. If **PM10-Pos 97 NAAQS** is selected, then ISC-AERMOD View automatically selects the following options for processing the new PM_{10} NAAQS:

- **Averaging Time Options:** 24-Hour and Annual (all other averaging time options will be disabled).

- **RECTABLE:** 4^{th} highest value (all other high values will be disabled).

- **Multi-Year Analyses:** This option will become disabled. Multiple year analyses cannot be used with the new PM_{10} NAAQS. Multiple year analyses should be accomplished by including the multiple years of met data in a single file.

The following restriction applies to the Post 1997 PM-10 processing:

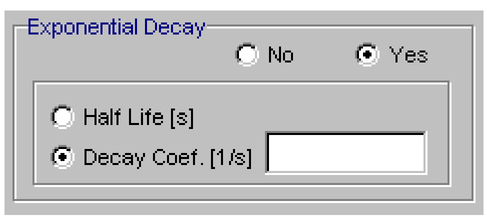
- **Met Data:** The model will only process complete years of meteorological data. If less than one complete year of data is processed, a fatal error message will be generated.

Exponential Decay (Optional)

You have the option to specify the exponential decay of the pollutant being modeled. Two options are available for this purpose, **Half Life [s]** and **Decay Coefficient [1/s]**. The **Half Life** option is used to specify the half life for exponential decay in seconds. The **Decay Coefficient** option is used to specify the decay coefficient in units of s^{-1}. The relationship between these parameters is:

Decay Coefficient = 0.693 / Half Life

To specify the pollutant exponential decay, select the **Yes** option button and then select one of the options, **Half Life** or **Decay Coef.**. Only one of these options may be specified. As soon as you select the appropriate option button, a text box for value input appears beside the selected option.



Exponential Decay fame

Note: The **Regulatory Default** option includes a **Half Life = 4 hrs (345,600 s)** for exponential decay of $\mathbf{SO_2}$ in **Urban** settings. If these options were selected, then ISC-AERMOD View automatically inputs the 4 hrs half life or corresponding decay coefficient in the text boxes.

Averaging Time Options

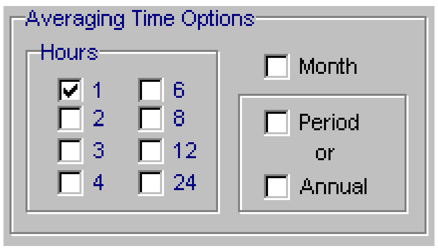
The following averaging time options are available:

- **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 data period.

- **Annual (ISCST3 and ISC PRIME Only):** refers to the average for the entire data period.

You can select for a given run up to the maximum number of short-term averages set in the model storage limits by the NAVE parameter. The NAVE parameter defines the maximum number of short-term averaging periods allowed for each model. In the ISC-PRIME and AERMOD original U.S. EPA models, NAVE is set equal to 4. The ISCST3 model does not have a limit (memory allocation). In ISC-AERMOD View, storage limits are defined in the **Preferences** dialog box (see Chapter 10). The **Period** average and the **Annual** average do not count towards the NAVE limit.



Averaging Time Options

Note: For **Concentration** calculations, the **Period** and **Annual** averaging time options produce the same results. Either one may be used to calculate the annual average for a full year of meteorological data, or to calculate the period average for a period other than a year.

Note: For **Deposition** calculations, the **Period** option will provide a total deposition flux for the full period of meteorological data that is modeled in units of g/m^2, including multiple-year data files. The **Annual** option will provide an annualized rate of the deposition flux in units of $g/m^2/yr$. 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 of 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.

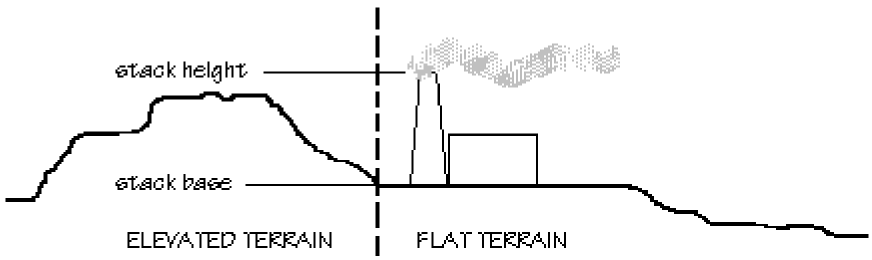
Terrain Height Options

Two terrain height options are available, **Flat** or **Elevated** terrain.

- **Flat:** Terrain height is assumed not to exceed stack base elevation (terrain height assumed to be 0.0 m). If this option is selected, then the ISCST3 model will assume

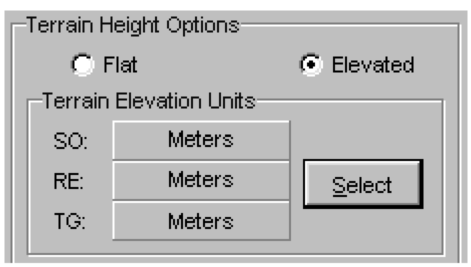
Flat terrain calculations to be used throughout, regardless of the input of **Terrain Elevations** in the Receptor Pathway. Any **Terrain Elevations** that are entered in the Receptor Pathway are ignored if the **Flat** terrain option is selected.

- **Elevated:** Terrain height exceeds stack base elevation. If this option is selected, then **Terrain Elevations** are allowed/expected on the Receptor Pathway. If **Elevated** terrain is selected and a receptor height is not specified, then it is assumed to have a value of 0.0 meters.

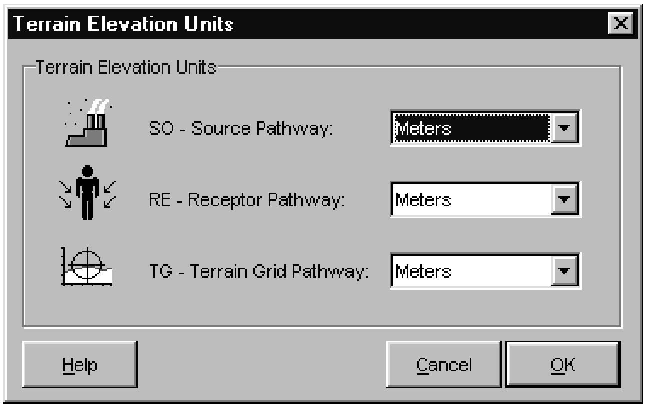


Terrain Elevation Units

If you select **Elevated** for the terrain height option, then you have the choice of selecting the units for the **Terrain Elevation** data that will be input in the Source (**SO**), Receptor (**RE**), and Terrain Grid (**TG**) Pathways.



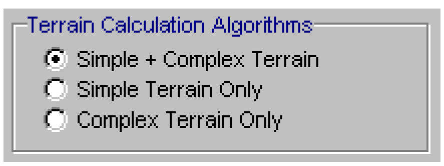
To select terrain elevation units, click on the **Select** button to display the **Terrain Elevation Units** dialog box. Select **Meters** or **Feet** from the drop-down list box for each Pathway and click the **OK** button. The selected units will be displayed on the **SO**, **RE**, and **TG** panels respectively.



Terrain Elevation Units dialog box

Terrain Calculation Algorithms (ISCST3 & ISC PRIME Only)

Depending on the **Terrain Height Option** (**Flat** or **Elevated**) that you have selected, the following terrain calculation algorithms will be available:



- **Simple + Complex Terrain:** This is the default option and it is available for both, the **Flat** and the **Elevated**, terrain height options. This option instructs the model to implement both simple and complex terrain algorithms and also apply 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, i.e., terrain between release height and plume height.

- **Simple Terrain Only:** This option is available for both, the **Flat** and the **Elevated** terrain height options. This option specifies that no complex terrain calculations will be made, i.e., uses ISCST algorithms only. You should not use this option if you are modeling with complex terrain (terrain above the release height of the source). Read **Note** below.

- **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, i.e., uses COMPLEX1 algorithms only.

Note: For terrain above the release height of the source (i.e., complex terrain), the model automatically truncates ("chops") the terrain to the physical release height(s) when modeling impacts at those receptors using the simple terrain (ISC) algorithm (**Simple Terrain Only** option). Terrain above the release height is not truncated when the COMPLEX1 algorithm is used in ISCST (**Complex Terrain Only** option).

Flagpole Receptors (Optional)

The **Flagpole Receptors** option allows you to specify receptor heights above local ground level.

Flagpole Receptors frame

- **No:** This option indicates that flagpole receptors are not going to be specified in the Receptor Pathway. This is the default option, and assumes a default receptor height of 0.0 m.

- **Yes:** This option indicates that flagpole receptors are allowed in the Receptor Pathway. If this option is selected, a text box will be available for the input of a default height other than 0.0 m. If no inputs are made, then the default flagpole receptor height of 0.0 m is used.

Note: Any **Flagpole Heights** that are entered in the Receptor Pathway are ignored if the **No** option was selected.

CO - Optional Files Window

In the **CO-Optional Files** window you can define files to be generated for specific purposes. The following optional options are available, the EVENT input file, the Re-Start File, the Initialization File, the Multiple Year Analyses, and the Error Listing File, etc.

You have access to the **CO-Optional Files** window by pressing the **Control** button located on the menu toolbar and then pressing the **Optional Files** button located on the lower right side of any Control Pathway window. From the menu select **<u>D</u>ata | <u>C</u>ontrol | <u>O</u>ptional Files**.

+

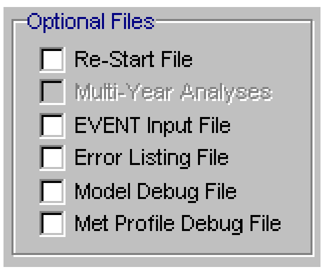

= Displays the **CO- Optional Files** window

Specifying Optional Files

The **CO - Optional Files** window contains options for specifying the following optional files:

- Re-Start Files,
- Multiple Year Analyses Files,
- EVENT Input File,
- Error Listing File,
- Model Debug File (AERMOD Only), and
- Met Profile Debug File (AERMOD Only).

You can specify any one of the above optional files by checking the **Optional Files** check boxes. Note that as you check each box, the corresponding frame containing the inputs and options for that optional file becomes available for use.



Optional Files check boxes

The following buttons are common to all optional files:

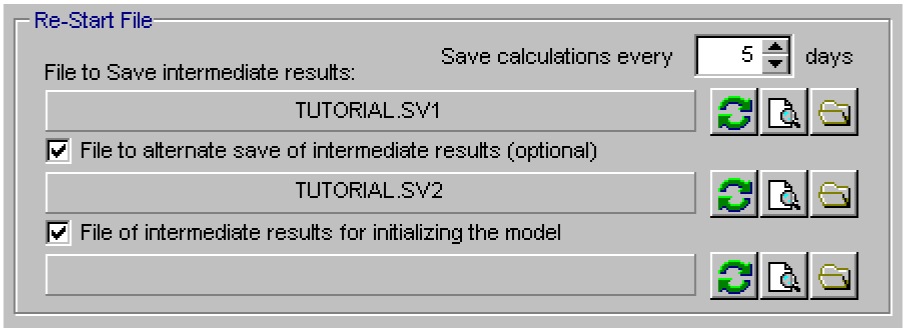
File button: This button displays a dialog box from where you can specify the name and path for the requested file.

Preview button: This button allows you to preview your file, if it exists, in Windows WordPad.

Default button: By pressing this button you get the default filename defined by ISC-AERMOD View.

Re-Start File (Optional)

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. The following are the necessary inputs for this option:



Re-Start File option

- **Save calculations every ... 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. ISC-AERMOD View uses 5 days as the default.

- **File to save intermediate results (Optional):** Here you specify the filename for saving intermediate results. The default filename `ProjectName.sv1` will be used if you do not specify a different filename.

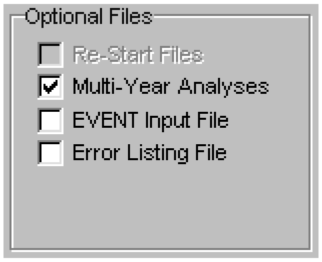
- **File to alternate save of intermediate results (Optional):** This option allows you to specify a file for the model to alternate the save of intermediate results. To make this option available, check the box. The default filename for this option is `ProjectName.sv2`.

Note: The advantage of this option is that the model will alternate between the two files for storing intermediate results. This approach however, requires additional disk space but avoids loosing the file in case of a power failure or similar event while the file is being overwritten.

- **File of intermediate results for initializing the model (Optional):** This file instructs the model to initialize the results arrays from a previously saved file. If this option is selected, then you need to specify the filename of the unformatted file of intermediate results to be used for initializing the model.

Multiple Year Analyses for PM-10 (Optional)

This option allows you to perform a multi-year analysis needed to determine the "high-sixth-high in five years" design value for determining PM-10 impacts. 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.

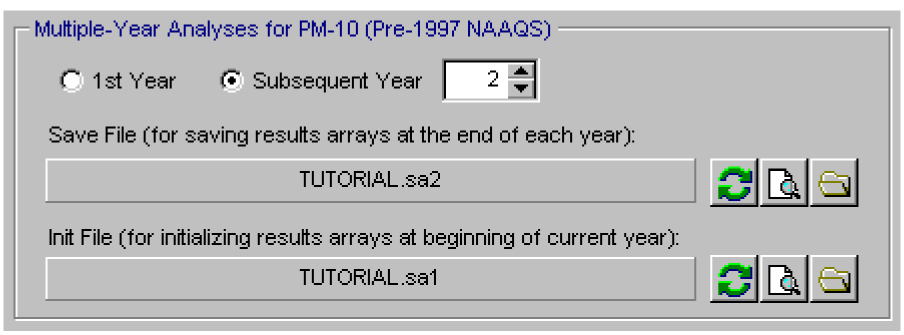


ISC-AERMOD View will automatically disable the **Re-Start File** option if you select the **PM10 - Pre 97 NAAQS** option from the **Pollutant** drop-down list box. If **OTHER** is selected from the **Pollutant** drop-down list box, then both options become available, however only one of them can be selected.

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.

For the **Multiple Year Analyses** option, the following inputs are necessary:



Multiple-Year Analyses for PM10 (Pre 97 NAAQS)

- **1st Year:** Select this option if you are performing multiple year analysis for the first (1st) year. If the **1st Year** option is selected, then only the **Save File** must be specified.

- **Subsequent Years:** Select this option if you are performing multiple year analysis for years other than the first (1st) year. If the **Subsequent Year** option is selected, then both filenames, 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 **1st year** in the multi-year series of runs, ISC-AERMOD View uses a default filename *ProjectName*.sa1. For the **Subsequent Years**, ISC-AERMOD View uses default filenames as shown in Table 3-3 below.

- **Initialization File:** This is the file for initializing the result arrays at the beginning of the current year. This file option is not available for the **1st year** in the multi-year series of runs. For the **Subsequent Years** option, you should specify here the file used as the **Save File** in the previous year run (see Table 3-3).

See Table 3-3 below for an example of how to setup the **Multiple Year Analyses** option for each run:

Table 3-3: Setup for the Multiple Year Analyses series of runs.

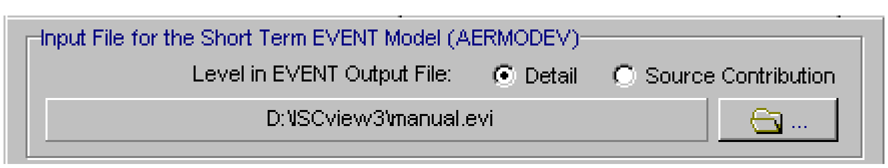
Year	Save File	Init File
First Year	*ProjectName*.sa1	---------
Second Year	*ProjectName*.sa2	*ProjectName*.sa1
Third Year	*ProjectName*.sa3	*ProjectName*.sa2
Fourth Year	*ProjectName*.sa4	*ProjectName*.sa3
Fifth Year	*ProjectName*.sa5	*ProjectName*.sa4

Note: For the **Subsequent Years** option, you should save the ISC-AERMOD View input file that you setup for the previous year with a different filename (select **File|Save As** from the menu). Change the necessary information such as the year parameters and meteorology filename on the ME Pathway, the title of the project (if wanted), and setup the inputs in the **Multiple Year Analyses** option for the year being analyzed (see Table 3-3 above).

Note: To obtain the **PM-10** design value, be sure to include the **6TH** (sixth) highest value for **RECTABLE** option in the **OU-Tabular Outputs** window.

Input File for the Short Term EVENT Model (Optional)

If the **EVENT Input File** option is selected, then the model will generate an input file for use with the EVENT model. The EVENT model is designed to provide source contributions to particular events, such as the design concentrations determined from the ISCST3 (or AERMOD) model, or user specified events. The following should be specified if this option is selected:



Input File for the Short Term EVENT Model (ISCEV or AERMODEV) option

- **Level in EVENT Output File:** Specifies the level of detail to be used in the EVENT output file. Two options are available, **Detail** (the default) or **Source Contribution**.

- **Filename:** Name of the EVENT input file to be generated. The default filename *`ProjectName.evi`* will be used if you do not specify a different filename. To specify a filename, click on the **[File]** button to display the **Specify File** dialog box, input the full path and name of the file to be created, and click on the **OK** button. The filename you specified will be displayed on the panel.

Note: 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 **CO-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.

Note: The U.S. EPA AERMOD EVENT model and ISC-PRIME EVENT model were not available at the time this User's Guide was written.

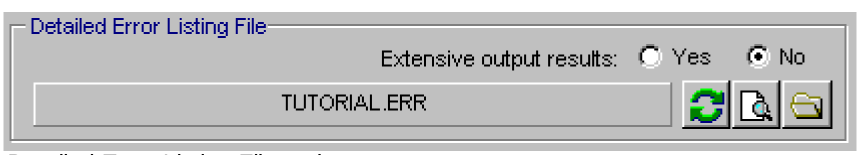
Detailed Error Listing File (Optional)

This option allows you to request a detailed listing file of all messages generated by the model. This 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.



Detailed Error Listing File option

The following should be specified if the **Detailed Error Listing File** option is selected:

- **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.

- **Filename:** You can specify a filename for the detailed error listing file or use the default filename *`ProjectName.err.`*

CAUTION ! : 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. ISC-AERMOD View gives you a warning message every time you select this option.

Model Debugging Output File (AERMOD Only)

This option allows you to request a detailed file of intermediate calculation results for debugging purposes. The intermediate calculations are related to the model results for each source and receptor such as dispersion parameters, plume heights, etc. You can specify a filename for the model debugging output file or use the default filename *`ProjectName.dbl.`*

Model Debugging Output File option

CAUTION !: The **Model Debugging Output File** option can produce a very large file.

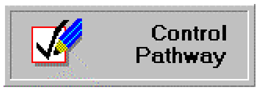
Meteorological Profile Debugging Output File (AERMOD Only)

This option allows you to request a detailed file with gridded profiles of meteorological variables for each hour of data for debugging purposes. You can specify a filename for the meteorological profile debugging output file or use the default filename *`ProjectName.dbp.`*

Meteorological Profile Debugging File option

CAUTION !: The **Meteorological Profile Debugging Output File** option can produce a very large file.

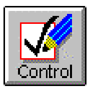
CO – Air Toxics Options Window



(ISCST3 Only)

In the **CO–Air Toxics Options** window you can define non-default options for air toxics applications. These options are only available for the ISCST3 model.

If you are in another Pathway window, press the **Control** toolbar button. Select the **Non-Default Options** radio button and then check the **Air Toxics Options** box and then press the **Air Toxics Options** button located on the lower right side of any Control Pathway window. From the menu select **Data | Control | Air Toxics Options**.

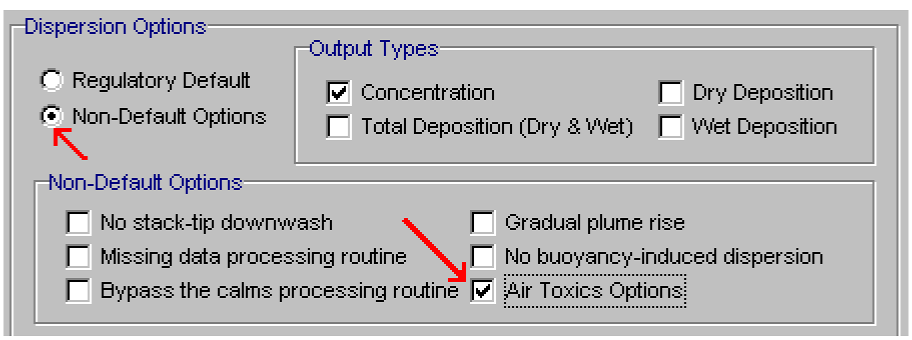


+


= Displays the **CO- Air Toxics Options** window

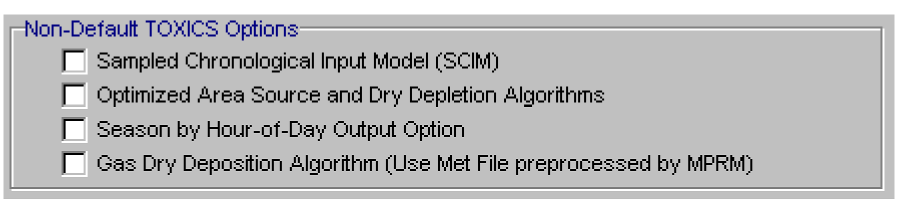
Specifying Air Toxics Options (ISCST3 Only)

Air Toxics options are non-default dispersion options and therefore they are only available if the **Non-Default Options** and the **Air Toxics Options** were selected for your modeling project.



The Non-Default TOXICS Options are as follows:

- Sampled Chronological Input Model (SCIM)
- Optimized Area Source and Dry Depletion Algorithms
- Season by Hour-of-Day Output Option
- Gas Dry Deposition Algorithm



Non-Default TOXICS Options

See the description of each one of these options in the sections that follow:

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 (i.e., ANNUAL) average impacts. The SCIM option has the following restrictions:

- Can only be used with the **ANNUAL** average option.
- 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.
- User must specify the SCIM sampling parameters on the **ME–SCIM Sampling** window. This option will be explained in Chapter 6 - Meteorology Pathway.

The SCIM sampling parameters, in the Meteorology Pathway, specify the starting hour and sampling interval for the regular or dry sample and for the wet sample (if used). The ISCST3 model uses the following approach depending on the type of sampling parameters specified:

Only Regular Sampling Selected:

All hourly impacts (concentration, dry deposition flux, and wet deposition flux) are calculated in the normal fashion for each sampled hour. See below:

- **Annual Average Concentration:** Calculated by dividing the cumulative concentration for the sampled hours by the number of hours sampled (arithmetic average).

  $$\mathbf{C = C_s / N_s}$$

  Where:

  C = Calculated Concentration
  C_s = Cumulative impacts for the sampled hours
  N_s = Number of sampled hours

- **Annual Dry and Wet Deposition Fluxes:** calculated by scaling the respective cumulative fluxes for the sampled hours by the ratio of the total hours to the sampled hours.

  $$\mathbf{D = D_s \ (N_t/N_s)}$$
  $$\mathbf{W = W_s \ (N_t/N_s)}$$

  Where:

D = Calculated Dry Deposition Fluxes
W = Calculated Wet Deposition Fluxes
D_s = Cumulative Dry Deposition impacts for the sampled hours
W_s = Cumulative Wet Deposition impacts for the sampled hours
N_s = Number of sampled hours
N_t = Total number of hours in the data period

Regular Sampling + Wet Sampling Selected:

When the wet hour sampling is also selected along with the regular (or dry) sampling, then the following approach is followed:

- **Annual Average Concentration and Dry Deposition Fluxes:** are based on the weighted contributions from regular samples, modeled as dry hours and wet hour samples. The regular samples consist of all the hours based on a regular sampling interval, but the effects of precipitation are ignored so that their contribution represents only dry conditions, while the contribution from the wet hour samples represents only wet conditions.

- **Annual Wet Deposition Fluxes:** are only based on the wet hour samples.

Optimized Area Source and Dry Depletion Algorithms

This optimization option is available to reduce model runtime. 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 Removal** option for non-area sources is automatically selected.

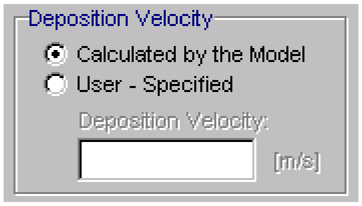
Season by Hour-of-Day Output Option

When selecting this non-default option, the user may request an output file containing the average results (Concentration, Total Deposition, Dry Deposition, and/or Wet Deposition) by season and hour-of-day. To select this option you must specify the required parameters in the **OU–Season Hour-of-Day Files** window, located in the Output Pathway. See Chapter 8 – Output Pathway for a detailed description of the Season by Hour-of-Day Output option.

Gas Dry Deposition Algorithm

This non-default option allows you to model the effects of dry deposition for gaseous pollutants. The Gas Dry Deposition Algorithm option requests that the following parameters be specified:

- **Deposition Velocity:** You are requested to either specify a deposition velocity in meters per second or allow the model to calculate the deposition velocities.



1. **Deposition Velocities Calculated by the Model:** If this option is selected, then you also need to specify the State of Vegetation, the Reference Parameters, Source Parameters, and the Meteorological Data File with the additional meteorological variables.

2. **User-Specified Deposition Velocity:** 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.

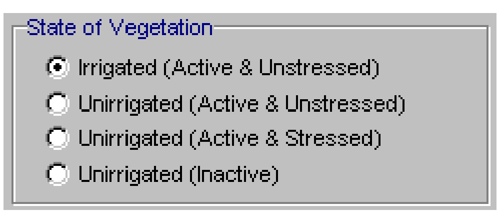
Note: A non-fatal warning message is generated if a value greater than 0.05 m/s is input for the deposition velocity.

- **State of Vegetation:** 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:

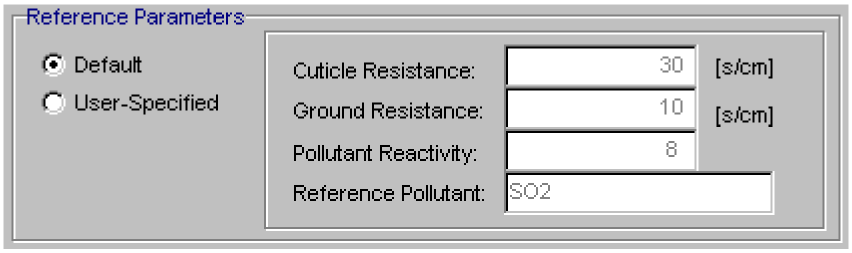
  1. Irrigated (Active & Unstressed)
  2. Unirrigated (Active & Unstressed)
  3. Unirrigated (Active & Stressed)
  4. Unirrigated (Inactive)

  For unirrigated vegetation, you should select the appropriate option based on existing soil moisture conditions.



- **Reference Parameters:** You can override the default reference parameters for use with the gas dry deposition algorithm. These reference parameters are:

  Cuticle resistance = 30 s/cm
  Ground resistance = 10 s/cm
  Pollutant reactivity = 8
  Name of the reference pollutant (optional) = SO2

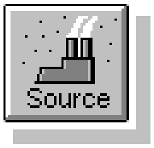


- **Source Parameters for Gas Dry Deposition:** You are requested to specify source parameters for the gas dry deposition option only if the deposition velocities are being calculated by the model. These source parameters must be specified in **the SO-Gas & Particle Data** window in the Source Pathway. See Chapter 4 – Source Pathway for a detailed description of these parameters.

- **Meteorological Data File 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. See Chapter 6 for a complete description of the meteorological data file format that is used with the Gas Dry Deposition Algorithm option.

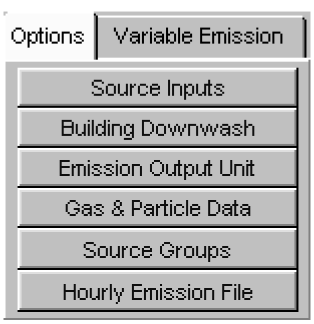
CHAPTER 4

Source Pathway

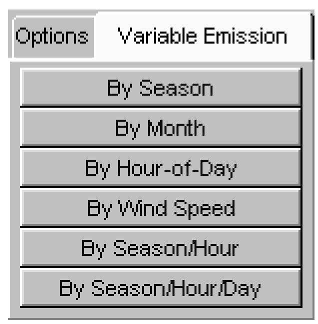


This chapter explains the ISC-AERMOD View inputs and options in the Source Pathway (SO). ISC-AERMOD View currently handles eight source types, identified as: point, volume, area, open pit, polygon area, circular area, flare, and line sources.

Contents



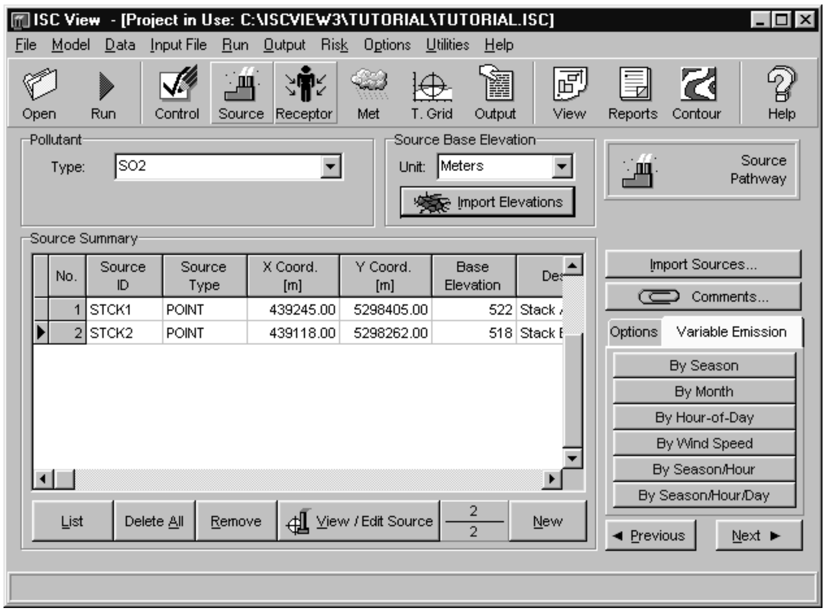
Options Tab



Variable Emission Tab

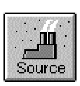
SO - Source Inputs Window

The SO-Source Inputs window gives you information on the number of sources specified for your current project, pollutant information, source base elevation unit selection, and import of sources base elevations from digital elevation model files. It is also from this window that you can specify all sources for your project.



SO-Source Inputs window

You have access to the **SO-Source Inputs** window by pressing the **Source** button located on the menu toolbar. If you are in any Source Pathway window, press the **Source Inputs** button located on the lower right side of any Source (SO) Pathway window. From the menu select **Data | Source | Source Inputs**.



or


= Displays the **SO-Source Inputs** window

Defining Sources Parameters

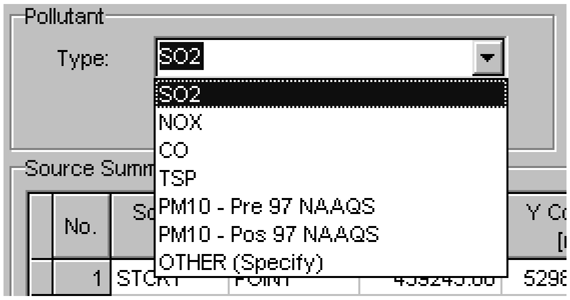
The following options are available on the **SO-Source Inputs** window:

- Pollutant
- Source Base Elevation
- Source Summary

See the description of each one of these options in the sections that follow.

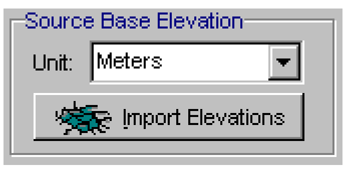
Pollutant

For each run, you need to specify the type of pollutant being modeled. This option is the same as the one described in the Control Pathway in the **CO – Pollutant / Avg Time/…** window. Changes in the pollutant type done in the Source Pathway will be automatically displayed in the Control Pathway and vice-versa.

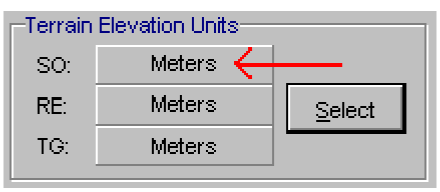


Source Base Elevation – Unit and Import

The **Source Base Elevation** frame contains two options:



- **Unit:** This option indicates in which unit (meters or feet) the source base elevation is being specified. This option is also used in the Control Pathway in the **CO – Pollutant / Avg Time/…** window. Any changes made in the Control Pathway for the SO Terrain Elevation Unit will be reflected in the Source Pathway and vice-versa.



Terrain Elevation Units in Control Pathway

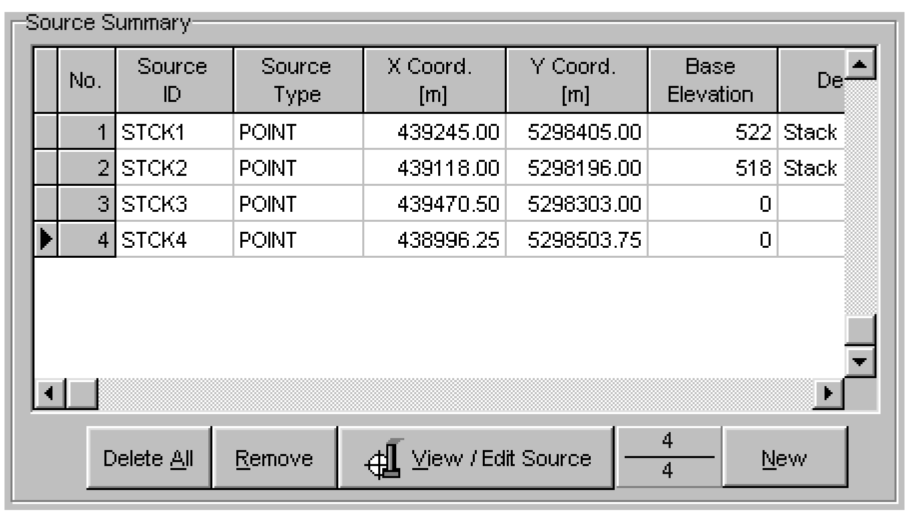
- **Import Elevations:** You can import the base elevation for all sources by pressing the **Import Elevations** button. The elevations will be imported from the digital elevation model files that you have specified (e.g., USGS DEMs, or UK NTFs, or UK DTMs, or XYZ files).

Source Summary

The **Source Summary** table displays a list of all the sources specified for the current project. To add, delete, edit, or view a source can be done easily by using the following buttons:

- **Delete All:** Use this button to delete all sources at once.
- **Remove:** Use this button to delete the selected sources. You can press down the Shift key to select more than one source at once, or the Alt + Ctrl keys to select alternate sources.
- **View / Edit Source:** Use this button to view or edit an existing source. The source being currently highlighted or the last source from a multiple selection of sources will be displayed first. You can also double-click on a specific source row to view or edit the parameters for the source.
- **Current Record/Record Count:** The top panel displays the record number for the source being currently highlighted or the record number for the last source from a multiple selection of sources. The bottom panel displays the total number of sources in the table.
- **New:** Use this button to add a new source.

The **Source Summary** table consists of 7 columns. These columns are as follow:



No.	Source ID	Source Type	X Coord. [m]	Y Coord. [m]	Base Elevation	De
1	STCK1	POINT	439245.00	5298405.00	522	Stack
2	STCK2	POINT	439118.00	5298196.00	518	Stack
3	STCK3	POINT	439470.50	5298303.00	0	
4	STCK4	POINT	438996.25	5298503.75	0	

Source Summary table

- **Column 1 - No:** Every time a new source is created, ISC-AERMOD View automatically creates an entry number for the new source.

- **Column 2 – Source ID:** This column displays the ID for the source.

- **Column 3 – Source Type:** This column displays the type of source (e.g., point, area, volume, etc.)

- **Column 4 – X Coord:** This column displays the X Coordinate for the source. The X (east-west) coordinate may be input as UTM (Universal Transverse Mercator) or may be referenced to a user-defined origin. The location for the X Coordinate for each source type is given in Table 4-1. Please note that for AREA POLY and LINE sources this coordinate is for the first vertex or point input by the user.

- **Column 5 – Y Coord:** This column displays the Y Coordinate for the source. The Y (north-south) coordinate may be input as UTM (Universal Transverse Mercator) or may be referenced to a user-defined origin. The location for the Y Coordinate for each source type is given in Table 4-1. Please note that for AREA POLY and LINE sources this coordinate is for the first vertex or point input by the user.

Table 4-1: Location for the X and Y Coordinates

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

- **Column 6 – Base Elevation:** This column displays base elevations for the sources. The base elevation can either be input in meters or feet. If the base elevation for one source is input in meters, for example, then the base elevation for all other sources must also be input in meters. You select the base elevation unit in the **Source Base Elevation** frame, which was explained above.

- **Column 7 – Description (Optional):** This column displays the description for the source (up to 68 characters long).

Defining Sources Parameters in the Source Inputs Dialog Box

If you are using the text mode, you can specify all your sources by using the **Source Inputs** dialog box

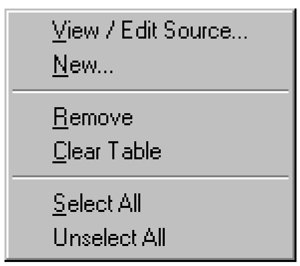
➤ **How to display the Source Inputs dialog box:**

You can display the **Source Inputs** dialog box in the following ways:

1. Click on the **View / Edit Source** button located on the bottom of the **Source Summary** table or

2. Double-click on the **Source Summary** table.

3. Right-Click on the **Source Summary** table to display the pop-up menu. Select the **View / Edit Source** option.



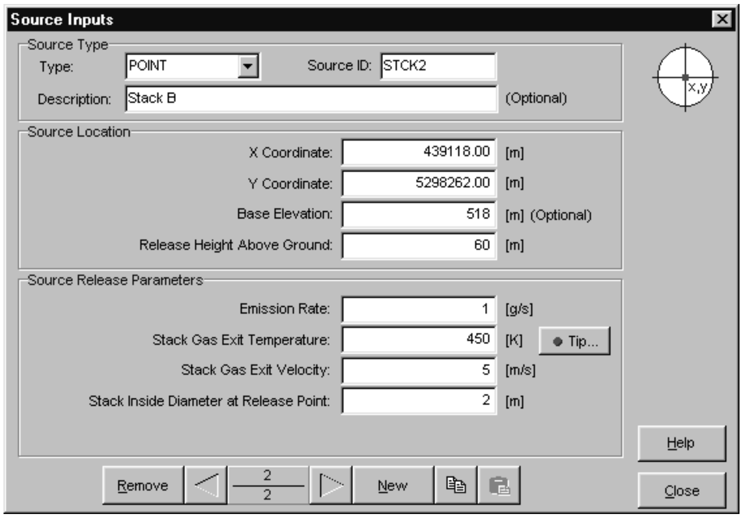
The parameters requested in the **Source Inputs** dialog box depend on the type of source. ISC-AERMOD View currently supports the following source types:

- POINT
- VOLUME
- AREA
- OPEN PIT
- CIRCULAR AREA
- POLYGON AREA
- FLARE
- LINE

See the next sections for the parameters requested for each one of these source types.

POINT Source Parameters

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:



Source Inputs dialog box – Point Source Parameters

Source Type

- **Type:** Select **POINT** from the drop-down list box.
- **Source ID:** Enter here an identification name for the source being defined. The ID can be up to 8 characters long. Source IDs are always in upper case.
- **Description (Optional):** Enter here any description for the source up to 68 characters long.

Source Location

- **X Coordinate [m]:** Enter here the x (east-west) coordinate for the source location in meters (center of the point source).
- **Y Coordinate [m]:** Enter here the y (north-south) coordinate for the source location in meters (center of the point source).
- **Base Elevation [m or ft]:** Enter here the source base elevation. The model only uses the source base elevation if **Elevated** terrain is being used. The default unit is meters. You can also specify source base elevations in feet by changing the **Unit** option in the **SO-Source Inputs** window to Feet.
- **Release Height above Ground [m]:** Enter the release height above the ground in meters.

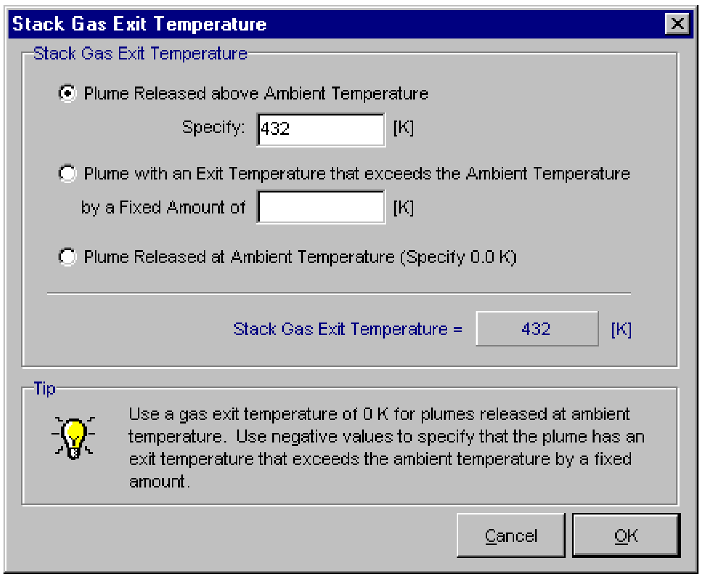
Note: The X and Y Coordinates for the source location may be input as Universal Transverse Mercator (UTM) coordinates, or may be referenced to a user-defined origin.

Source Release Parameters

- **Emission Rate [g/s]:** Enter the emission rate of the pollutant in grams per second. The same emission rate is used for both concentration and deposition calculations.

- **Stack Gas Exit Temperature [K]:** Enter the temperature of the released gas in degrees Kelvin.

The **Tip** button will give you tips on the many ways the EPA model interprets values entered for this parameter. The following dialog box is displayed when the **Tip** button is pressed:



Stack Gas Exit Temperature dialog box

Three options are available for the Stack Gas Exit Temperature input:

1. **Plume Released above Ambient Temperature:** Use this option to specify the plume exit temperature that is above ambient temperature.

2. **Plume with an Exit Temperature that exceeds the Ambient Temperature by a Fixed Amount:** You can model a plume with an exit temperature that exceeds the ambient temperature by a fixed amount by entering a negative value for the exit temperature equal in magnitude to the temperature difference. The EPA model will add the absolute value of a negative exit temperature to the ambient temperature for each hour to obtain the exit temperature used in computing the buoyancy flux of the plume.

3. **Plume Released at Ambient Temperature:** If a value of 0.0 is input for the exit temperature, the EPA model will adjust the exit temperature for each hour to match the ambient temperature. This option allows the user to model a plume that is released at ambient temperature.

- **Stack Gas Exit Velocity [m/s]:** Enter the stack gas exit velocity in meters per second. The exit velocity can be determined from the following formula:

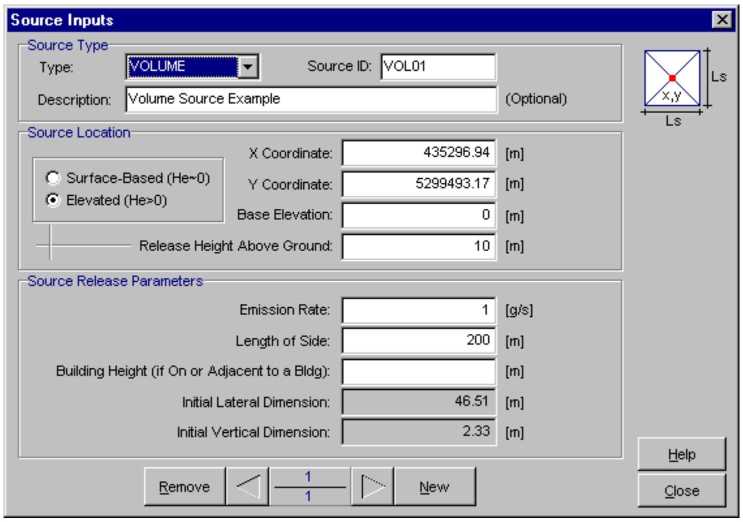
$$V_s = \frac{4V}{\pi d_s^2}$$

V_s = Exit Velocity
V = Flow Rate
d_s = Stack Inside Diameter

- **Stack Inside Diameter [m]:** Enter the stack inside diameter in meters.

VOLUME Source Parameters

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:



Source Inputs dialog box – Volume Source Parameters

Source Type

- **Type:** Select **VOLUME** from the drop-down list box.
- **Source ID:** Enter here an identification name for the source being defined. The ID can be up to 8 characters long. Source IDs are always in upper case.
- **Description (Optional):** Enter here any description for the source up to 68 characters long.

Source Location

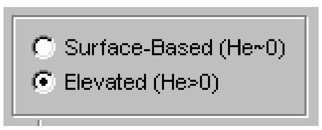
- **X Coordinate [m]:** Enter here the x (east-west) coordinate for the source location in meters. This location is the center of the volume source.

- **Y Coordinate [m]:** Enter here the y (north-south) coordinate for the source location in meters. This location is the center of the volume source.

- **Base Elevation [m or ft]:** Enter here the source base elevation. The model only uses the source base elevation if **Elevated** terrain is being used. The default unit is meters. You can also specify source base elevations in feet by changing the **Unit** option in the **SO-Source Inputs** window to Feet.

- **Release Height above Ground [m]:** Enter the release height above ground in meters (center of volume).

- **Type of Volume Source:** You should specify what type of volume source applies, surface-based or elevated. The selection of one option or the other will determine how the Initial Vertical Dimension of the source will be calculated.



1. **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 the procedure used to estimate the Initial Vertical Dimension of the source, if this option is selected.

   Initial Vertical Dimension = Release Height above Ground / 2.15

2. **Elevated (He > 0):** Select this option if the effective emission height (Release Height above Ground) is greater than zero. An example of an elevated 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 = Release Height above Ground / 4.3

<u>Source Release Parameters</u>

- **Emission Rate [g/s]:** Enter the emission rate of the pollutant in grams per second. 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. The volume source cannot be rotated and has the X side equal to the Y side (square).

- **Building Height (If On or Adjacent to a Bldg):** If your volume source is Elevated and 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.

- **Initial Lateral Dimension [m]:** This parameter is automatically calculated by ISC-AERMOD View. See Table 4-2 below for guidance on determining initial dimensions.

- **Initial Vertical Dimension [m]:** This parameter is automatically calculated by ISC-AERMOD View. See Table 4-2 below for guidance on determining initial dimensions.

Table 4-2: Summary of Suggested Procedures for Estimating Initial Lateral Dimension (σ_{yo}) and Initial Vertical Dimension (σ_{zo}) for Volume and Line Sources

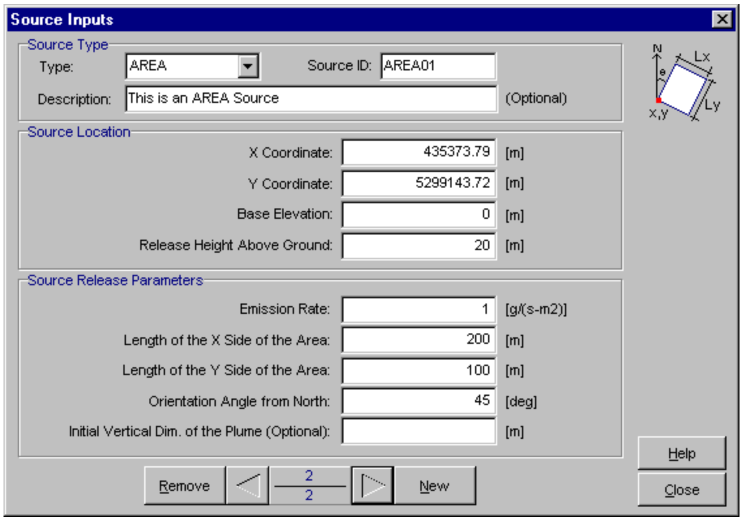
Type of Source	Procedure for Obtaining Initial Dimension
Initial Lateral Dimension	
Single Volume Source	$\sigma_{yo} = \frac{\text{side length}}{4.3}$
Line Source Represented by Adjacent Volume Sources	$\sigma_{yo} = \frac{\text{side length}}{2.15}$
Line Source Represented by Separated Volume Sources	$\sigma_{yo} = \frac{\text{center to center distance}}{2.15}$
Initial Vertical Dimension	
Surface-Based Source ($h_e \sim 0$)	$\sigma_{zo} = \frac{\text{vertical dimension of source}}{2.15}$
Elevated Source ($h_e > 0$) on or Adjacent to a Building	$\sigma_{zo} = \frac{\text{building height}}{2.15}$
Elevated Source ($h_e > 0$) not on or Adjacent to a Building	$\sigma_{zo} = \frac{\text{vertical dimension of source}}{4.3}$

Source: 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.

AREA Source Parameters

AREA sources are used to model low level or ground level releases with no plume rise (e.g., storage piles, slag dumps, and lagoons). The U.S. EPA models (ISCST3, AERMOD, and ISC-PRIME) 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:



Source Inputs dialog box – Area Source Parameters

Source Type

- **Type:** Select **AREA** from the drop-down list box.
- **Source ID:** Enter here an identification name for the source being defined. The ID can be up to 8 characters long. Source IDs are always in upper case.
- **Description (Optional):** Enter here any description for the source up to 68 characters long.

Source Location

- **X Coordinate [m]:** Enter here the x (east-west) coordinate for the vertex (corner) of the area source that occurs in the southwest quadrant of the source.
- **Y Coordinate [m]:** Enter here the y (north-south) coordinate for the vertex (corner) of the area source that occurs in the southwest quadrant of the source.
- **Base Elevation [m or ft]:** Enter here the source base elevation. The model only uses the source base elevation if **Elevated** terrain is being used. The default unit is meters. You can also specify source base elevations in feet by changing the **Unit** option in the **SO-Source Inputs** window to Feet.

- **Release Height above Ground [m]:** Enter the release height above ground in meters.

<u>Source Release Parameters</u>

- **Emission Rate [g/(s-m^2)]:** Enter 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.

- **Length of the X Side of the Area [m]:** If the Angle is 0 degrees, then this is the length in meters of the side of the Area source that is in the east-west direction. If Angle does 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 4-1 for illustration of this parameter.

- **Length of the Y Side of the Area [m] (Optional):** If the Angle is 0 degrees, then this is the length in meters of side of the Area source that is in the north-south direction. If Angle does 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 4-1 for illustration of this parameter. If this parameter is omitted, then the model assumes that the area is a square.

Note: The aspect ratio, **Length/Width**, for area sources should be less than 10 to 1. If this is exceeded, then the area should be divided to achieve a 10 to 1 aspect ratio (or less) for all sub-areas.

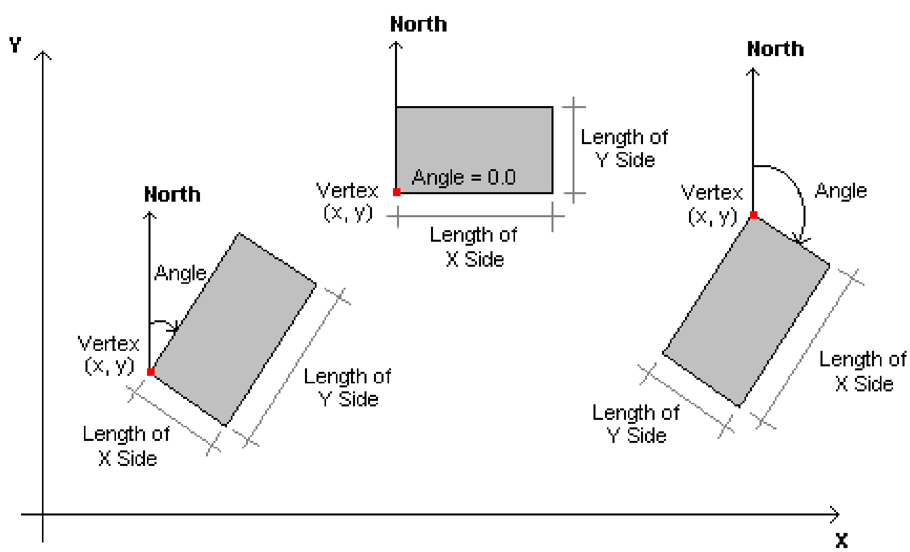


Figure 4-1. Relationship of area source parameters. Based on Figure 3-1 of the U.S. Environmental Protection Agency, 1995 User's Guide for the Industrial Source

Complex (ISC3) Dispersion Models - Volume I, EPA-454/B-95-003a. U.S. Environmental Protection Agency. Research Triangle Park, NC 27711.

- **Orientation Angle from North [deg] (Optional):** 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 Coordinate** and **Y Coordinate** location), i.e. the side with Y Side length. If this optional 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 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 angle is between -180 degrees and 180 degrees.

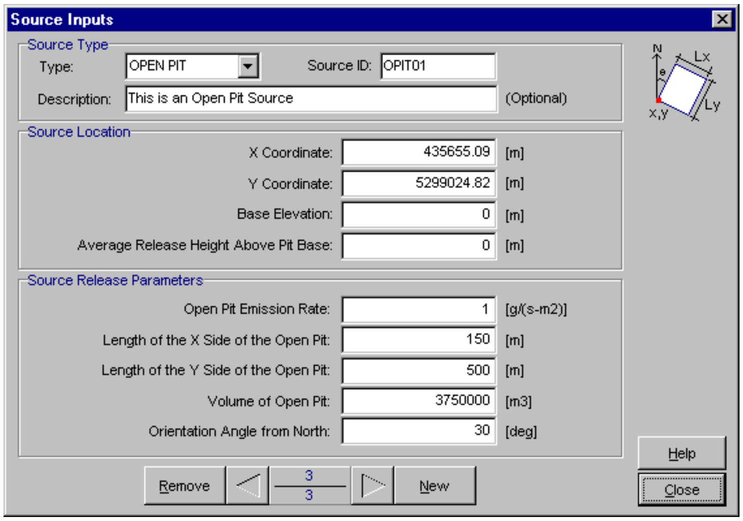
- **Initial Vertical Dim. of the Plume [m] (Optional):** This optional parameter may be used to specify an initial vertical dimension of the area source plume in meters. This parameter is similar to the **Initial Vertical Dimension** parameter for **VOLUME** sources. This parameter may be important when 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.

Note: 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 Source Inputs (ISCST3 and ISC-PRIME Only)

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 ISCST3 model accepts 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:



Source Inputs dialog box – Open Pit Source Parameters

Source Type

- **Type:** Select **OPEN PIT** from the drop-down list box.
- **Source ID:** Enter here an identification name for the source being defined. The ID can be up to 8 characters long. Source IDs are always in upper case.
- **Description (Optional):** Enter here any description for the source up to 68 characters long.

Source Location

- **X Coordinate [m]:** Enter here the x (east-west) coordinate for the vertex (corner) of the open pit source that occurs in the southwest quadrant of the source.
- **Y Coordinate [m]:** Enter here the y (north-south) coordinate for the vertex (corner) of the open pit source that occurs in the southwest quadrant of the source.
- **Base Elevation [m or ft]:** Enter here the source base elevation. The model only uses the source base elevation if **Elevated** terrain is being used. The default unit is meters. You can also specify source base elevations in feet by changing the **Unit** option in the **SO-Source Inputs** window to Feet.

- **Average Release Height above Pit Base [m]:** Enter the average release height above the base of the pit in meters. 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.

Effective Pit Depth = Pit Volume / (Pit Width x Pit Length)

Source Release Parameters

- **Open Pit Emission Rate [g/(s-m^2)]:** Enter 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.

- **Length of the X Side of the Open Pit [m]:** If the Angle is 0.0 degrees, then this is the length in meters of the side of the open pit that is in the east-west direction. If Angle does 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 4-1 for illustration of this parameter.

- **Length of Y the Side of the Open Pit [m]:** If the Angle is 0.0 degrees, then this is the length in meters of the side of the open pit that is in the north-south direction. If Angle does 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 4-2 for illustration of this parameter.

Note: 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. Aspect ratios of greater than 10 to 1 will not be allowed.

Note: Since open pit sources cannot be subdivided, the user should characterize irregularly-shaped pit areas by a rectangular shape of equal area.

- **Volume of Open Pit [m^3]:** This is the volume of the open pit in cubic meters.

- **Orientation Angle from North [deg] (Optional):** 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 optional 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 angle is between -180 degrees and 180 degrees.

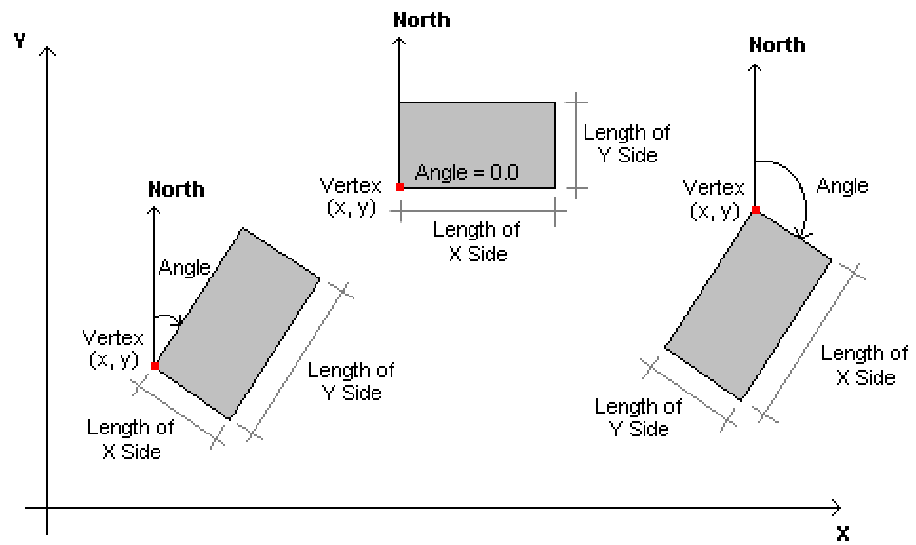


Figure 4-2. Relationship of open pit source parameters. Based on Figure 3-1 of the U.S. Environmental Protection Agency, 1995 User's Guide for the Industrial Source Complex (ISC3) Dispersion Models - Volume I, EPA-454/B-95-003a. U.S. Environmental Protection Agency. Research Triangle Park, NC 27711.

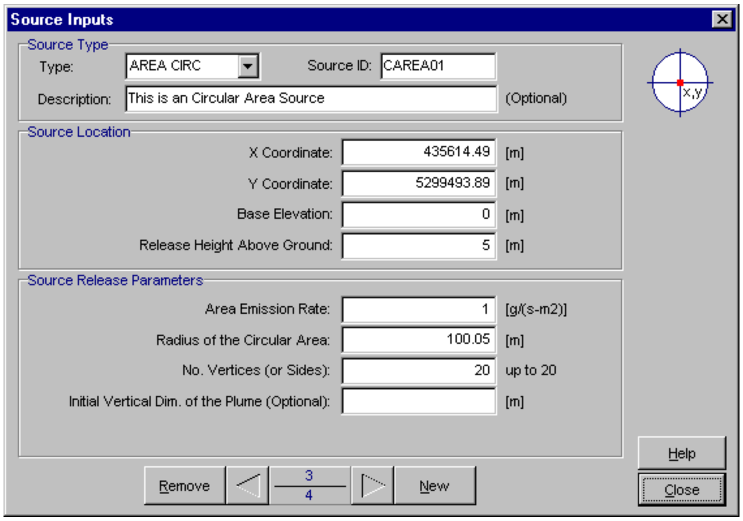
Note: Receptors should not be located within the boundaries of the pit. The ISCST3 model will set the concentration and/or deposition values at such receptors to zero. Such receptors will be identified during model setup and will be flagged in the summary of inputs.

Note: Since the open pit algorithm is applicable for particulate emissions, the particle categories for an **OPEN PIT** source MUST be defined on the **SO-Gas & Particle Data** window.

CIRCULAR AREA (AREA CIRC) Source Parameters

The **AREA CIRC** source may be used to specify a circular-shaped area source. The U.S. EPA models (ISCST3, AERMOD, and ISC-PRIME) treat the AREA CIRC source as an equal-area polygon of up to 20 sides. The AREA CIRC source type uses the same numerical integration algorithm for estimating impacts from area sources. The AREA CIRC source is merely a different option for specifying the shape of the area source.

For an **AREA CIRC** source, you must provide the following information:



Source Inputs dialog box – Circular Area Source Parameters

Source Type

- **Type:** Select **AREA CIRC** from the drop-down list box.
- **Source ID:** Enter here an identification name for the source being defined. The ID can be up to 8 characters long. Source IDs are always in upper case.
- **Description (Optional):** Enter here any description for the source up to 68 characters long.

Source Location

- **X Coordinate [m]:** Enter here the x (east-west) coordinate for the center of the circular area source.
- **Y Coordinate [m]:** Enter here the x (north-south) coordinate for the center of the circular area source.
- **Base Elevation [m or ft]:** Enter here the source base elevation. The model only uses the source base elevation if **Elevated** terrain is being used. The default unit is meters. You can also specify source base elevations in feet by changing the **Unit** option in the **SO-Source Inputs** window to Feet.
- **Release Height above Ground [m]:** Enter the release height above ground in meters.

Source Release Parameters

- **Area Emission Rate [g/(s-m^2)]:** 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.

- **Radius of the Circular Area [m]:** Enter here the radius of the circular area in meters.

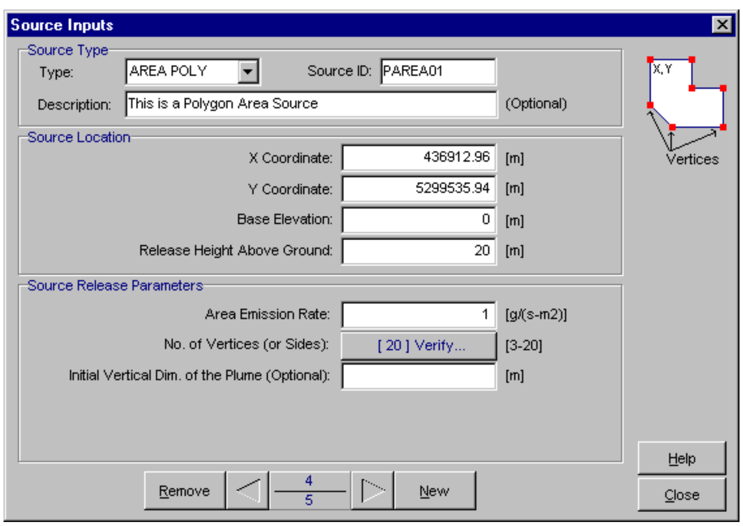
- **No. of Vertices (or Sides) (Default=20) [up to 20]:** This is the number of vertices or sides of the area source polygon. The U.S. EPA models (ISCST3, AERMOD, and ISC-PRIME) will automatically generate a regular polygon of up to 20 sides to approximate the circular area source. The polygon will have the same area as that specified for the circle. The default value for this parameter is 20.

- **Initial Vertical Dim. of the Plume [m] (Optional):** This optional parameter may be used to specify an initial vertical dimension of the circular area source plume in meters. 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 (AREA POLY) Source Parameters

The **AREA POLY** source may be used to specify an area source as an arbitrarily shaped polygon of between 3 and 20 sides. This source type option gives considerable flexibility for specifying the shape of an area source. The AREA POLY source type uses the same numerical integration algorithm for estimating impacts from area sources. The AREA POLY source is merely a different option for specifying the shape of the area source.

For an **AREA POLY** source, you must provide the following information:



Source Inputs dialog box – Polygon Area Source Parameters

__**Source Type**__

- **Type:** Select **AREA POLY** from the drop-down list box.

- **Source ID:** Enter here an identification name for the source being defined. The ID can be up to 8 characters long. Source IDs are always in upper case.

- **Description (Optional):** Enter here any description for the source up to 68 characters long.

Source Location

- **X Coordinate [m]:** Enter here the x (east-west) coordinate for one of the vertices of the polygon area source.

- **Y Coordinate [m]:** Enter here the x (north-south) coordinate for one of the vertices of the polygon area source.

- **Base Elevation [m or ft]:** Enter here the source base elevation. The model only uses the source base elevation if **Elevated** terrain is being used. The default unit is meters. You can also specify source base elevations in feet by changing the **Unit** option in the **SO-Source Inputs** window to Feet.

- **Release Height above Ground [m]:** Enter the release height above ground in meters.

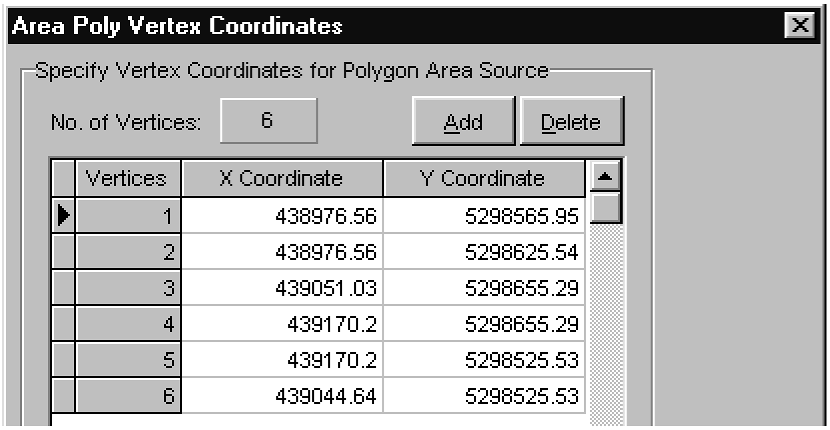
Source Release Parameters

- **Area Emission Rate [g/(s-m^2)]:** 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.

- **No. of Vertices (or Sides) [3 to 20]:** A button is displayed here identifying the number of vertices already specified for the current polygon area source. If the vertices were not identified yet, then the button will display a question mark between square brackets followed by the word **Specify**.

If vertices were already defined, the button will display the number of vertices between square brackets followed by the word **Verify**.

You can specify or verify the coordinates for the vertices of your polygon area by pressing the **Specify** or **Verify** button. By pressing this button, you have access to the **Area Poly Vertex Coordinates** dialog box.



Press this button to add a vertex to the polygon area source.

Press this button to delete the selected vertex from the polygon area source.

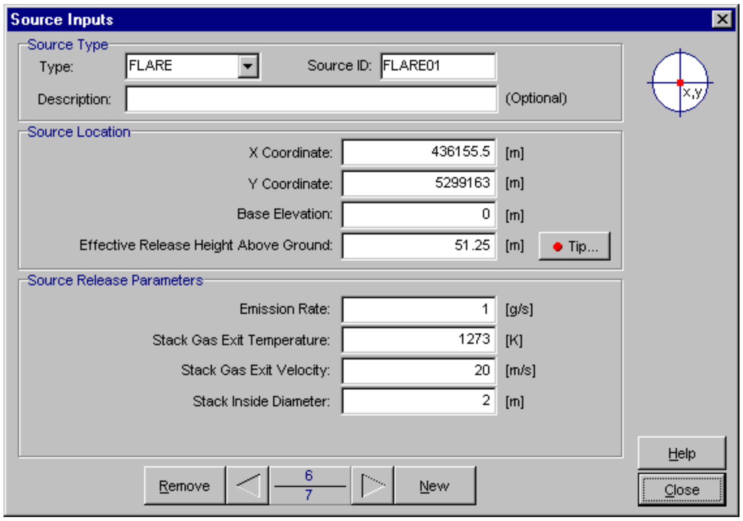
- **Initial Vertical Dim. of the Plume [m] (Optional):** This optional parameter may be used to specify an initial vertical dimension of the polygon area source plume in meters. This parameter is similar to the **Initial Vertical Dimension** parameter for **VOLUME** sources. The **Initial Vertical Dimension** parameter may be omitted, which is equivalent to using an initial sigma-z of zero.

FLARE Source Parameters

FLARE sources are used as control devices for a variety of sources. Flares must comply with requirements specified in 40 CFR 60.18. A minimum 98% reduction of all combustible components of the original emission must be achieved (U.S. EPA 1992). The U.S. EPA models (ISCST3, AERMOD, and ISC-PRIME) do not have a specific source type option for flare sources. 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 1992). ISC-AERMOD View implemented a FLARE source type option, which will treat flare sources in the following way:

- Flare sources will be written to the input file as Point sources.
- The Effective Release Height above ground for Flare sources should be given by adding the flare height to the stack height. ISC-AERMOD View can perform this calculation for you by pressing the Tip button.

For a **FLARE** source, you must provide the following information:



Source Inputs dialog box – Point Source Parameters

Source Type

- **Type:** Select **FLARE** from the drop-down list box.

- **Source ID:** Enter here an identification name for the source being defined. The ID can be up to 8 characters long. Source IDs are always in upper case.

- **Description (Optional):** Enter here any description for the source up to 68 characters long.

Source Location

- **X Coordinate [m]:** Enter here the x (east-west) coordinate for the source location in meters (center of the flare source).

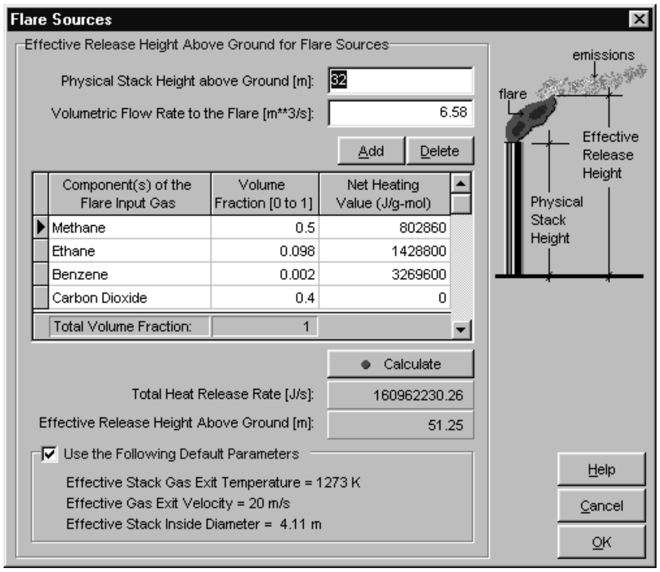
- **Y Coordinate [m]:** Enter here the y (north-south) coordinate for the source location in meters (center of the flare source).

- **Base Elevation [m or ft]:** Enter here the source base elevation. The model only uses the source base elevation if **Elevated** terrain is being used. The default unit is meters. You can also specify source base elevations in feet by changing the **Unit** option in the **SO-Source Inputs** window to Feet.

- **Effective Release Height above Ground [m]:** Enter the effective release height above the ground in meters. The effective release height should be given as the stack height plus the flare height.

  The **Tip** button allows you to calculate the effective release height above ground for flare sources. The following dialog box is displayed when the **Tip** button is pressed:



Effective Release Height dialog box

Calculating Effective Release Height

ISC-AERMOD View follows the procedures presented on the "Workbook of Screening Techniques for Assessing Impacts of Toxic Air Pollutants" (U.S. EPA 1992) for calculating the effective release height above ground for flare sources.

To calculate the effective flare release height, the following parameters are requested:

- **Physical Stack Height Above Ground [m]:** Enter the height of the stack above ground in meters.

- **Volumetric Flow Rate to the Flare [m^3/s]:** Enter the volumetric feed gas flow rate in cubic meters per second.

- **Component(s) of the Flare Input Gas:** Enter here all the components of the gas stream for the flare source.

- **Volume Fraction [0 to 1]:** 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.

- **Net Heating Value [J/g-mol]:** Enter here the heat of combustion for each component of the gas stream. If a component is not combustible (does not affect flame heat), then enter 0 for the net heating value.

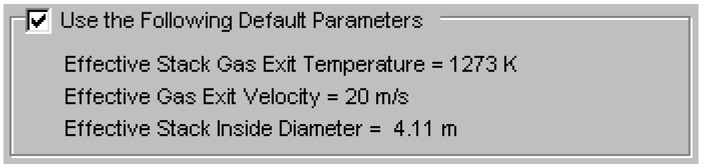
After you enter all the above parameters, you must click on the **Calculate** button, to get the calculated values for the following parameters:

- **Total Heat Release Rate [J/s]:** This parameter is calculated by ISC-AERMOD View by using the Lahey & Davis equation as presented *in U.S. EPA, 1992*.

Approximately 45% of the total heat release is assumed to be radiated as sensible heat.

- **Effective Release Height Above Ground [m]:** This parameter is calculated by adding the flare height to the stack height using Beychok equation as presented in *U.S. EPA, 1992*.

In addition to the effective release height parameter calculated as described above, you can also use default values for the effective stack parameters listed below. These default values follow the same implementation used in the SCREEN3 model for flare sources.



- **Effective Stack Gas Exit Temperature [K]:** This parameter is assumed to be 1,273K.

- **Effective Stack Gas Exit Velocity [m/s]:** This parameter is assumed to be 20 m/s.

- **Effective Stack Diameter [m]:** 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. (U.S. EPA 1995b).

If more realistic stack parameters can be determined for the above parameters, then you should not check the **Default Effective Stack Parameters** box and instead specify your own values on the **Source Inputs** dialog under **Source Release Parameters**.

<u>Source Release Parameters</u>

- **Emission Rate [g/s]:** Enter the emission rate of the pollutant in grams per second. The same emission rate is used for both concentration and deposition calculations.

- **Stack Gas Exit Temperature [K]:** Enter the temperature of the released gas in degrees Kelvin.

- **Stack Gas Exit Velocity [m/s]:** Enter the stack gas exit velocity in meters per second. The exit velocity can be determined from the following formula:

$$V_s = \frac{4\,V}{\pi d_s^2}$$

V_s = Exit Velocity
V = Flow Rate
d_s = Stack Inside Diameter

- **Stack Inside Diameter at Release Point [m]:** Enter the stack inside diameter in meters.

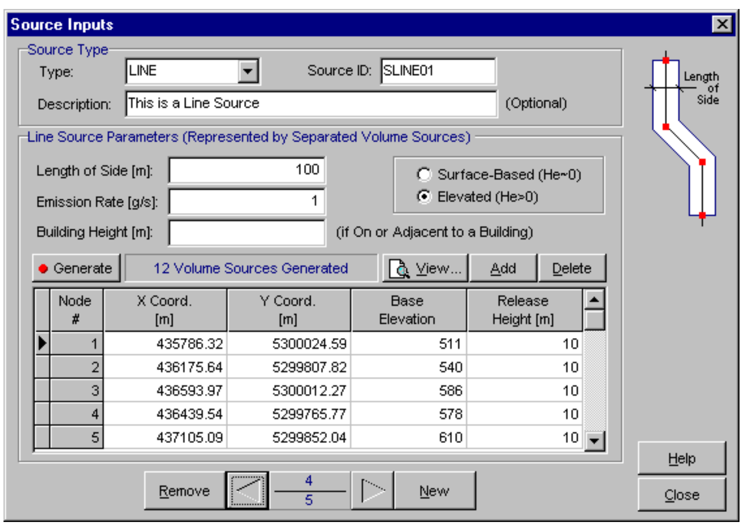
Note: The X and Y Coordinates for the source location may be input as Universal Transverse Mercator (UTM) coordinates, or may be referenced to a user-defined origin.

LINE Source Parameters

The U.S. EPA models (ISCST3, AERMOD, and ISC-PRIME) handle line sources as volume sources. ISC-AERMOD View can automatically generate these volume sources to represent the line source that you specify.

At the present time ISC-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. Examples of line sources are conveyor belts and rail lines.

For a **LINE** source, you must provide the following information:



Source Inputs dialog box – Line Source Parameters

<u>**Source Type**</u>

- **Type:** Select **LINE** from the drop-down list box.
- **Source ID:** Enter here an identification name for the source being defined. The ID can be up to 8 characters long. Source IDs are always in upper case.
- **Description (Optional):** Enter here any description for the source up to 68 characters long.

Line Source Parameters (Represented by Separated Volume Sources)

- **Length of Side [m]:** Enter here the length of the side of the line source.

- **Emission Rate [g/s]:** Enter the emission rate of the pollutant. The same emission rate is used for both concentration and deposition calculations.

- **Type of Volume Source:** You should specify what type of line/volume source applies, surface-based or elevated. The selection of one option or the other will determine how the Initial Vertical Dimension of the generated volume sources will be calculated.

1. **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 the procedure used to estimate the Initial Vertical Dimension of the source, if this option is selected.

   Initial Vertical Dimension = Release Height above Ground / 2.15

2. **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

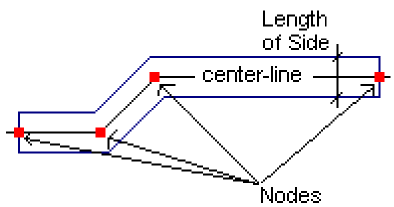
   Initial Vertical Dimension = Release Height above Ground / 4.3

- **Building Height (If On or Adjacent to a Bldg) [m]:** If your line source is Elevated and 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.

<u>Line Source Parameters</u>

Line source nodes are located at the center-line of a line source at the intersection of two line segments. If a line source is composed of only one line segment, then the nodes will be located at the edges of the center-line segment. You must specify the location, base elevation, and release height for all the nodes.

The following is the description of the parameters that must be specified for each node:



- **X Coord. [m]:** Enter here the x (east-west) coordinate for the node in meters.
- **Y Coord. [m]:** Enter here the y (north-south) coordinate for the node in meters.
- **Base Elevation [m or ft]:** Enter here base elevation at the node location. The model only uses the base elevation if **Elevated** terrain is being used. The default unit is meters. You can also specify base elevations in feet by changing the **Unit** option in the **SO-Source Inputs** window to Feet.
- **Release Height [m]:** Enter the release height above ground in meters at the node location.

You can use the **Add** and the **Delete** buttons (located above the table) to add a new node to your line source or to remove a node from the table respectively. See below the function of the buttons and panel located above the table.

Press this button if you want to generate the volume sources that will represent your line source.

This panel displays the number of volume sources generated for the current line source.

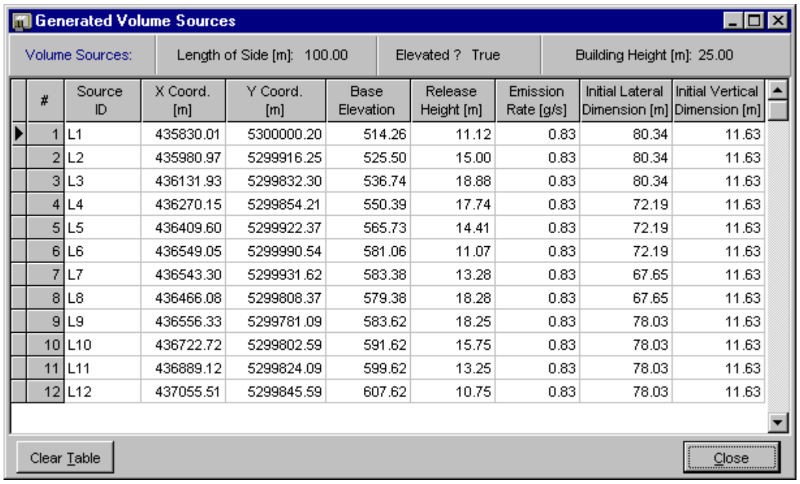
This panel is displayed when no volume sources were generated yet or if any parameters were changed.

Press this button to generate and view the volume sources generated for the current line source.

Press this button to add a new row to the table, so you can add the parameters for a new node. To add a new row, you can also click the arrow down key from your keyboard.

Press this button to delete a node (row) from your line source.

The following parameters are automatically calculated by ISC-AERMOD View for the generated volume 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]
1	L1	435830.01	5300000.20	514.26	11.12	0.83	80.34	11.63
2	L2	435980.97	5299916.25	525.50	15.00	0.83	80.34	11.63
3	L3	436131.93	5299832.30	536.74	18.88	0.83	80.34	11.63
4	L4	436270.15	5299854.21	550.39	17.74	0.83	72.19	11.63
5	L5	436409.60	5299922.37	565.73	14.41	0.83	72.19	11.63
6	L6	436549.05	5299990.54	581.06	11.07	0.83	72.19	11.63
7	L7	436543.30	5299931.62	583.38	13.28	0.83	67.65	11.63
8	L8	436466.08	5299808.37	579.38	18.28	0.83	67.65	11.63
9	L9	436556.33	5299781.09	583.62	18.25	0.83	78.03	11.63
10	L10	436722.72	5299802.59	591.62	15.75	0.83	78.03	11.63
11	L11	436889.12	5299824.09	599.62	13.25	0.83	78.03	11.63
12	L12	437055.51	5299845.59	607.62	10.75	0.83	78.03	11.63

Generated Volume Sources dialog box

- **Length of Side [m]:** Length of the side of the volume source in meters. The volume source cannot be rotated and has the X side equal to the Y side (square). The length of side will be the same for all generated volume sources and is equal to the length of side specified for the line source.

- **Elevated ?** If your line source is elevated then True is displayed. If the line source is surface-based then False is displayed.

- **Building Height [m]:** If your line source is elevated and is on or adjacent to a building, then the specified building height will be displayed here. The building height is then used to calculate the Initial Vertical Dimension of the volume source.

- **Source ID:** Identification name for the generated volume source.

- **X Coord. [m]:** X (east-west) coordinate for the source location in meters. This location is for the center of the volume source.

- **Y Coord. [m]:** Y (north-south) coordinate for the source location in meters. This location is for the center of the volume source.

- **Base Elevation [m or ft]:** Source base elevation. The base elevation is interpolated from the base elevation specified for the two nodes of the same line segment.

- **Release Height [m]:** Release height above ground in meters (center of volume). The release height is interpolated from the release height specified for the two nodes of the same line segment.

- **Emission Rate [g/s]:** 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.

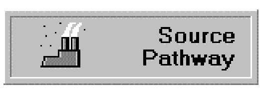
- **Initial Lateral Dimension [m]:** This parameter is calculated by ISC-AERMOD View by using the equations presented in Table 4-2 (See Line Source Represented by Separated Volume Sources).

- **Initial Vertical Dimension [m]:** This parameter is calculated by ISC-AERMOD View by using the equations presented in Table 4-2.

Comments (Optional)

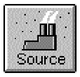
This option allows you to add any extra comments you feel are necessary to the Source Pathway portion of the input file.

SO - Building Downwash Window



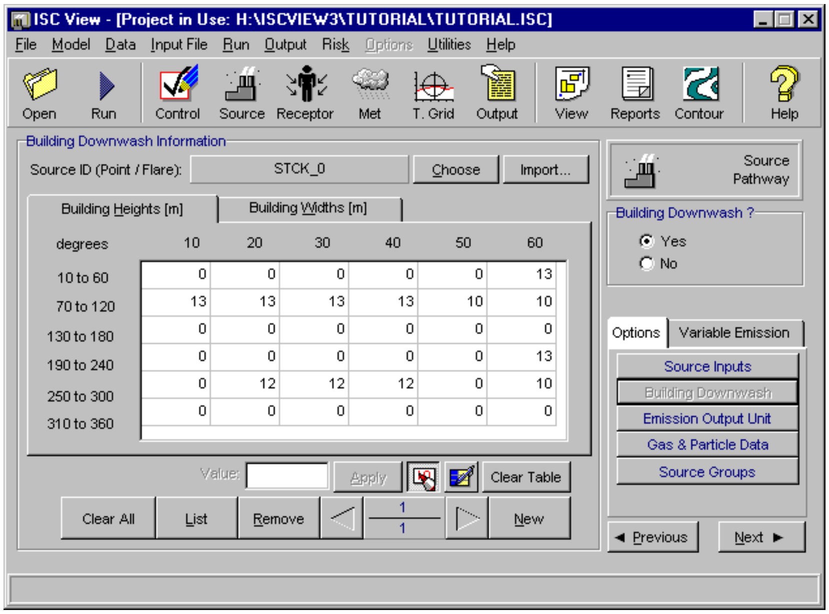
The U.S EPA models (ISCST3, AERMOD, and ISC-PRIME) include algorithms to model the effects of building downwash on emissions from nearby or adjacent **POINT** sources. The building downwash algorithms only apply to **POINT** sources. Since ISC-AERMOD View implements the FLARE source as a POINT source, then **FLARE** sources can also be included in the building downwash calculations. Refer to Volume II of EPA's ISC User's Guide for a technical description of the building downwash algorithms.

You have access to the **SO-Building Downwash** window by pressing the **Source** button located on the menu toolbar and then press the **Building Downwash** button. If you are in any Source Pathway window, press the **Building Downwash** button located on the lower right side of any Source (SO) Pathway window. From the menu select **Data | Source | Building Downwash**.



+ 

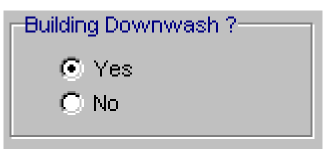
= Displays the **SO-Building Downwash** window



SO-Building Downwash Window

Defining Building Downwash Information

You can specify building downwash information for point and flare sources. Before specifying any data you first need to check the **Yes** option under the Building Downwash? frame.



The following information is requested for the building downwash option:

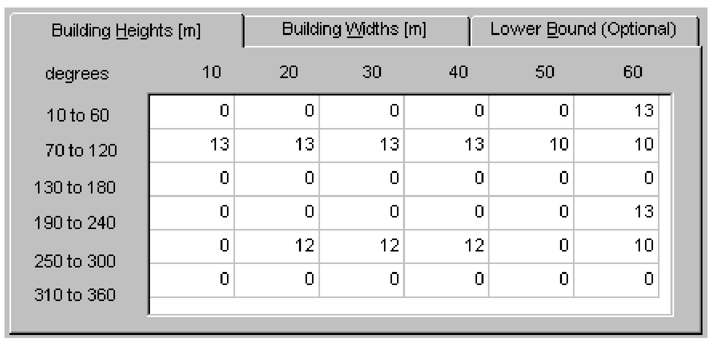
- **Source ID (Point/Flare):** This is the ID for the point or flare source for which the building downwash information will apply.

Press this button to select the BPIP output file from where the building downwash information should be imported. If you are in the ISC-PRIME mode, then you should import the BPIP-PRIME output file (extension **PRO** if generated using BPIP View). If you are in the ISCST3 or

AERMOD mode, then you should import the BPIP output file (extension **BPO** if generated using BPIP View).

If you are not using the **Import** function, then, for each record, you must press this button to select the **Source ID** for which building downwash information will be specified. You select **Source ID** from the **Source ID** dialog box, which contains a list of all point and flare sources already defined in your project.

- **Building Heights [m]:** The **Building Heights [m] or 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. This way, it is easy for you to check the angle for the cell you are inputting data. The row heading displays the angle range for that specific row (e.g., 10 to 60 deg).



degrees	10	20	30	40	50	60
10 to 60	0	0	0	0	0	13
70 to 120	13	13	13	13	10	10
130 to 180	0	0	0	0	0	0
190 to 240	0	0	0	0	0	13
250 to 300	0	12	12	12	0	10
310 to 360	0	0	0	0	0	0

Building Heights tab

- **Building Widths [m]:** The **Building Widths [m] 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.

The following parameters are only applicable to **ISC-PRIME**:

- **Lengths [m]:** 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 [m]:** 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 [m]:** 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.

The following parameters are only applicable to **ISCST3 and AERMOD**:

- **Lower Bound (Optional) [m]:** The **Lower Bound (Optional)** tab displays a table containing 6 columns and 6 rows with a total of 36 cells. These 36 values correspond 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 Low Bound option is used to exercise the **Non-Regulatory Default** option (see Control Pathway) 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. For use of upper bound input 0. See below the meaning of the 0 and 1 values that you input on the table.

  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).

Note: If the **Regulatory Default** option has been selected on the **CO-Dispersion Options** window, then the **Lower Bound (Optional)** tab becomes invisible and any low bound inputs will be ignored, and the model will calculate the upper bound estimates only (value = 0).

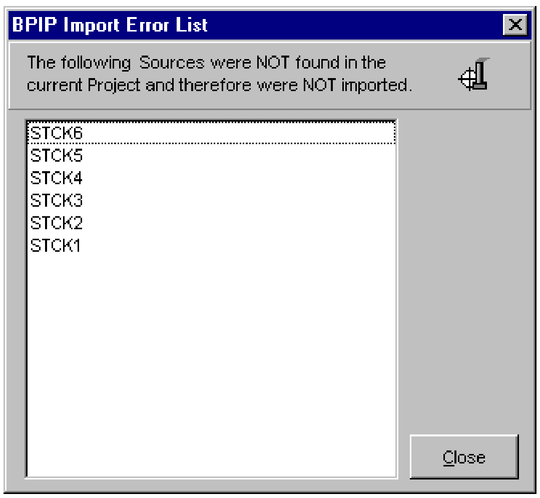
Tip: Double-click on a cell in the **Lower Bound** table to switch from the upper bound value (0) to a lower bound value (1) and vice-versa.

Inputting Data on the Building Downwash Tables

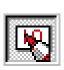
You can input information on the **Building Downwash** tables in two ways:

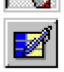
- **Importing:** Click the **Import** button and select the BPIP output file to be used. If using BPIP View, the BPIP output file will have an extension **BPO**. Building heights and projected building widths for each stack will be automatically placed in the appropriate building downwash tables.

  Make sure that all the sources defined in the BPIP output file are also defined in your ISC-AERMOD View project. Source **Name** in the BPIP output file must be the same as the **Source ID** you defined in ISC-AERMOD View. ISC-AERMOD View will not import building downwash data for sources not found in your project. A list of these sources will be displayed for your information.



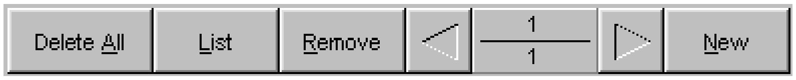
- **Typing:** Type the values directly on the table. Make sure the **Edit** button is pressed down (edit mode). You can also use the **Mark** button to multi-select rows in order to apply the same value to all selected rows. To select a row, click on the **Mark** button and then pressing down the Shift, click with the mouse the rows you want to select. To apply the same value to selected rows, type in the value on the **Value** field and click on the **Apply** button. The **Clear Table** button is used to clear all values from the table being displayed.

 Edit button

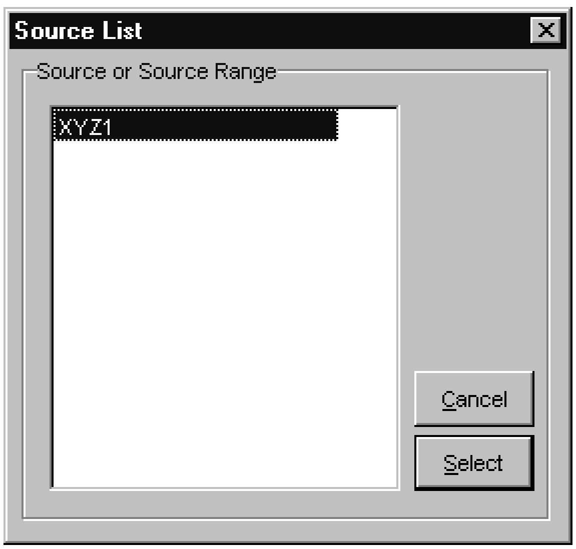
 Mark button

Specifying Building Downwash for More than One Source

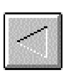
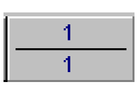
Building downwash can be specified for different point and flare sources. The following buttons will help you with managing information that was defined for more than one source. The function of each one of these buttons is explained below:

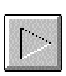


- **Delete All:** Press this button to delete all records created for the building downwash option.
- **List:** Displays the **List** dialog box. This box contains a list of all Point and Flare sources to which Building Downwash information is defined.



Source List dialog box

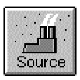
- **Remove:** Removes the current record.
-  Press the Previous button to display the Building Downwash information for the previous record.
- 

 The top panel will display the entry number for the current record. The bottom panel will display the total number of records already defined.
-  Press the Next button to display the building downwash information for the next record.
- **New:** Creates a new record.

SO - Emission Output Unit Window

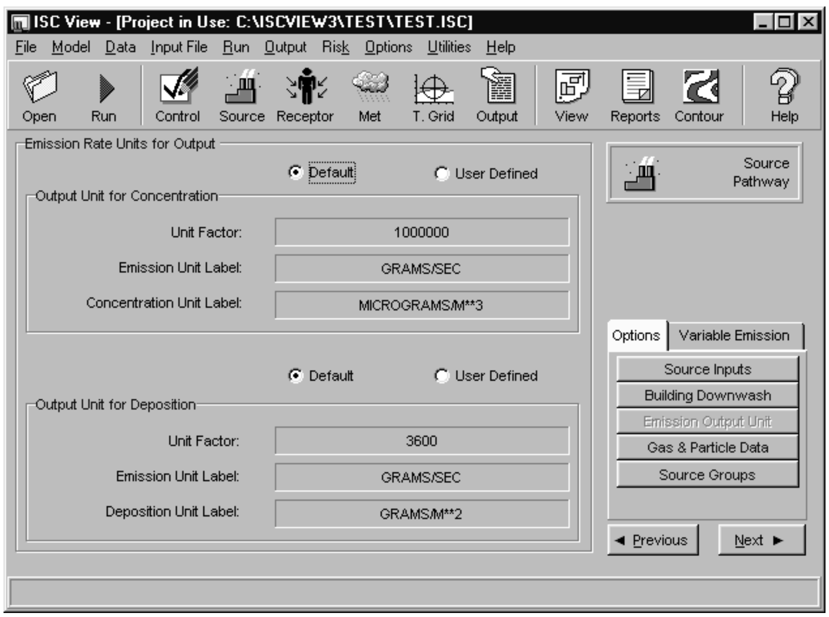
The U.S. EPA models (ISCST3, AERMOD, and ISC-PRIME) use default output units of micrograms per cubic meter ($\mu g/m^3$) for concentration calculations and grams per square meter (g/m^2) for deposition calculations. In ISC-AERMOD View, you can specify different output units on the **SO-Emission Output Unit** window.

You have access to the **SO-Emission Output Unit** window by pressing the **Source** button located on the menu toolbar and then press the **Emission Output Unit** button. If you are in any Source Pathway window, press the **Emission Output Unit** button located on the lower right side of any Source (SO) Pathway window. From the menu select **Data | Source | Emission Output Unit**.



+


= Displays the **SO-Emission Output Unit** window



SO – Emission Output Unit window

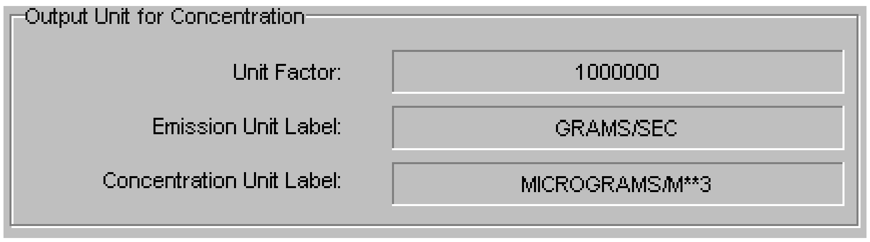
Defining Emission Output Units

In the **SO-Emission Output Unit** window you have two options:

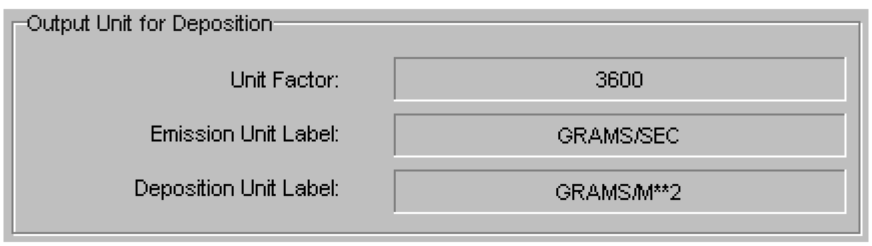
1. Default - This option uses the following default output units:

SOURCE TYPE	INPUT UNITS	OUTPUT	UNITS
		Concentration	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/m^3):** The unit factor of 1000000 converts grams to micrograms.



- **Deposition (Grams/m^2):** The unit factor of 3600 essentially converts grams/sec to grams/hour.



2. **User Defined:** this option allows you to specify output units, other then the default, for **Concentration** calculations and/or **Deposition** calculations. Note that if **Total Deposition**, **Dry Deposition**, or **Wet Deposition** option was not selected on **CO-Dispersion Options** window, then only the Concentration output unit can be specified (the Deposition options are not visible).

   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 concentration or deposition output units. The emission rate unit factor applies to all sources for a given run.

   - **Emission Unit Label:** This is the emission rate unit label and can be up to 40 characters in length. No blank spaces are allowed in this field. Instead of spaces you can use characters like / or -, e.g., GRAMS/SEC or GRAMS-PER-SECOND.

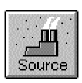
- **Concentration or Deposition Unit Label:** This is the output unit label for concentration or deposition calculations. No blank spaces are allowed in this field. Instead of spaces you can use characters like / or -, e.g., MICROGRAMS/M**3 or MICROGRAMS-PER-CUBIC-METER.

SO – Gas & Particle Data Window

(ISCST3 and ISC-PRIME Only)

In the **SO-Gas & Particle Data** window, you can specify gas deposition parameters, particle data, and/or scavenging coefficients. The options available will depend on the output type being modeled. The gas dry deposition parameters are only applicable to the ISCST3 model and the particle data. The scavenging coefficient options are only applicable to ISCST3 and ISC-PRIME models. None of the gas and particle data options are applicable to AERMOD. Deposition algorithms were not implemented by the U.S. EPA in the AERMOD model, at the time this user's guide was written.

You have access to the **SO-Gas & Particle Data** window by pressing the **Source** button located on the menu toolbar and then press the **Gas & Particle Data** button. If you are in any Source Pathway window, press the **Gas & Particle Data** button located on the lower right side of any Source (SO) Pathway window. From the menu select **Data | Source | Gas & Particle Data**.



+ 

= Displays the **SO-Gas & Particle Data** window

SO – Gas & Particle Data window

Gas-Phase or Particle-Phase ?

The following parameters are requested in the **SO-Gas & Particle Data** window, depending on the type of calculations being performed and the source phase being modeled:

Gas-Phase

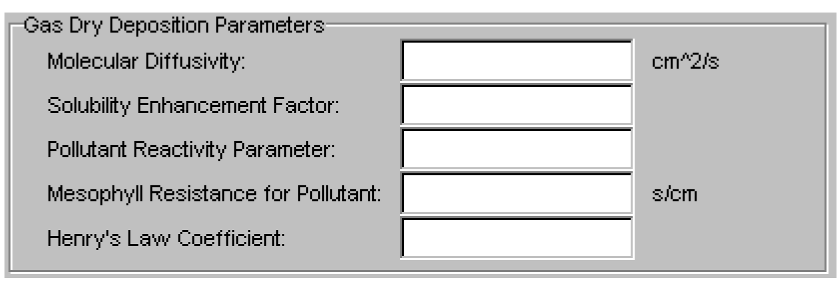
If you are modeling the gas-phase of a particular source, then the following may apply:

- **Scavenging Coefficients:** You must specify scavenging coefficients for the source **if one or more** of the following options were selected:

  - ❑ Wet Deposition
  - ❑ Plume Depletion due to Wet Removal

- **Gas Dry Deposition Parameters:** Gas dry deposition parameters are only requested if you are modeling with the following options:

  - ❑ Non-Regulatory Default
  - ❑ Air Toxics Options
  - ❑ Gas Dry Deposition Algorithm option



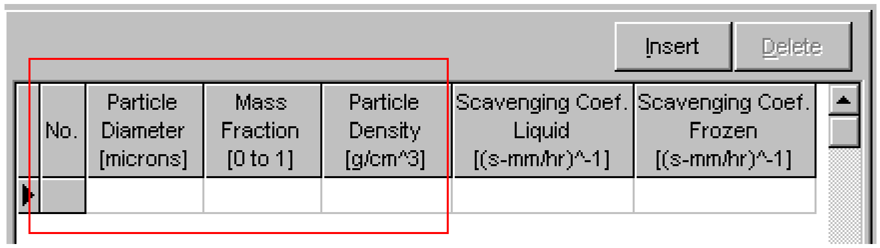
Particle-Phase

If you are modeling the particle-phase of a source, then the following may apply:

- **Particle Information:** You must specify particle information (particle diameter, mass fraction, and particle density) for the source if one or more of the following options were selected:

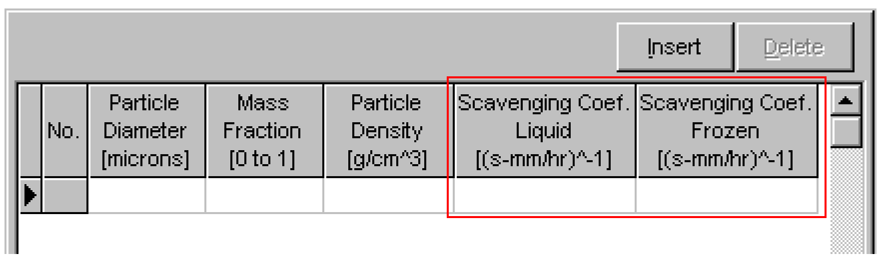
  - ❑ Dry Deposition

- ❑ Wet Deposition
- ❑ Total Deposition
- ❑ Plume Depletion due to Dry Removal
- ❑ Plume Depletion due to Wet Removal
- ❑ Open Pit Source



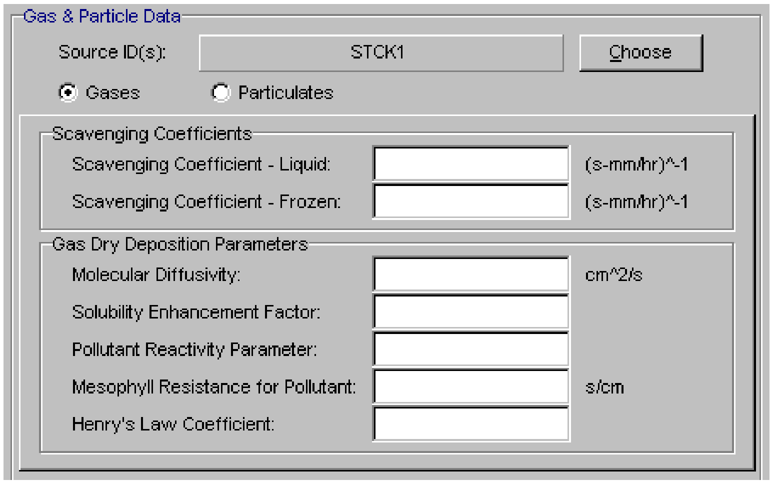
- **Scavenging Coefficients:** You must specify scavenging coefficients for the source if one or more of the following options were selected:

  - ❑ Wet Deposition
  - ❑ Total Deposition
  - ❑ Plume Depletion due to Wet Removal



Gas-Phase Options

If you are modeling the gas-phase of a particular source, then the following may apply depending on the options you selected in the Control Pathway (see Gas-Phase or Particle-Phase? section):



- **Source ID:** You must specify the source for which the gas data applies (source range not supported). Click the **Choose** button to display the **Source ID** dialog box, which contains a list of all sources already defined for the current run.

Scavenging Coefficients

- **Scavenging Coefficient - Liquid [(s-mm/hr)**-1]:** Specify the scavenging coefficient for liquid precipitation.

- **Scavenging Coefficient - Frozen [(s-mm/hr)**-1]:** Specify the scavenging coefficient for frozen precipitation.

Gas Dry Deposition Parameters

The following physical parameters for the pollutant being modeled must be specified.

- **Molecular Diffusivity [cm^2/s]:** 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.

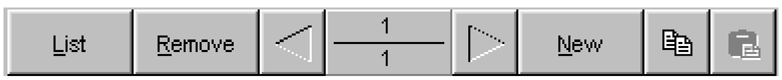
- **Pollutant Reactivity Parameter:** This is the pollutant reactivity parameter.

- **Mesophyll Resistance for Pollutant [s/cm]:** 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.

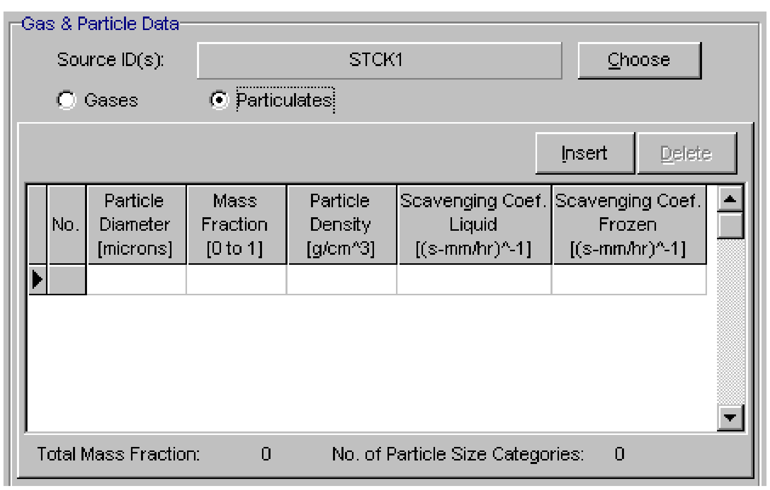
Note: The gas dry deposition parameters for the pollutant being modeled may be found in chemical engineering handbooks and various publications, such as the Air/Superfund National Technical Guidance Study Series (EPA, 1993).

Gas data can be specified for more than one source. The buttons shown below, also called the record navigator buttons, will help you in managing the information that is defined for more than one source. See the function of each one of these buttons on the Record Navigator Buttons section.



Particle-Phase Options

If you are modeling the gas-phase of a particular source, then the following may apply depending on the options you selected in the Control Pathway (see Gas-Phase or Particle-Phase? section):



- **Source ID:** You must specify the source for which the particle data applies (source range not supported). Click the **Choose** button to display the **Source ID** dialog box, which contains a list of all sources already defined for the current run.

Particle Information

- **No. (of Particle Size Categories):** Identifies the number of particle size categories specified for a particular source. You can only specify up to a maximum of 20 categories. You add a category by pressing the **Insert** button or the arrow down key or enter key after you have entered information for the previous category.

- **Particle Diameter [microns]:** Enter in this column, the particulate diameter in microns for each of the particle size categories, up to a maximum of 20.

- **Mass Fraction [0 to 1]:** 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 sum to 1.0 (within 2 percent). Note that the current total for the mass fraction is displayed on the bottom of the table.

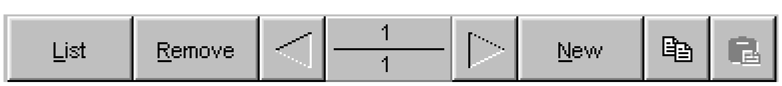
- **Particle Density [g/cm^3]:** In this column, you define the particle density in grams per cubic centimeter for each of the categories you have defined.

Scavenging Coefficients

- **Scavenging Coefficient - Liquid [(s-mm/hr)**-1]:** Enter in this column the particulate scavenging coefficient for liquid precipitation. The liquid scavenging coefficient should be entered for each of the particle size categories.

- **Scavenging Coefficient - Frozen [(s-mm/hr)**-1]:** Enter in this column the particulate scavenging coefficient for frozen precipitation. The frozen scavenging coefficient should be entered for each of the particle size categories.

Particle data can be specified for more than one source. The buttons shown below, also called the record navigator buttons, will help you in managing the information that is defined for more than one source.



See the function of each one of these buttons on the Record Navigator Buttons section.

Defining Scavenging Coefficients

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 (e.g., 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). As an initial approximation, you may use scavenging coefficients from:

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.

Table 4-3 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 4-3 - Wet scavenging rate coefficients as a function of particle size

Particle Diameter (microns)	Scavenging Rate Coefficient (s-mm/hr)**-1	Particle Diameter (microns)	Scavenging Rate Coefficient (s-mm/hr)**-1
0.2	1.25E-04	3.0	2.2E-04
0.3	0.8E-04	4.0	2.8E-04
0.4	0.6E-04	5.0	3.6E-04
0.5	0.5E-04	6.0	4.2E-04
0.6	0.4E-04	7.0	4.6E-04
0.7	0.4E-04	8.0	5.2E-04
0.8	0.4E-04	9.0	6.0E-04
0.9	0.4E-04	10.0	6.8E-04
1.0	0.4E-04	>10	6.8E-04
2.0	1.4E-04		

The scavenging rate coefficients for frozen precipitation are expected to be reduced to about 1/3 of the values presented on Table 4-3.

Record Navigator Buttons

Gas and particle information can be specified for more than one source. ISC-AERMOD View offers a series of buttons, the Record Navigator buttons, which helps you with managing the data that is defined for more than one source. The function of each one of these buttons is explained below:

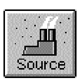
SO - Source Groups Window

You can analyze group contributions from particular sources together by creating source groups. For each run, at least one source group must be defined, up to the total number of source groups defined by the parameter NAVE. The NAVE parameter is defined for the original U.S. EPA models as follows:

- ISCST3 = Unlimited
- AERMOD = 12
- ISC-PRIME = 4

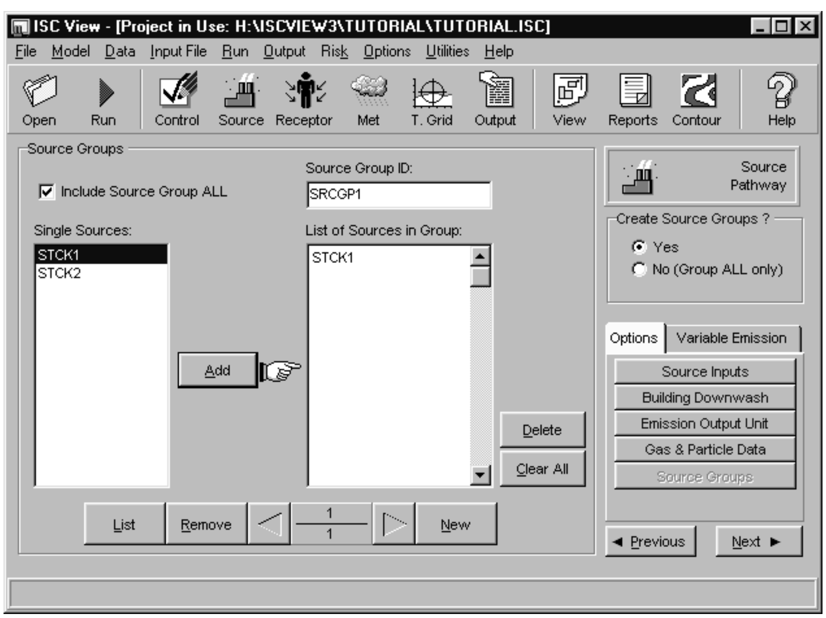
Sources groups may be used, for example, to model impacts from the source being permitted, the group of increment consuming PSD sources, and the group of all sources for comparison to a NAAQS in a single run. In ISC-AERMOD View, source groups are defined on the **SO-Source Groups** window.

You have access to the **SO- Source Groups** window by pressing the **Source** button located on the menu toolbar and then pressing the **Source Groups** button. If you are in any Source Pathway window, press the **Source Groups** button located on the lower right side of any Source (SO) Pathway window. From the menu select **Data | Source | Source Groups**.



+


= Displays the **SO- Source Groups** window



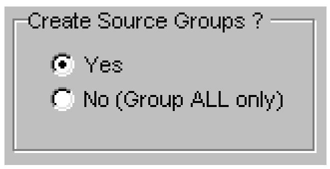
SO – Source Groups window

Defining Source Groups

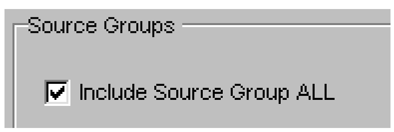
For each run, there must be at least one source group, which may consist of all sources. If you do not define any source group, ISC-AERMOD View automatically sets up a source group containing all sources you have defined for the current run. The ID for this source group is **ALL**.

To define one or more source groups, you have to specify the following:

- **Create Source Groups ?** If you want to define source groups other than the Source Group ALL, then you must select **Yes**. The option **No (Group ALL only)** automatically creates the group **ALL**, which contains all sources.



- **Include Source Group ALL:** If you decided to create your own source groups, you still have the option of including the Source Group ALL in you project. You can do this by checking the **Include Source Group ALL** option.



- **Source Group ID:** This is the name that identifies the source group and can be up to 8 alphanumeric characters. When you press the **New** button, ISC-AERMOD View automatically creates default names for the source group (e.g., SRCGP1, SRCGP2, etc.).

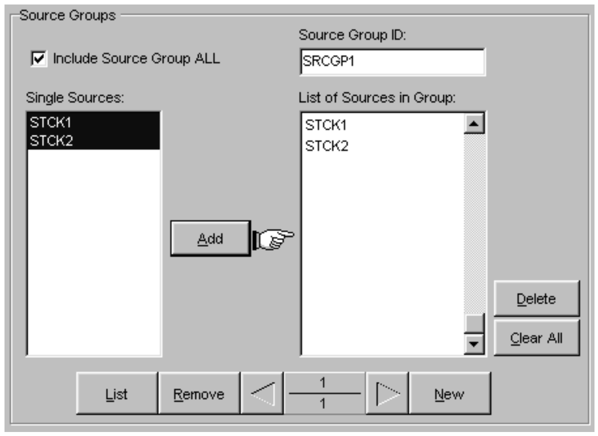
- **List of Sources in Group:** This is the list containing all the sources for the current source group. Use the **Add** button to add sources to this list.

Note: You will not be able to define source groups before defining the sources for the current run.

➤ **How to Add Sources to a Source Group:**

Step 1: Select the **Source ID** from the **Single Sources** list. You can select from the list more than one source at a time, by pressing down the Shift key while selecting the sources.

Step 2: After you have selected all the sources click the **Add** button to add these sources to the **List of Sources in Group**.



Step 3: You can delete information already entered on the **List of Sources in Group** using two buttons:

Deletes from the list, the selected (highlighted) items.

Deletes all the items added to the **List of Sources in Group**.

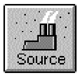
Note: You can only specify up to the maximum number of source groups as defined for the NAVE parameter for the model. The NAVE parameter is defined in the Preferences dialog box.

SO – Urban Sources Window

(AERMOD Only)

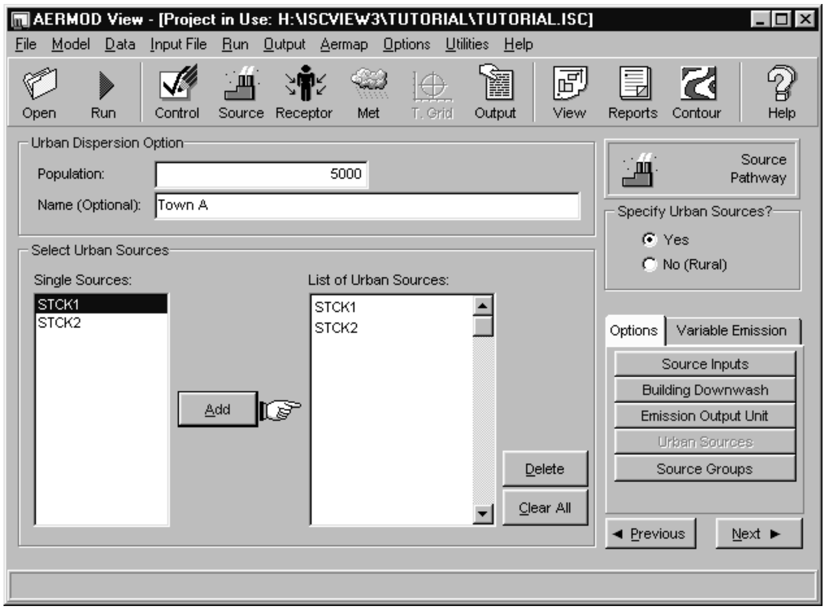
The AERMOD model allows you to incorporate the effects of increased surface heating from an urban area on pollutant dispersion under stable atmospheric conditions. This is the **Urban Dispersion Option** and can be specified in the **SO- Urban Sources** window.

You have access to the **SO-Urban Sources** window by pressing the **Sources** button located on the menu toolbar and then pressing the **Urban Sources** button. If you are in any Source Pathway window, press the **Urban Sources** button located on the lower right side of any Source (SO) Pathway window. From the menu select **Data | Source | Urban Sources**.



+


= Displays the **SO-Urban Sources** window



SO-Urban Sources window

Defining the Urban Dispersion Option

You can specify Urban Dispersion Option parameters and select Urban Sources if the **Yes** option in the **Specify Urban Sources?** frame is selected. If the **No (Rural)** option is selected, then you are modeling with rural dispersion coefficient and the **Urban Dispersion Option** and **Urban Sources** will not apply.

See below the description of all the options available in the **SO-Urban Sources** window.

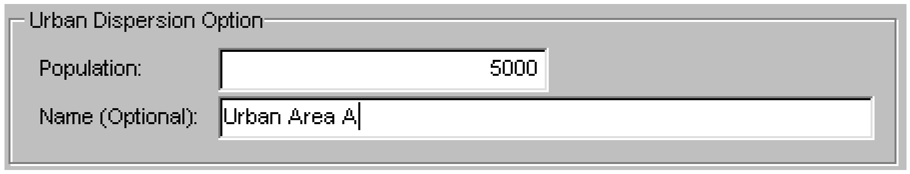
- **Specify Urban Sources?**

  1. **Yes:** This option indicates that you have selected the **Urban** dispersion coefficient option in CO-Dispersion Options window and therefore, AERMOD requests you to define the sources to be modeled with urban effects. Any sources not included in the list will be modeled without the urban effects.

  2. **No (Rural):** This option indicates that you have selected the **Rural** dispersion coefficient option in CO-Dispersion Options window and therefore sources will be modeled with no urban effects.

- **Population:** this is the population of the urban area.

- **Name (Optional):** this may be used to identify the name of the urban area.
- **Urban Sources:** You also need to identify which sources are to be modeled with urban effects.



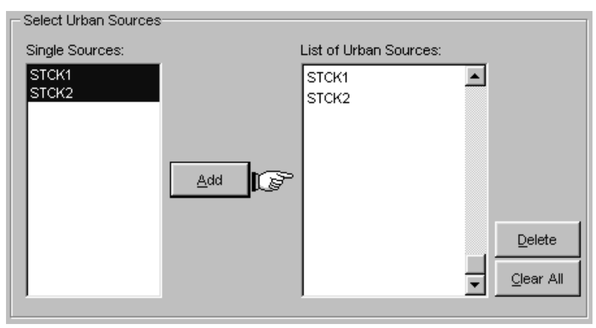
The **Population** and **Name** parameters can also be specified in the **CO-Dispersion Options** window.

➤ **How to Specify Urban Sources:**

The sources to be modeled with urban effects must be added to the **List of Urban Sources**. If a source is not included in the list, then it will be modeled without the urban effects.

Step 1: Select the **Source ID** from the **Single Sources** list. You can select from the list more than one source at a time, by pressing down the Shift key while selecting the sources.

Step 2: After you have selected all the sources click the **Add** button to add these sources to the **List of Urban Sources**.



Step 3: You can delete information already entered on the **List of Urban Sources** using two buttons:

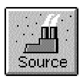
Deletes from the list, the selected (highlighted) items.

Deletes all the items added to the **List of Urban Sources**.

SO - Hourly Emission File Window

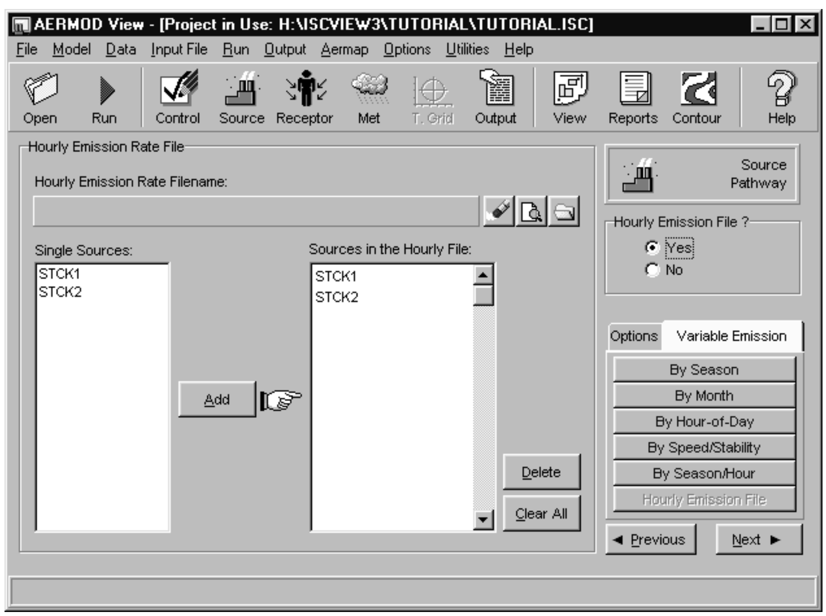
The Hourly Emission Rate File option allows you to specify hourly emission rates for one or more sources. Hourly emission rates must be provided in a separate file. You can only provide one hourly emission file per run.

You have access to the **SO-Hourly Emission File** window by pressing the **Sources** button located on the menu toolbar and then pressing **Hourly Emission File** button. If you are in any Source Pathway window, press the **Hourly Emission File** button located on the lower right side of any Source (SO) Pathway window. From the menu select **Data | Source | Hourly Emission File**..



+ 

= Displays the **SO- Hourly Emission File** window

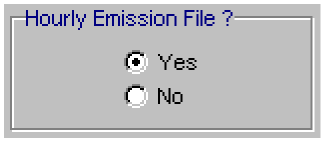


SO – Hourly Emission File Window

Specifying the Hourly Emission File

For each model run, you can specify only one hourly emission rate file. To be able to use the Hourly Emission File option, you must define the following:

- **Hourly Emission File?:** Since the **Hourly Emission Rate File** option is optional, you need to select the **Yes** option to be able to use this option.



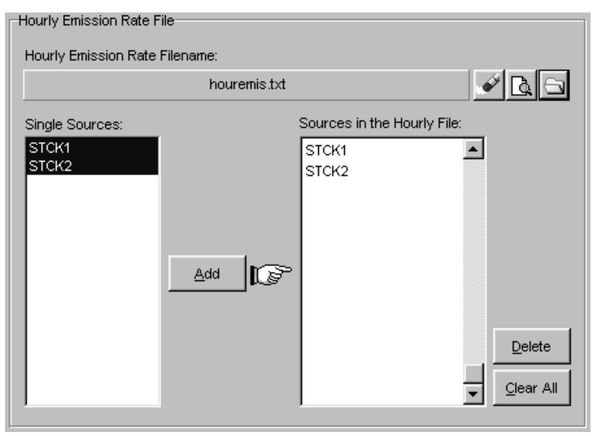
- **Hourly Emission Rate Filename:** Click on the folder button () to specify the name and location of the hourly emission file. Note that if the file is located in the same location as the project than only the filename is displayed on the panel, otherwise the full path is displayed.

- **Sources in the Hourly File:** You must identify the sources for which hourly emission rates are defined in the hourly emission file. The source IDs must match the ones you have defined on **the SO-Source Inputs** window.

➤ **How to Specify the Sources Included in the Hourly File:**

Step 1: Select the **Source ID** from the **Single Sources** list. You can select from the list more than one source at a time, by pressing down the Shift key while selecting the sources.

Step 2: After you have selected all the sources click the **Add** button to add these sources to the **Sources in the Hourly File** list.



Step 2: You can delete information already entered on the **Sources in the Hourly File** list, using two buttons:

Deletes from the list the selected (highlighted) sources.

Clears the list, deleting all the sources added to the list.

Format of the Hourly Emission Rate File

The format of each record (line) of the hourly emission rate file includes 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:

- Stack Gas Exit Temperature (K)
- Stack Gas Exit Velocity (m/s)

An example of the hourly emission rate file for a point source is given below:

```
SO HOUREMISS  88  8  16  1  STACK1  52.467  382.604  12.27
SO HOUREMISS  88  8  16  2  STACK1  22.321  377.882   9.27
SO HOUREMISS  88  8  16  3  STACK1  51.499  373.716  11.87
SO HOUREMISS  88  8  16  4  STACK1  36.020  374.827   9.63
```

The above parameters must be separated by at least one space. You do not need to include the **SO HOUREMISS** pathway and keyword on each line, however the **Year** parameter cannot begin before column 13.

The data in the hourly emission file must include the exact same dates as are included in the meteorological input files, and the **Source IDs** must correspond to the **Source IDs** defined on the **SO-Source Inputs** window and be in the same order.

Note: The model will check for a date mismatch between the hourly emissions file and the meteorological data, and also for a **Source ID** mismatch.

Note: 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** options on the **ME-Data Period** window are used, as long as all the dates (including those that are processed and those that are skipped) match the meteorological data files.

Note: The model will use the **Stack Release Height** and **the Stack Inside Diameter** defined on the **SO-Source Inputs** window, but will use the **Emission Rate**, **Stack Gas Exit Temperature** and **Stack Gas Exit Velocity** from the hourly emission file.

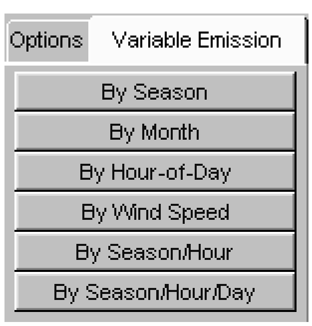
CAUTION: If the Emission Rate, the Stack Gas Exit Temperature, and the Stack Gas Exit Velocity are not included for a particular hour (any or all of these fields are blank), the model will interpret emissions data for that hour as missing and will set the parameters to zero. Since the emission rate will be zero, there will be no calculations made for that hour and that source.

SO - Variable Emission Factors Windows

The U.S. EPA models (ISCST3, AERMOD, and ISC-PRIME) provide the option of specifying variable emission rate factors for single sources or for groups of sources. The factors may vary on different time scales such as:

- Seasonally
- Monthly
- Hourly
- By wind speed
- By season and hour-of-day
- By Season, hour-of-day, and day-of-week

In ISC-AERMOD View, these different time scales options are provided to you in six windows:



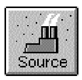
- **SO-By Season** window
- **SO-By Month** window
- **SO-By Hour-of-Day** window
- **SO-By Wind Speed** window

- ❒ **SO-By Season/Hour** window
- ❒ **SO-By Season/Hour/Day** window

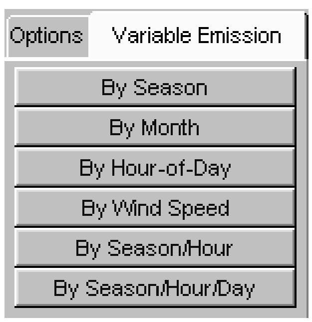
Since the layout and function of the Variable Emission Factors windows are very similar, we will explain them all in this section.

Displaying the SO – Variable Emission Factors Window

1. If you are in another Pathway window, press the **Source** toolbar button, press the **Variable Emission** tab and then press one of the variable emission rate buttons, **By Season**, **By Month**, **By Hour-of-Day**, **By Speed/Stability**, **By Season/Hour**, or **By Season/Hour/Day**; or



2. If you are in any Source Pathway window, press the **Variable Emission** tab and then press one of the variable emission rate buttons, **By Season**, **By Month**, **By Hour-of-Day**, **By Wind Speed**, or **By Season/Hour**; or



3. From the menu select **Data | Source | Variable Emission Factors** and then select one of the appropriate variable emission factor option: **By Season**, **By Month**, **By Hour-of-Day**, **By Wind Speed**, **By Season/Hour**, or **By Season/Hour/Day**.

Defining Variable Emission Rate Factors

The parameters necessary to specify variable emission rate factors are described below:

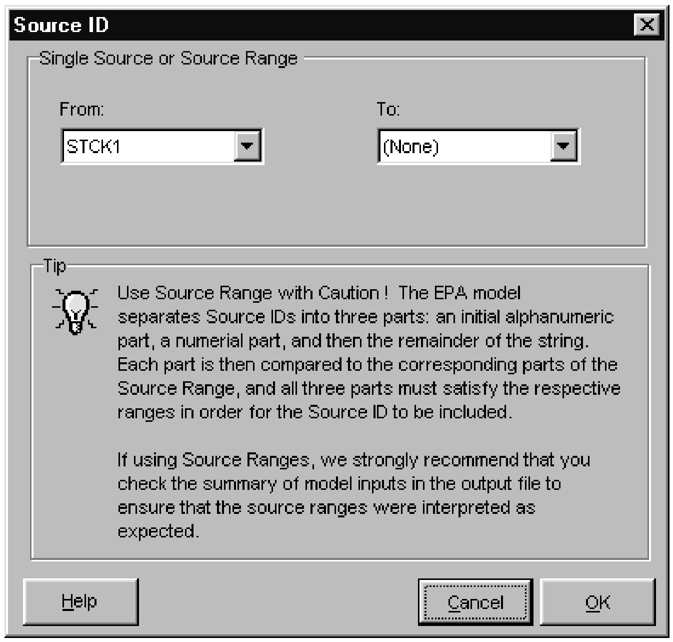
- **Source ID:** You can specify a single source or a source range for which the emission rate factors apply. Click the **Choose** button to display the **Source ID** dialog box, which contains a list of all sources already defined for the current run.

➤ **To select a Single Source or a Source Range, do the following:**

Step 1: Click on the left drop-down list box and select the **Source ID** for which you want to specify variable emission rate factors. If you want to choose a **Source Range**, this **Source ID** will be the first in the range.

Step 2: If you are specifying a **Source Range**, click the right drop-down list box and select the **Source ID** that will be the last in the range. For a **Single Source**, you should select the **(none)** option for this drop-down list box.

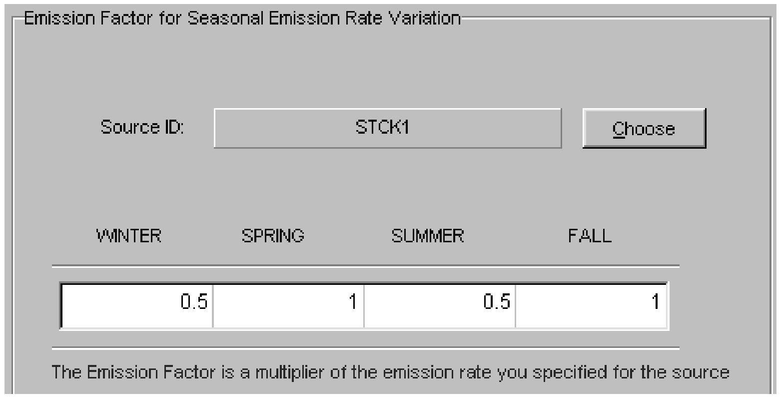
Step 3: Press the **OK** button to have your selection placed on the **Source ID** panel.



Source ID dialog box

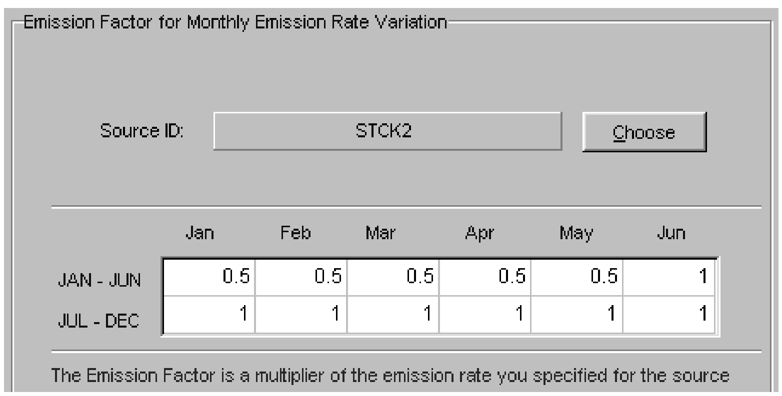
- **Variable Emission Rate Factors:** Variable emission rate factors are a multiplier of the emission rate you have specified for the source in the **Source Inputs** dialog box. A factor of 0.0, for example, means the source is not emitting. A factor of 0.5 means that the source is emitting 50% of the specified emission rate. A factor of 1.0 means that the source is emitting 100% of the specified emission rate. The number of emission factors to be input into the model depends on the variation type:

  1) By Season: For sources with emission rate varying seasonally, you should specify emission factors for the four seasons: WINTER, SPRING, SUMMER, and FALL.



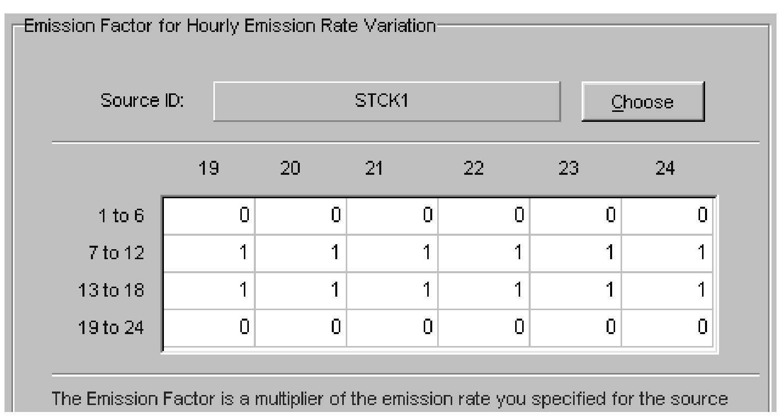
SO – By Season window

2. **By Month:** For sources with emission rate varying monthly, you should specify emission factors for each month. In the first row of the table, you specify emission rate factors for the months of January to June (**JAN-JUN**). In the second row of the table, you specify emission rate factors for the months of July to December (**JUL-DEC**).



SO – By Month window

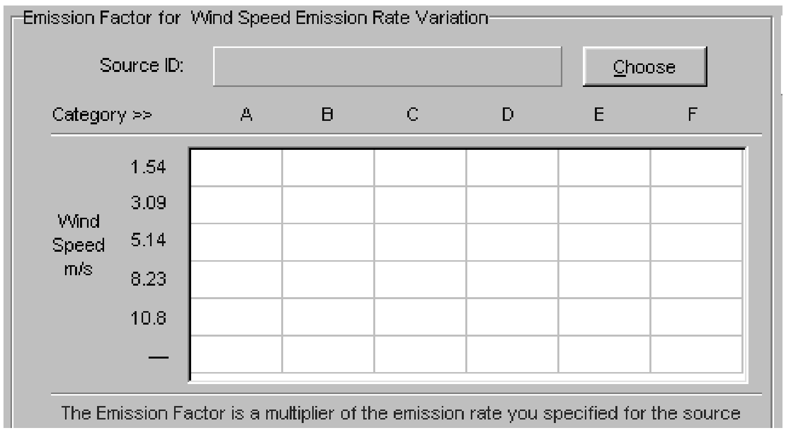
3. **By Hour-of-Day:** For sources with emission rate varying by hour-of-day, you should specify emission factors for the twenty four (24) hours of the day. In the first row of the table, you specify emission rate factors for hours **1** to **6**, row two for hours **7** to **12**, row three for hours **13** to **18**, and row four for hours **19** to **24**.



SO – By Hour-of-Day window

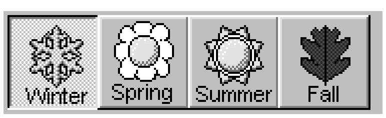
4. **By Wind Speed:** For sources with emission rate varying by wind speed. In the first row of the table, you specify emission rate factors for the first wind speed category (**1.54 m/s** by default) for each one of the six categories (**A** through **F**). In row two, you specify emission rate factors for the second wind speed category (**3.09 m/s** by default) for each one of the six categories (**A** through **F**), and so on. Note that the wind speed values displayed are the ones you have specified in **ME-Wind Speed Categories** window.

   You can change the default wind speed categories in the **ME-Wind Speed Categories** window. You can go to this window by pressing the **Wind Speed** button located on the right hand side of this window (Wind Speed).

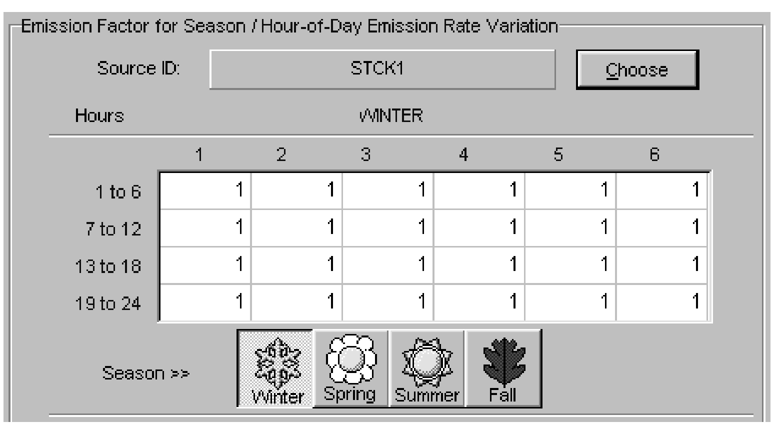


SO – By Wind Speed window

5. **By Season/Hour-of-Day:** For sources with emission rate varying by season and hour-of-day. To specify the variable emission rate factors for each season, you must click the appropriate **Season** button located on the bottom of the table. The table heading will specify which season you are currently working on.

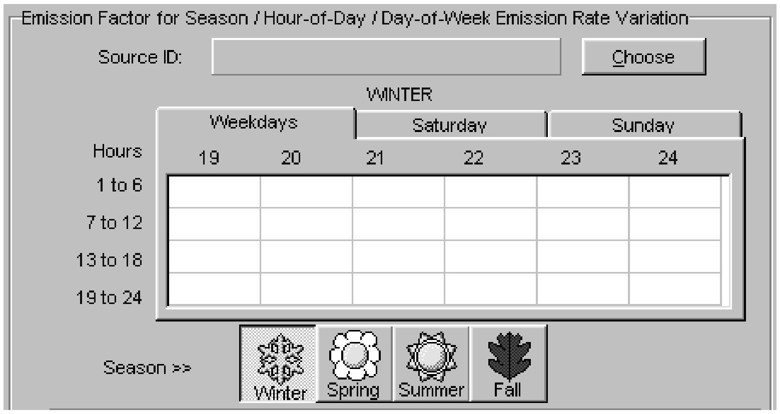


Season buttons



SO – By Season/Hour window

6. **By Season/Hour-of-Day/Day-of-Week:** For sources with emission rate varying by season, hour-of-day, and day-of-week. To specify the variable emission rate factors for each season, you must click the appropriate **Season** button located on the bottom of the table. The table heading will specify which season you are currently working on. Three tabs are available, one for Weekdays, one for Saturdays, and one for Sundays. Each tab contains a table with 24 cells, where you can specify the emission rate factor for each hour of the day.

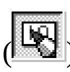


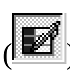
SO – By Season/Hour/Day window

Specifying Variable Emission Rate Factors

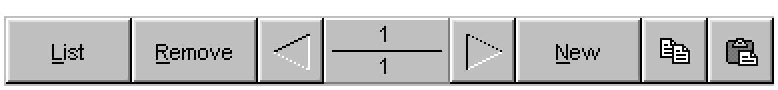
You can specify variable emission rate factors for a specific source or a range of sources in the following way:

1. Click on the **Choose** button and select the source or source range.

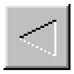
2. With the **Edit** button () pressed down, type the values directly in each cell.

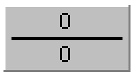
3. You can also click the **Mark** button () and holding down the Shift key select one or more rows from the table. Marked cells will be highlighted in blue. Type the emission factor value on the Emission Factor field and press the **Apply** button. The **Clear All** button is used to clear all values from the table.

The following buttons are used to manage information when more than one record is created:

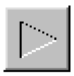


Removes the current record. The current record is the one being displayed on the current window. In cases where all the records are being displayed in the same table, the **Remove** button removes the record that is currently being highlighted or marked with an arrow.

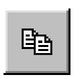

Press this button to display the information that was defined for a previous record.



The top panel will display the entry number for the current record. The bottom panel will display the number of records already defined.


Press this button to display the information that was defined for the next record from the list.

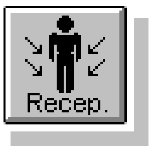
Allows you to create a new record.


Allows you to copy the contents of the current record into memory so you can paste it into a new record.


Allows you to paste a previously copied information into the current record.

CHAPTER 5

Receptor Pathway

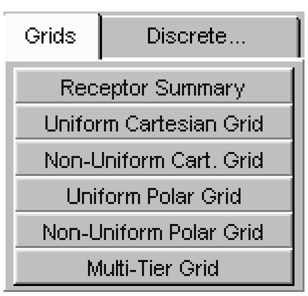


In the Receptor Pathway, you define receptor information for a particular run. You can define Cartesian grid receptor networks and/or polar grid receptor networks, with either uniform or non-uniform grid spacing up to the maximum defined by the parameter NNET (Number of Gridded Networks) in the U.S. EPA models.

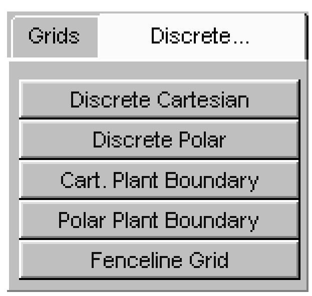
In addition to the receptor networks, you may also specify discrete receptor locations for modeling impacts at specific locations of interest. This may be used to model critical receptors, such as the locations of schools or houses, nearby Class I areas, or locations identified as having high concentrations by previous modeling analyses. The discrete receptors may be input as either Cartesian (x, y points) or as polar (distance and direction). Any or all of these types of receptors may be identified in a single run.

In the Receptor Pathway, the specification of gridded receptor networks and discrete receptor locations is optional. However, at least one of these options should be defined for each run. The maximum number of receptors that can be defined for each run is defined by the parameter NREC (Number of Receptors) in the U.S. EPA models.

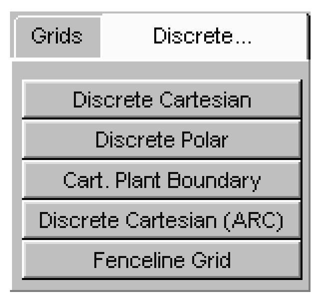
In ISC-AERMOD View, the Receptor Pathway inputs and options are available in several windows. The contents of these windows will be explained in detail in the sections that follows:



All Models



ISCST3 & ISC-PRIME

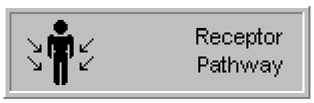


AERMOD

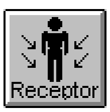
- ❒ RE-Receptor Summary Window
- ❒ RE-Uniform Cartesian Grid Window
- ❒ RE-Non-Uniform Cartesian Grid Window
- ❒ RE-Uniform Polar Grid Window
- ❒ RE-Non-Uniform Polar Grid Window
- ❒ Terrain Elevations for Receptor Grid Networks
- ❒ Flagpole Heights for Receptor Grid Networks
- ❒ Record Navigator Buttons
- ❒ RE-Multi-Tier Grid Window

- ❐ RE-Discrete Cartesian Window
- ❐ RE-Discrete Polar Window
- ❐ RE-Cartesian Plant Boundary Window
- ❐ RE-Discrete Cartesian (ARC) Window
- ❐ RE-Polar Plant Boundary Window
- ❐ RE-Fenceline Grid Window
- ❐ Terrain Elevations
- ❐ The Risk Mode

RE – Receptor Summary Window



The **RE-Receptor Summary** window is the first window displayed when you press the **Receptor** menu button. From this window, you can check the number and type of receptors already defined in your project, delete selected receptors, define receptor groups, define terrain height options, define flagpole options, and run AERMAP.



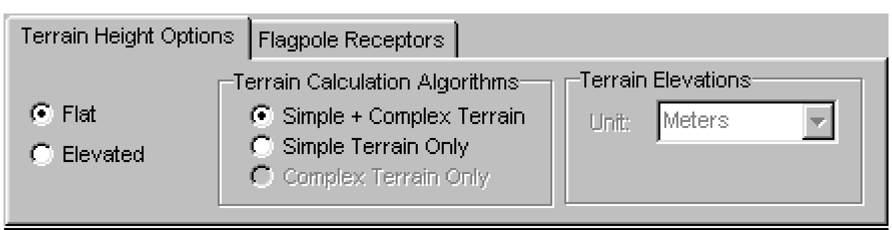
or


= Displays the **RE-Receptor Summary** window

See below the description of each one of these options.

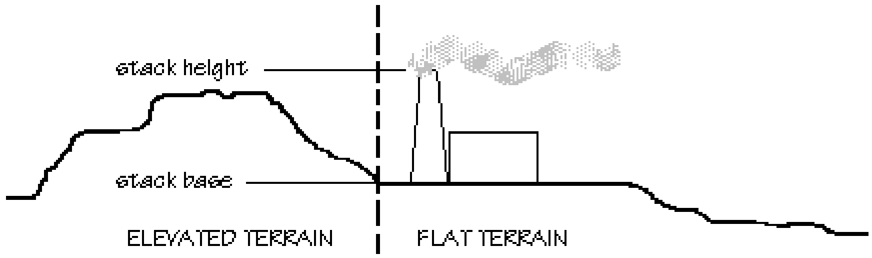
Terrain Height Options

Terrain height options can also be defined in the **CO-Pollutant/Avg Time/…** window. Any changes made in the terrain height options will be automatically updated in the **RE-Receptor Summary** window and vice-versa. Two terrain height options are available, **Flat** or **Elevated** terrain.



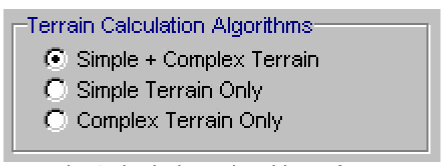
- **Flat:** Terrain height is assumed not to exceed stack base elevation (terrain height assumed to be 0.0 m). If this option is selected, then the model will assume **Flat** terrain calculations to be used throughout, regardless of the input of **Terrain Elevations** in the Receptor Pathway. Any **Terrain Elevations** that are entered in the Receptor Pathway are ignored if the **Flat** terrain option is selected.

- **Elevated:** Terrain height exceeds stack base elevation. If this option is selected, then **Terrain Elevations** are allowed/expected on the Receptor Pathway. If **Elevated** terrain is selected and a receptor height is not specified, then it is assumed to have a value of 0.0 meters.



Terrain Calculation Algorithms (ISCST3 & ISC PRIME Only)

Depending on the selected **Terrain Height Option** (**Flat** or **Elevated**) the following terrain calculation algorithms will be available:



Terrain Calculation Algorithms frame

- **Simple + Complex Terrain:** This is the default option and it is available for both, the **Flat** and the **Elevated**, terrain height options. This option instructs the model to implement both simple and complex terrain algorithms and also apply 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, i.e., terrain between release height and plume height.

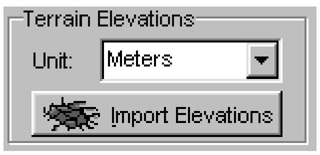
- **Simple Terrain Only:** This option is available for both, the **Flat** and the **Elevated** terrain height options. This option specifies that no complex terrain calculations will be made, i.e., uses ISCST algorithms only. You should not use this option if you are modeling with complex terrain (terrain above the release height of the source). Read **Note** below.

- **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, i.e., uses COMPLEX1 algorithms only.

Note: For terrain above the release height of the source (i.e., complex terrain), the model automatically truncates ("chops") the terrain to the physical release height(s) when modeling impacts at those receptors using the simple terrain (ISC) algorithm (**Simple Terrain Only** option). Terrain above the release height is not truncated when the COMPLEX1 algorithm is used (**Complex Terrain Only** option).

Terrain Elevations

If you have selected the **Elevated** terrain height option, then you can also specify the unit for the terrain elevation at the receptor location. You can select either **Meters** or **Feet** from the drop-down list box.

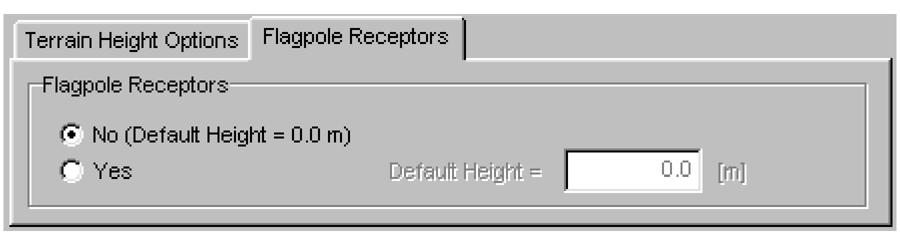


This button allows you to import terrain elevations for all receptors already defined in your project. The elevations will always be imported in **Meters** even if your DEM data is in feet. See section **Import Terrain Elevations** at the end of this chapter for more information on how to import elevations from various digital terrain data files.

Note: Please note that if you imported terrain elevations from USGS DEMs using ISC-AERMOD View then all the terrain heights will always be in Meters. ISC-AERMOD View can read DEMs in meters and in Feet (together or separately), but the resulting elevations are always output to meters.

Flagpole Receptors

The **Flagpole Receptors** option allows you to specify receptor heights above local ground level. Two options are available:



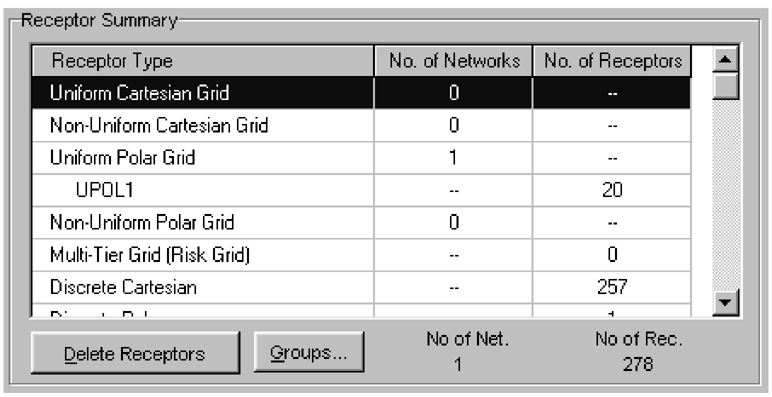
- **No:** This option indicates that flagpole receptors are not going to be specified in the Receptor Pathway. This is the default option, and assumes a default receptor height of 0.0 m.

- **Yes:** This option indicates that flagpole receptors are allowed in the Receptor Pathway. If this option is selected, then you can also specify a default flagpole receptor height other than 0.0 m. If no default height is specified than the model uses 0.0 m as the default.

Note: Any **Flagpole Heights** that are entered in the Receptor Pathway are ignored if the **No** option was selected.

Receptor Summary Table

The **Receptor Summary Table** summarizes the type of receptors, number of grids, and total number of receptors already specified for the current project.



You can select a particular receptor type from the **Receptor Summary Table**; double-click on it and the window for that particular receptor type will be displayed.

This button allows you 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, press the Shift key while selecting the receptors.

Receptor Groups (Optional)

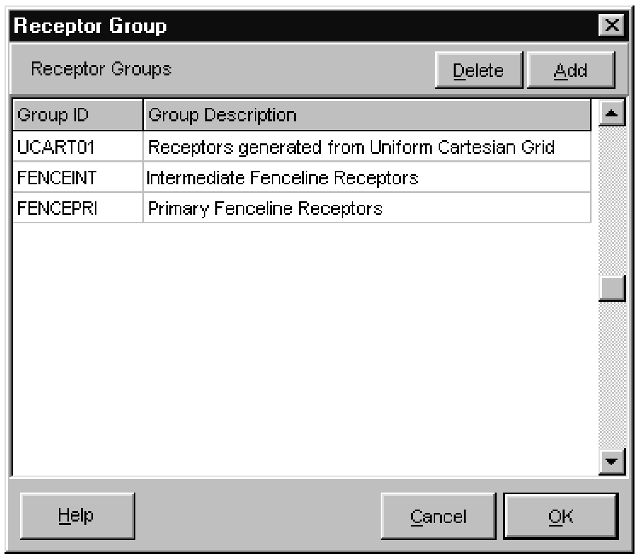
Receptor Group is a feature of ISC-AERMOD View that allows you to arrange receptors in groups. Some groups are created automatically, as is the case of the Risk Grid, Fenceline Grid, Fenceline Primary Receptors, and Fenceline Secondary Receptors. Also, when converting network grids to discrete Cartesian 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. See below the steps you should take to create groups or delete groups.

➤ **How to Create and Delete Receptor Groups:**

Step 1: Click on the **Groups** button (

). The **Receptor Group** dialog box is displayed.

Step 2: Click on the **Add** button and type in the **Group ID** and a **Group Description** (optional). You can add as many groups as you wish.



Step 3: To delete a group, you must select the group by clicking with the left mouse button and then press the **Delete** button. Note that if you delete a group, all the receptors in this group will also be deleted.

AERMAP Options (AERMOD Only)

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.

Height scale is the terrain height and location that has the greatest influence on dispersion for an individual receptor.

To run AERMAP, you need to specify the Digital Elevation Model (DEM) data, from the United States Geological Survey (USGS), for the area you are modeling. You can use the following DEM data:

- **7.5-Minute DEM data:** This data can be purchased from USGS or a digital terrain data supplier.

- **1-Degree DEM data:** This data is available through the USGS Internet site.

The original U.S. EPA AERMAP is set with the following parameter limits:

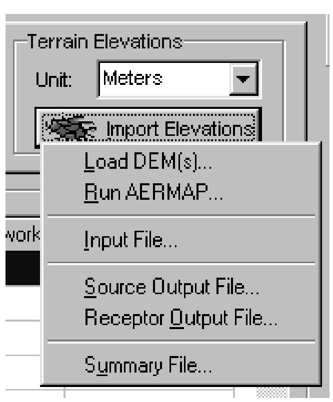
Parameter	Description	Limit
NREC	Maximum number of receptors	2500
IXM	Maximum number of x-coordinates (or distance) values per receptor network	100
IYM	Maximum number of y-coordinates (or distance) values per receptor network	100
NNET	Maximum number of receptor networks	10
NDEM	Maximum number of DEM terrain data files	100
MAXNOD	Maximum number of nodes per profile in a DEM file	1220
MAXPRF	Maximum number of profiles per DEM file	1220
NARC	Maximum number of receptor arcs for EVALCART receptors	50
NSRC	Maximum number of source locations	500

The user can adjust these limits and recompile the Fortran code for AERMAP to get a new executable.

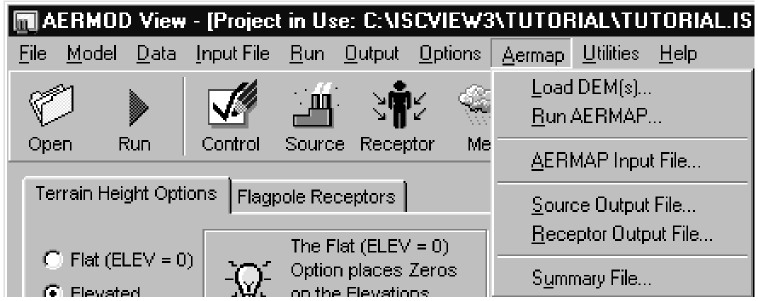
- **How to Run AERMAP:**

Step 1: Specify all your sources and receptors. Make sure you are in the **AERMOD** mode.

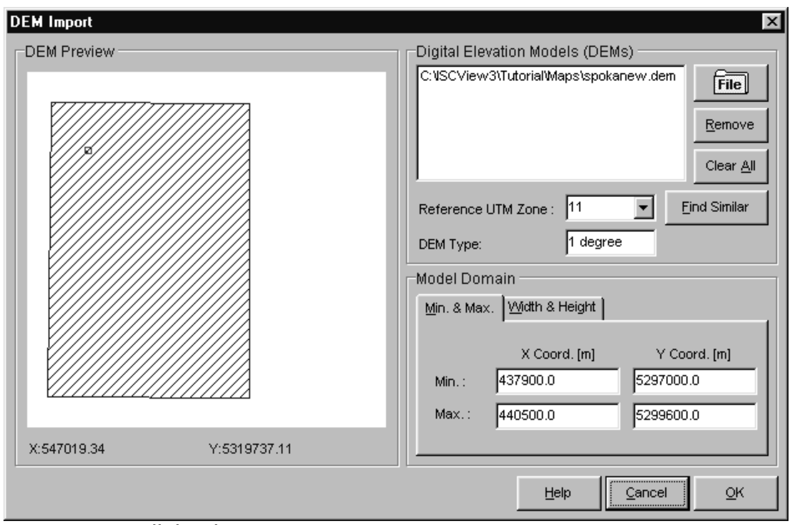
Step 2: Press the **Receptor** menu toolbar button. The **RE-Receptor Summary** window is displayed. Click on the **Import Elevations** button and select the **Load DEM(s)** option from the popup menu.



Step 3: The AERMAP options are also available through the menu.

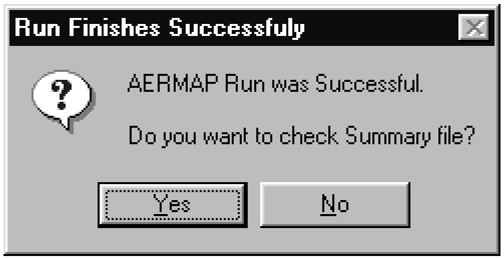


Step 4: The **DEM Import** dialog box is displayed. Press the **File** button to locate the DEM files for your modeling area. The selected DEM(s) are then shown in the **DEM Preview** area outlined in blue and hatched. ISC-AERMOD View automatically draws a green rectangle representing your current modeling domain extents. These extents include the location of all the sources and receptors already defined in your project. Press the **OK** button to close the dialog.



DEM Import dialog box

Step 5: Now that you specified the DEM data for your modeling area, it is time to run the AERMAP model. Click on the **AERMAP** button and select the **Run AERMAP** option from the popup menu. DOS windows are displayed and the AERMAP model is run. When AERMAP finishes running, a message is displayed asking if you want to check the Summary file. 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.



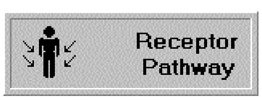
Step 6: AERMAP also produces two other files:

- **The Source Output File:** This file contains the calculated base elevations for all your sources.
- **The Receptor Output File:** This file contains the calculated terrain elevations and scale height for each receptor. This file is used as an INCLUDED file in the input runstream file (AERMOD Input File) for the Receptor Pathway.

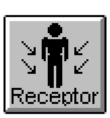
Comments (Optional)

This option allows you to add any extra comments you feel are necessary to the Receptor Pathway portion of the input file.

RE - Uniform Cartesian Grid Window

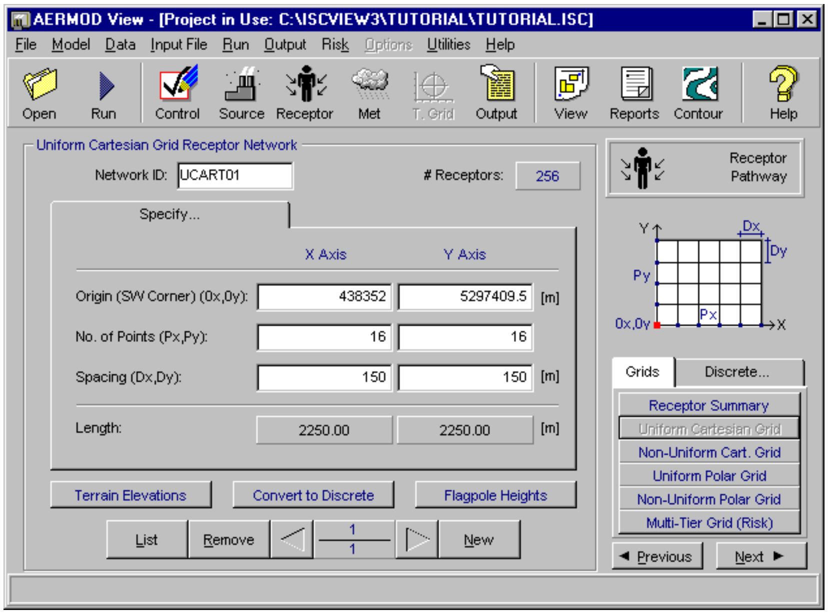


In the **RE - Uniform Cartesian Grid** window, you can define Cartesian grid receptor networks with uniform grid spacing. You have access to the **RE - Uniform Cartesian Grid** window by pressing the **Receptor** button located on the menu toolbar and then pressing the **Uniform Cartesian Grid** button located on the lower right side of any Receptor (RE) Pathway window. From the menu select **<u>D</u>ata | <u>R</u>eceptor | <u>N</u>etwork Types | <u>U</u>niform Cartesian Grid**.



+ 

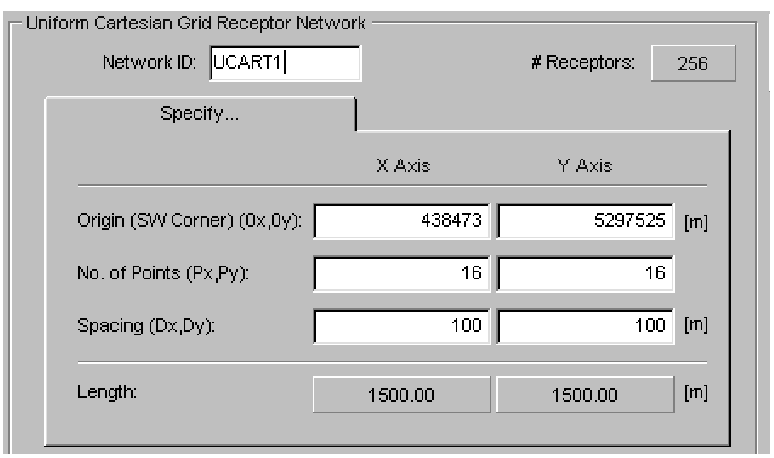
= Displays the **RE-Uniform Cartesian Grid** window



RE-Uniform Cartesian Grid window

Defining Uniform Cartesian Grid Receptor Networks

The following parameters are necessary to define a Uniform Cartesian Grid Receptor Network:



- **Network ID:** This is the identification code for the receptor network and can be up to eight alphanumeric characters. ISC-AERMOD View automatically creates default IDs (e.g., UCART01, UCART02, etc.). These IDs, however, can be changed at any time.

- **Origin (SW Corner) (Ox, Oy):** This is the X (east-west) and Y (north-south) coordinates of the grid origin (Southwest corner) in meters.

- **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 ISC-AERMOD View.

- **# Receptors:** This field displays the total number of receptors defined for the current receptor grid network.

- **Terrain Elevations:** Receptor elevations are optional and are only needed if the **Elevated** terrain option was selected. If the **Flat** terrain option was selected instead, then the **Terrain Elevations** button will become disabled, not allowing you to define terrain elevations. To specify terrain elevations, press the **Terrain Elevations** button.

Note: If no terrain elevations are defined when you are modeling with **Elevated** terrain, then the elevations will default to 0.0 meters.

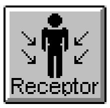
- **Flagpole Heights [m]:** This is the receptor height above ground in meters. Flagpole receptor heights are optional and are only needed if the **Yes** option was selected for the **Flagpole Receptors** option. If the **No** option was selected instead, then the **Flagpole Heights** button will become disabled, not allowing you to define flagpole receptor heights. To specify flagpole heights, press the **Flagpole Heights** button.

Note: 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.

- **Convert to Discrete:** Press this button if you want to covert the current receptor network 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, e.g., eliminate receptors within plant boundary.

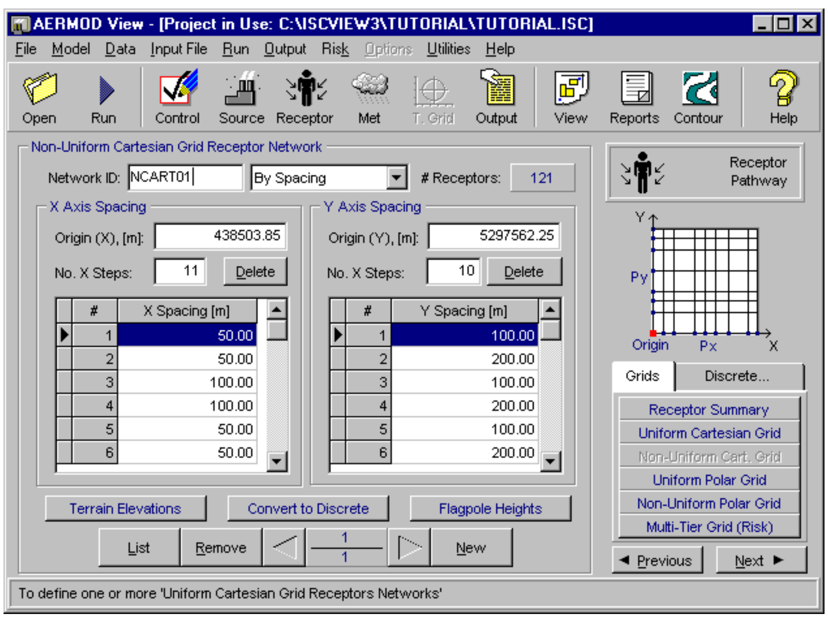
RE - Non-Uniform Cartesian Grid Window

In the **RE-Non-Uniform Cartesian Grid** window, you can define Cartesian grid receptor networks with non-uniform grid spacing. You have access to the **RE–Non-Uniform Cartesian Grid** window by pressing the **Receptor** button located on the menu toolbar and then the **Non-Uniform Cart. Grid** button located on the lower right side of any Receptor (RE) Pathway window. From the menu select **Data | Receptor | Network Types | Non-Uniform Cart. Grid**.



+


= Displays the **RE–Non-Uniform Cartesian Grid** window.



RE-Non-Uniform Cartesian Grid window

Defining Non-Uniform Cartesian Grid Receptor Networks

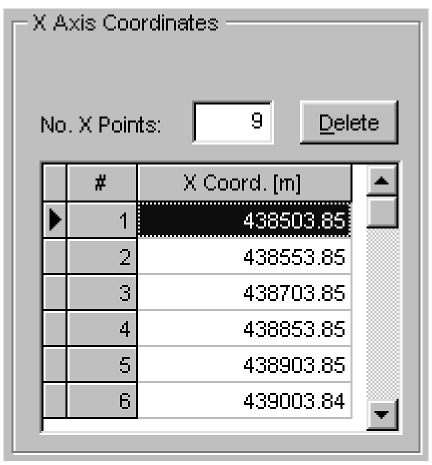
The following parameters are necessary to define a Non-Uniform Cartesian Grid Receptor Network:

- **Network ID:** This is the identification code for the receptor network and can be up to eight alphanumeric characters. ISC-AERMOD View automatically creates default

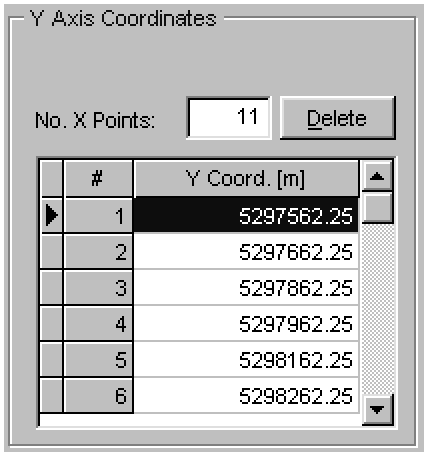
IDs (e.g., NCART01, NCART02, etc.). These IDs, however, can be changed at any time.

- **Input by:** Two types of inputs are accepted, by **Coordinates** and by **Spacing**. If you enter values for the coordinates of each x- axis and y-axis points then the spacing is automatically calculated and vice-versa.

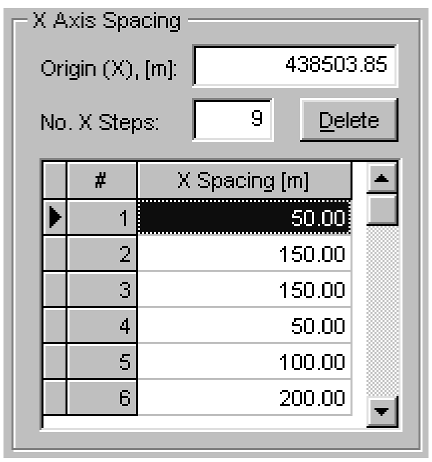
- **X Axis Coordinates:** These are the location values in meters for the first x-coordinate to the 'n-th' x-coordinate. To input the location values in the table cells, you must first specify the number of x-coordinate points on the **No. X Points** field. The table will automatically be divided with the number of rows equal to the number of x points. You must then input all the x-coordinate location values on the table. The **Delete** button is used to delete the highlighted (selected) x-axis point (row) from the table.



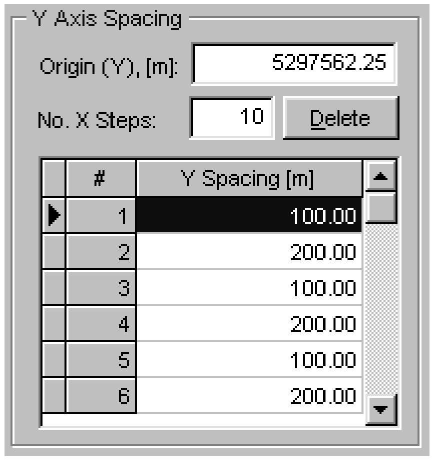
- **Y Axis Coordinates:** These are the location values in meters for the first y-coordinate to the 'n-th' y-coordinate. To input the location values in the table cells, you must first specify the number of y-coordinate points on the **No. Y Points** field. The table will automatically be divided with the number of rows equal to the number of y points. You must then input all the y-coordinate location values on the table. The **Delete** button is used to delete the highlighted (selected) y-axis point (row) from the table.



- **X Axis Spacing:** If the input type was selected to be defined **By Spacing** then the x-coordinate for the origin and the number 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. The **Delete** button is used to delete the highlighted (selected) x-axis spacing (row) from the table.



- **Y Axis Spacing:** If the input type was selected to be defined **By Spacing** then the y-coordinate for the origin and the number 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. The **Delete** button is used to delete the highlighted (selected) y-axis spacing (row) from the table.



- **# Receptors:** This field displays the total number of receptors defined for the current receptor grid network.

- **Terrain Elevations:** Receptor elevations are optional and are only needed if the **Elevated** terrain option was selected. If the **Flat** terrain option was selected instead, then the **Terrain Elevations** button will become disabled, not allowing you to define terrain elevations. To specify terrain elevations, press the **Terrain Elevations** button.

Note: If no terrain elevations are defined when you are modeling with **Elevated** terrain, then the elevations will default to 0.0 meters.

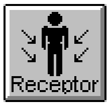
- **Flagpole Heights [m]:** This is the receptor height above ground in meters. Flagpole receptor heights are optional and are only needed if the **Yes** option was selected for the **Flagpole Receptors** option. If the **No** option was selected instead, then the **Flagpole Heights** button will become disabled, not allowing you to define flagpole receptor heights. To specify flagpole heights, press the **Flagpole Heights** button.

Note: 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.

- **Convert to Discrete:** Press this button if you want to covert the current receptor network 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 (i.e., eliminate receptors within plant boundary).

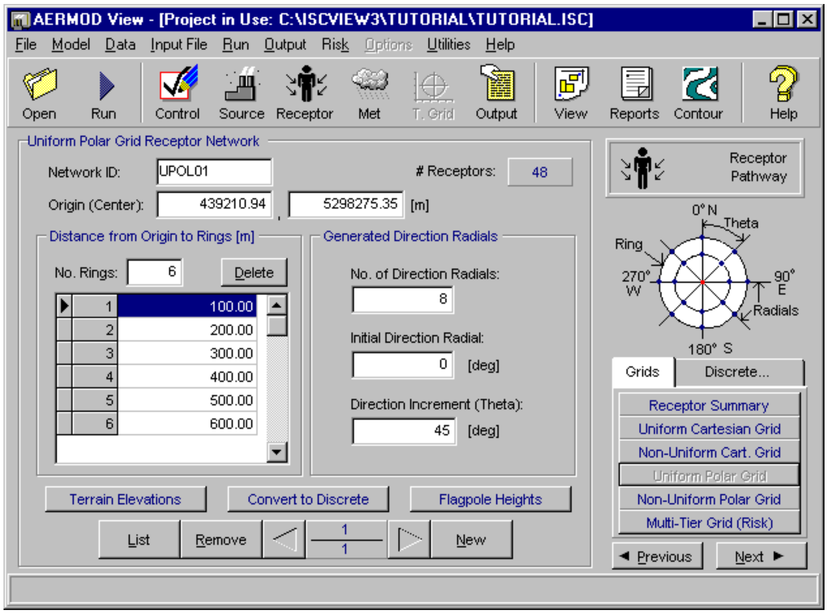
RE - Uniform Polar Grid Window

In the **RE - Uniform Polar Grid** window, you can define polar grid receptor networks with uniform grid spacing. You have access to the **RE – Uniform Polar Grid** window by pressing the **Receptor** button located on the menu toolbar and then the **Uniform Polar Grid** button located on the lower right side of any Receptor (RE) Pathway window. From the menu select **Data | Receptor | Network Types | Uniform Polar Grid**.



+


= **RE–Uniform Polar Grid** window.



RE-Uniform Polar Grid window

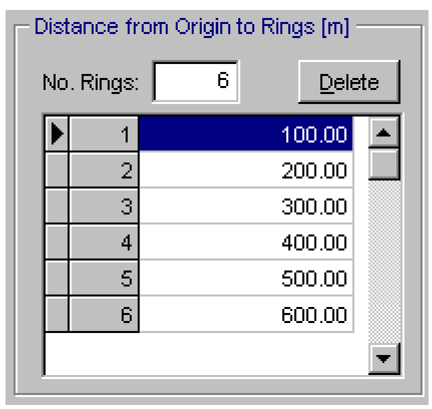
Defining Uniform Polar Grid Receptor Networks

The following parameters are necessary to define a Uniform Polar Grid Receptor Network:

- **Network ID:** This is the identification code for the receptor network and can be up to eight alphanumeric characters. ISC-AERMOD View automatically creates default IDs (e.g., UPOL01, UPOL02, etc.). These IDs, however, can be changed at any time.

- **Origin (Center):** You must specify the x- and y-coordinates in meters for the origin (center) of the grid.

- **Distance from Origin to Rings:** These are 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 should then input all the distances from the origin to the rings in the table cells. The **Delete** button is used to delete the highlighted (selected) ring (row) from the table.

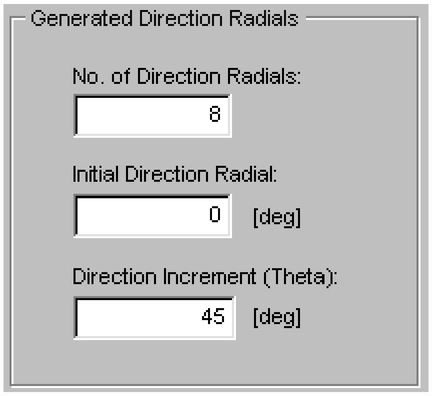


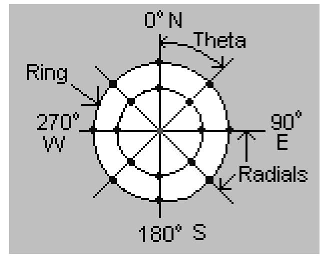
- **Generated Direction Radials:** You also need to specify the direction radials for the polar network by supplying the following information:

<u>No. of Directions Radials</u>: This is the number of directions used to define the polar system.

<u>Starting Direction Radial</u>: This is the starting direction of the polar network in degrees.

<u>Direction Increment (Theta)</u>: This is the increment in degrees for defining directions.





- **# Receptors:** This field displays the total number of receptors defined for the current receptor grid network.

- **Terrain Elevations:** Receptor elevations are optional and are only needed if the **Elevated** terrain option was selected. If the **Flat** terrain option was selected instead, then the **Terrain Elevations** button will become disabled, not allowing you to define terrain elevations. To specify terrain elevations, press the **Terrain Elevations** button.

Note: If no terrain elevations are defined when you are modeling with **Elevated** terrain, then the elevations will default to 0.0 meters.

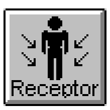
- **Flagpole Heights [m]:** This is the receptor height above ground in meters. Flagpole receptor heights are optional and are only needed if the **Yes** option was selected for the **Flagpole Receptors** option. If the **No** option was selected instead, then the **Flagpole Heights** button will become disabled, not allowing you to define flagpole receptor heights. To specify flagpole heights, press the **Flagpole Heights** button.

Note: 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.

- **Convert to Discrete:** Press this button if you want to covert the current receptor network 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, e.g., eliminate receptors within plant boundary.

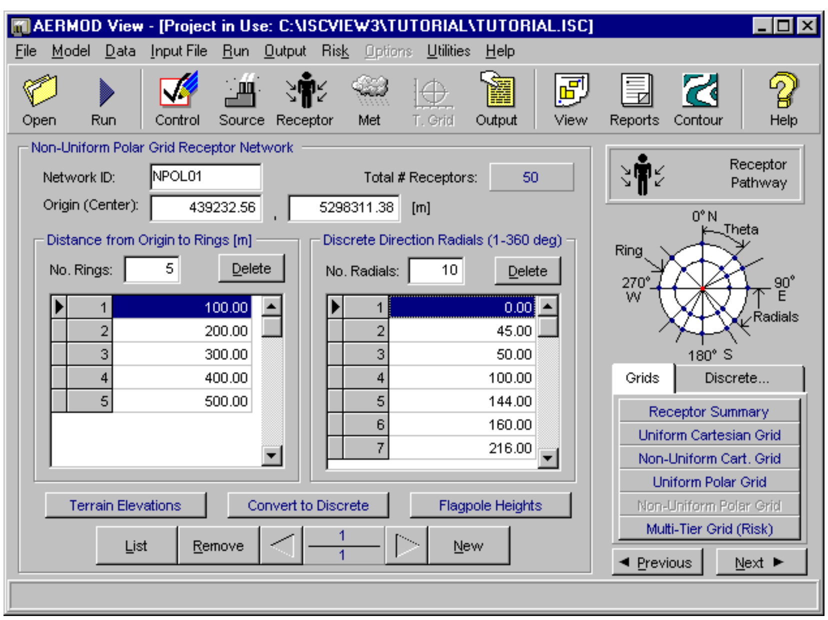
RE - Non-Uniform Polar Grid Window

In the **RE–Non-Uniform Polar Grid** window, you can define polar grid receptor networks with non-uniform grid spacing. You have access to the **RE–Non-Uniform Polar Grid** window by pressing the **Receptor** button located on the menu toolbar and then the **Non-Uniform Polar Grid** button located on the lower right side of any Receptor (RE) Pathway window. From the menu select **Data | Receptor | Network Types | Non-Uniform Polar Grid**.



+


= **RE–Non-Uniform Polar Grid** window



RE-Non-Uniform Polar Grid window

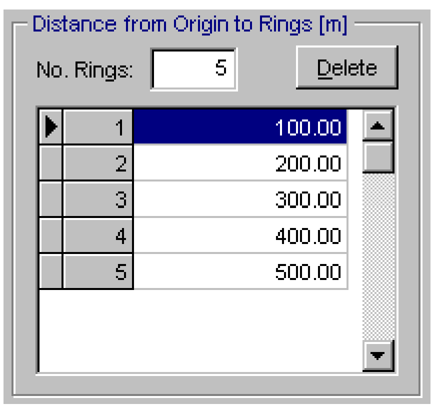
Defining Non-Uniform Polar Grid Receptor Networks

The following parameters are necessary to define a Uniform Polar Grid Receptor Network:

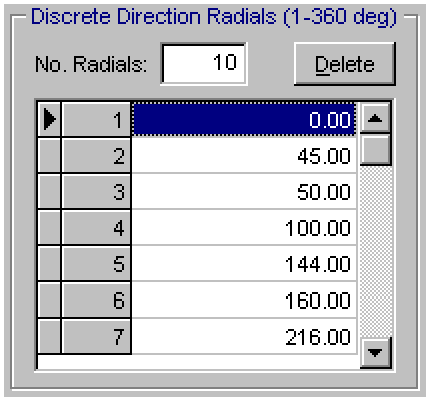
- **Network ID:** This is the identification code for the receptor network and can be up to eight alphanumeric characters. ISC-AERMOD View automatically creates default IDs (e.g., NPOL01, NPOL02, etc.). These IDs can be changed at any time.

- **Origin (Center):** You must specify the x- and y-coordinates in meters for the origin (center) of the grid.

- **Distance from Origin to Rings:** These are 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 should then input all the distances from the origin to the rings in the table cells. The **Delete** button is used to delete the highlighted (selected) ring (row) from the table.



- **Discrete Direction Radials (1 to 360 degrees):** These are 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 all the direction radials in degrees on each cell. The **Delete** button is used to delete the highlighted (selected) radial (row) from the table.



- **# Receptors:** This field displays the total number of receptors defined for the current receptor grid network.

- **Terrain Elevations:** Receptor elevations are optional and are only needed if the **Elevated** terrain option was selected. If the **Flat** terrain option was selected instead, then the **Terrain Elevations** button will become disabled, not allowing you to define terrain elevations. To specify terrain elevations, press the **Terrain Elevations** button.

Note: If no terrain elevations are defined when you are modeling with **Elevated** terrain, then the elevations will default to 0.0 meters.

- **Flagpole Heights [m]:** This is the receptor height above ground in meters. Flagpole receptor heights are optional and are only needed if the **Yes** option was selected for the **Flagpole Receptors** option. If the **No** option was selected instead, then the **Flagpole Heights** button will become disabled, not allowing you to define flagpole receptor heights. To specify flagpole heights, press the **Flagpole Heights** button.

Note: 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.

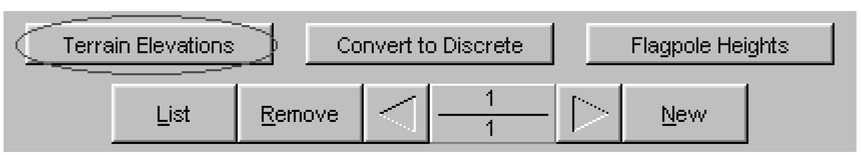
- **Convert to Discrete:** Press this button if you want to covert the current receptor network grid into discrete Cartesian receptors. This option is used when there is a need to eliminate one or more receptors from the receptor network grid, e.g., eliminate receptors within plant boundary.

Terrain Elevations for Receptor Grid Networks

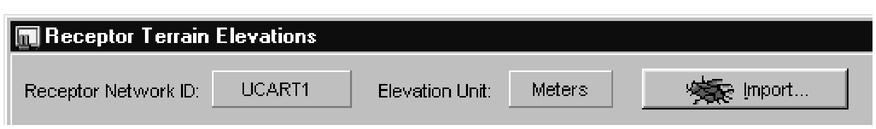
This section describes how to specify terrain elevations for the following receptor grid networks:

❒ Uniform Cartesian Grid window
❒ Non-Uniform Cartesian Grid window
❒ Uniform Polar Grid window
❒ Non-Uniform Polar Grid window

Terrain elevations for receptor grid networks are specified on the **Receptor Terrain Elevations** dialog box. You have access to this dialog box by pressing the **Terrain Elevations** button located at the bottom section of all the receptor grid network windows. If this button is disabled, this means that you are modeling with flat terrain (the **Flat** terrain option was selected).



The top section of the **Receptor Terrain Elevations** dialog box contains the following information:



- **Receptor Network ID:** This field displays the ID for the network, for which terrain elevations are to be specified.

- **Elevation Unit:** This field displays the elevation unit that should be used when specifying terrain elevations for each receptor location. This unit is the one selected in the **RE-Receptor Summary** window. Please note that if you are using the **Import** button, then the unit will always be Meters. This is due to the fact that ISC-AERMOD View always converts USGS DEMs from Feet to Meters.

- 

This button allows you to import terrain elevations from a variety of file formats: USGS DEM, UK DTM, UK NTF, and XYZ files. To import terrain elevations, you need to select the digital terrain files, define a domain, and then select the elevation import options. See more details on importing terrain elevation at the end of this chapter.

The table displayed on the **Receptor Terrain Elevations** dialog box contains the following information:

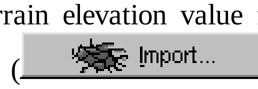
Y \ X	438743.31	438808.15	438872.99	438937.83	439002.67	439067.51	439132.35
5297889.75	0	0	0	0	0	0	0
5297950.54	0	0	0	0	0	0	0
5298011.33	0	0	0	0	0	0	0
5298072.12	0	0	0	0	0	0	0

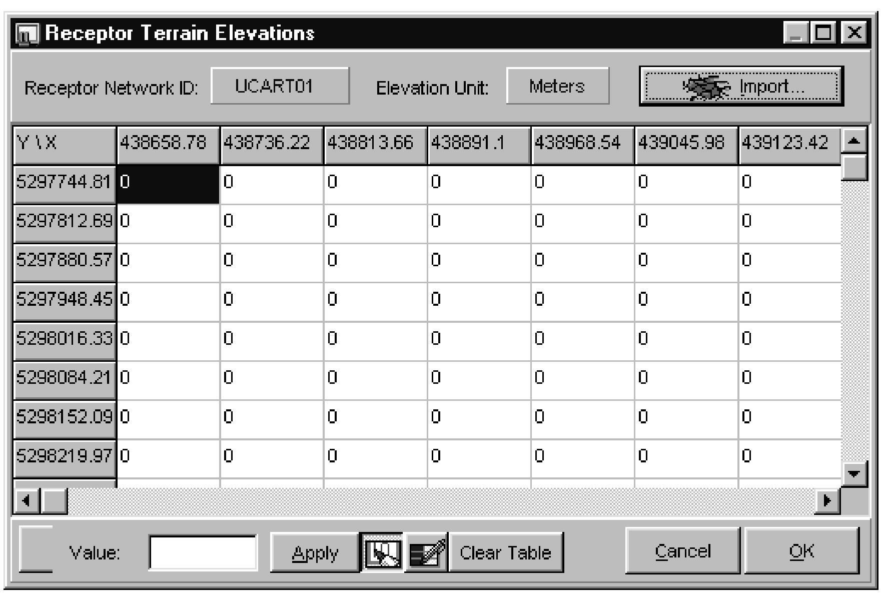
- **No. of Columns in Table:** The number of columns is defined by the number of X-axis receptors you have defined for the current grid.

- **No. of Rows in Table:** The number of rows is defined by the number of Y-axis receptors you have defined for the current grid.

- **Columns Heading:** Column headings contain the X-axis coordinate value for each point.

- **Rows Heading:** Row headings contain the Y-axis coordinate value for each point.

- **Table Cells:** Each cell of the table corresponds to one grid point (node). You should specify the terrain elevation value for each grid point location. You can use the **Import** button (
) to import terrain elevations from a variety of digital elevation files.



Y \ X	438658.78	438736.22	438813.66	438891.1	438968.54	439045.98	439123.42
5297744.81	0	0	0	0	0	0	0
5297812.69	0	0	0	0	0	0	0
5297880.57	0	0	0	0	0	0	0
5297948.45	0	0	0	0	0	0	0
5298016.33	0	0	0	0	0	0	0
5298084.21	0	0	0	0	0	0	0
5298152.09	0	0	0	0	0	0	0
5298219.97	0	0	0	0	0	0	0

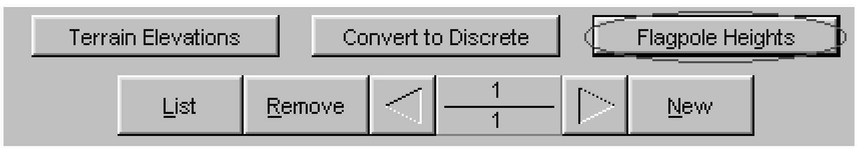
Receptor Terrain Elevations dialog box

Flagpole Heights for Receptor Grid Networks

This section describes how to specify flagpole heights for the following receptor grid networks:

- ❒ Uniform Cartesian Grid window
- ❒ Non-Uniform Cartesian Grid window
- ❒ Uniform Polar Grid window
- ❒ Non-Uniform Polar Grid window

Flagpole heights for receptor grid networks are specified on the **Flagpole Heights** dialog box. You have access to this dialog box by pressing the **Flagpole Heights** button located at the bottom section of all the receptor grid network windows. If this button is disabled, this means that you are not modeling with flagpole receptors (the **No** option was selected for the **Flagpole Receptors** option).



The top section of the **Flagpole Heights** dialog box contains the following information:

- ◆ **Receptor Network ID:** This field displays the ID for the network, for which flagpole heights are to be specified.

- ◆ **Height Unit:** This field displays the flagpole height unit. Flagpole heights are only accepted in Meters.

- ◆ **Blank cells will default to:** This field displays the default flagpole height that will be used for the cells (receptor locations) that are left blank.

The table displayed on the **Flagpole Heights** dialog box contains the following information:

Y \ X	438743.31	438808.15	438872.99	438937.83	439002.67	439067.51	439132.35	439197.19	43926
5297889.75									
5297950.54									
5298011.33									
5298072.12									

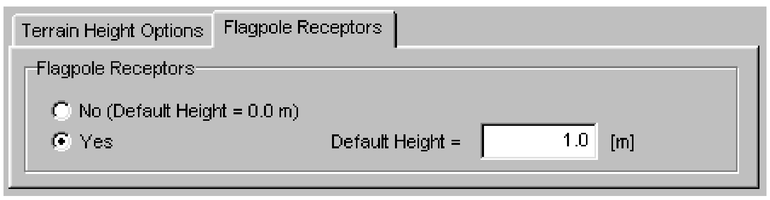
- **No. of Columns in Table:** The number of columns is defined by the number of X-axis receptors you have defined for the current grid.

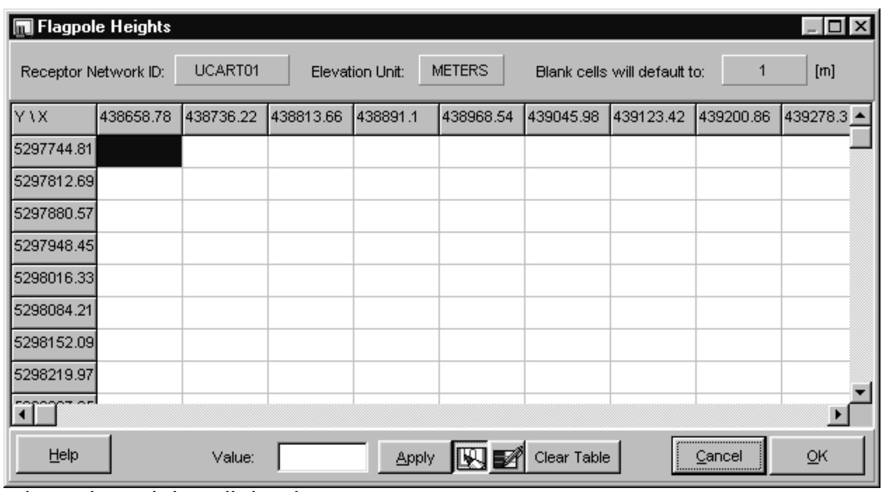
- **No. of Rows in Table:** The number of rows is defined by the number of Y-axis receptors you have defined for the current grid.

- **Columns Heading:** Column headings contain the X-axis coordinate value for each point.

- **Rows Heading:** Row headings contain the Y-axis coordinate value for each point.

- **Table Cells:** Each cell of the table corresponds to one grid point (node). You can specify flagpole heights for each grid point location. Cells left blank are assumed to have a flagpole height equal to the default flagpole height. The default flagpole height is the one that was specified for the Flagpole Receptors option on the **RE-Receptor Summary** window.



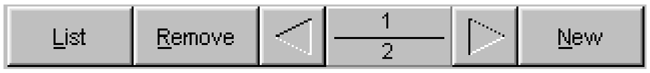


Flagpole Heights dialog box

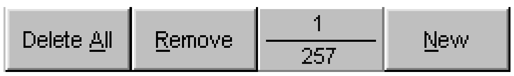
Record Navigator Buttons

The Record Navigator is used in the windows where you can specify more than one record of the same type.

In the case of receptor grid networks, the Record Navigator is composed of the following buttons:



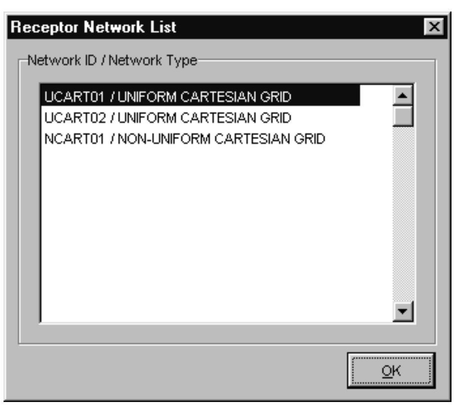
In the case of most discrete receptors, the Record Navigator is composed of the following buttons:



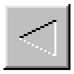
The buttons contained in the Record Navigator will vary from window to window. See below all the buttons that may be part of the Record Navigator tool:

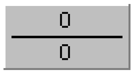
Displays a list of all the records already created for the type of information defined on the current window. In the case of receptor grid networks, this button displays the **Receptor Network List** dialog box. This dialog box contains a list of all receptor networks already defined for the current project.

Deletes all records already defined in the current window.

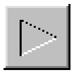


Removes the current record. The current record is the one being displayed on the current window. In cases where all the records are being displayed in the same table, the Remove button removes the record that is currently highlighted or marked with an arrow.

 Press this button to display the information that was defined for a previous record.



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

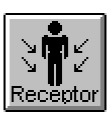
 Press this button to display the information that was defined for the next record from the list.

Allows you to create a new record.

RE – Multi-Tier Grid Window

In the **RE–Multi-Tier Grid** window, you can define a multi-tier grid. The multi-tier grid is defined by discrete Cartesian receptors, square in shape, and with origin at the center of the grid. Multi-tiers 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).

You have access to the **RE–Multi-Tier Grid** window by pressing the **Receptor** button located on the menu toolbar and then the **Multi-Tier Grid** button located on the lower right side of any Receptor (RE) Pathway window. From the menu select **Data | Receptor | Discrete Locations | Multi-Tier Grid**.



+ 

= Displays the **RE–Multi-Tier Grid** window.

Defining a Multi-Tier Grid

The **RE-Multi-Tier Grid** window consists of two tabs: the **Grid Settings** tab and **the Generated Discrete Receptors** tab. These two tabs are described in the sections that follow.

Grid Settings Tab

- 

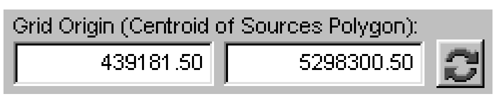
Press this button to get the default grid settings according to the HHRAP and SLERAP protocols. According to these two U.S. EPA-OSW protocols, the risk grid should be:

  1. A 100-meter spaced grid from the centroid of the emission sources out to a radius of 3 km.

  2. A 500-meter spaced grid extending from 3 km to 10 km.

  3. Origin of the grid should be the centroid of the polygon formed by all sources.

- **Grid Origin (Centroid of Sources Polygon):** You must define the X and Y coordinates for the origin (center) of the grid.



Press this button to get the centroid of the polygon formed by linking the origin of all the sources already defined in your project.

- **No. of Tiers:** Tiers are the segments with different grid spacing. You can specify the number of tiers in three ways.

  1. Typing the number in the **No. of Tiers** field.
  2. Clicking on the arrow up button, until the right number is displayed.
  3. Pressing the **Add** button and completing the requested information for each tier.

Press this button to add a new tier (segment) to the grid

Press this button to delete the selected tier (segment)

- **Distance from Center (Origin) [m]:** You must specify the distance in meters from the center of the grid (origin) to each tier.

Tier	Distance from center (origin)[m]	Tier Spacing[m]
1	3000	100
2	10000	500

- **Tier Spacing [m]:** For each segment (tier), you must specify the spacing between receptors in the X and Y directions.

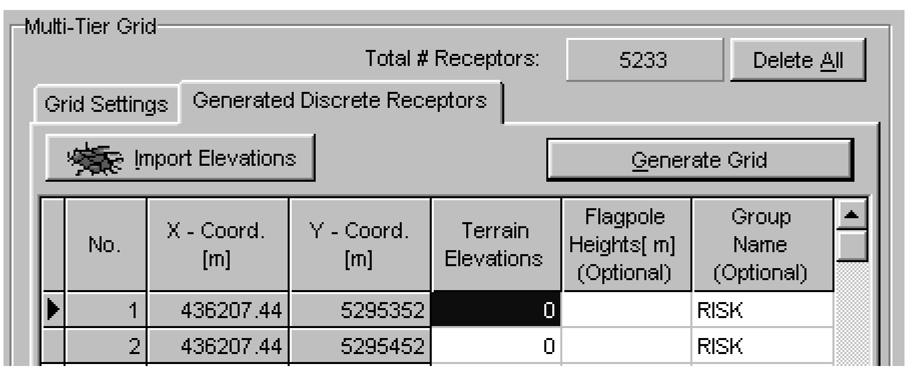
Tier	Distance from center (origin)[m]	Tier Spacing[m]
1	3000	100
2	10000	500

- Generate Grid Press this button to generate the Risk Grid. When ISC-AERMOD finishes generating the grid, the total number of receptors created is automatically displayed. Use the **Delete All** button to delete all Risk Grid receptors.

Total # Receptors: 5233 Delete All

Generated Discrete Receptors Tab

After generating the grid, the **Generated Discrete Receptors** tab is automatically displayed. Note that the X and Y coordinates for all Risk Grid receptors are automatically calculated. Some additional parameters need to be specified depending on your modeling options. These additional parameters are described below:



- **Terrain Elevations [m or ft]:** This parameter is only requested if the **Elevated** terrain option was selected. If the **Flat** terrain option was selected, then this column is hidden and any previous inputs will be ignored. The default unit is meters but may be specified in feet if you select Feet for the **Terrain Elevations Unit** option.

- **Flagpole Heights [m] (Optional):** This is the receptor height above ground in meters. This parameter is only requested if the **Flagpole Receptors** option was selected. Otherwise, this column is hidden and any previous inputs will be ignored. Any missing values in the **Flagpole Heights** column, will be interpreted by the U.S. EPA models (ISCST3, AERMOD, and ISC-PRIME) 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.

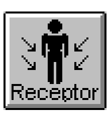
- **Group Name (Optional):** The group name, RISK, is automatically assigned to every Risk Grid receptor. This group name, however, can be changed at any time.

Note: If you are modeling with **Elevated** terrain and no value is defined in the **Terrain Elevations** column, then the missing terrain elevations will default to 0.0 meters.

Press this button if you want to import terrain elevations for all the multi-tier grid receptors. See Importing Terrain Elevations at the end of this Chapter for more information on types of files supported.

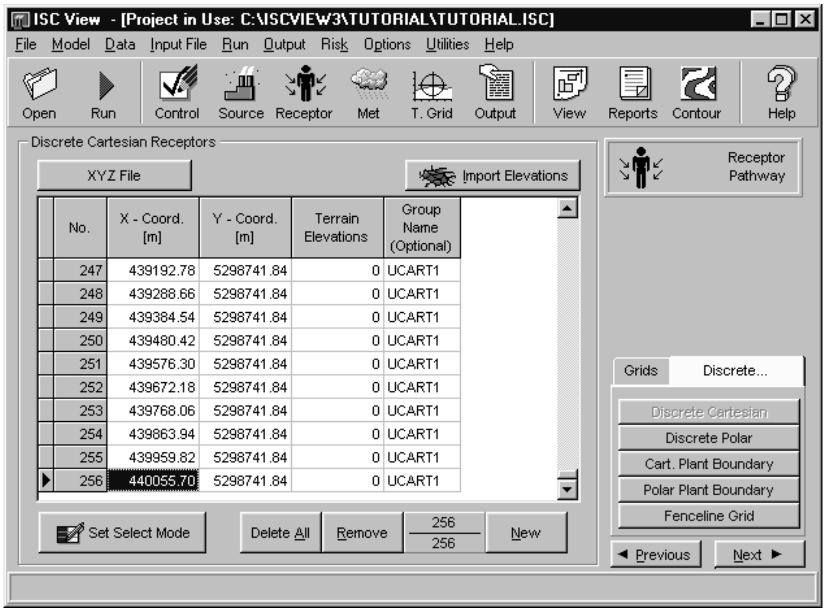
RE - Discrete Cartesian Window

In the **RE-Discrete Cartesian** window, you can define one or more discrete Cartesian receptors. You have access to the **RE-Discrete Cartesian** window by pressing the **Receptor** button located on the menu toolbar and then the **Discrete Cartesian** button located on the lower right side of any Receptor (RE) Pathway window. From the menu select **Data | Receptor | Discrete Locations | Discrete Cartesian**.



+


= **RE–Discrete Cartesian** window.



RE-Discrete Cartesian window

Defining Discrete Cartesian Receptors

The following parameters are necessary to define a discrete Cartesian receptor:

No.	X-Coord. [m]	Y-Coord. [m]	Terrain Elevations	Flagpole Heights [m]	Group Name (Optional)
246	439105.85	5298813.42	0		UCART01
247	439179.47	5298813.42	0		UCART01

- **No.:** This column displays the entry number or record number for each discrete Cartesian receptor you have defined.

- **X Coord. [m]:** This is the x (east-west) coordinate, in meters, for the receptor location.

- **Y Coord. [m]:** This is the y (north-south) coordinate, in meters, for the receptor location.

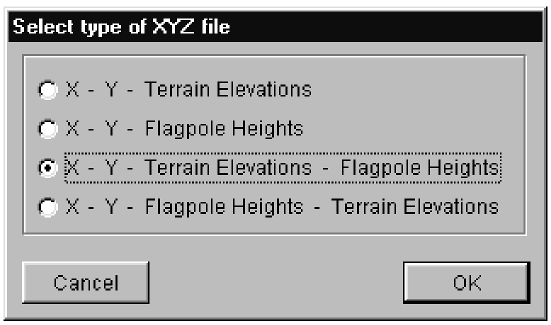
- **Terrain Elevations [m or ft]:** This parameter is only requested if the **Elevated** terrain option was selected. If the **Flat** terrain option was selected instead, then this column is hidden and any previous inputs will be ignored. The default unit is meters but may be specified in feet if you select Feet for the **Terrain Elevations Unit** option.

- **Flagpole Heights [m] (Optional):** This is the receptor height above ground in meters. This parameter is only requested if the **Flagpole Receptors** option was selected. Otherwise, this column is hidden and any previous inputs will be ignored. The U.S. EPA models (ISCST3, AERMOD, and ISC-PRIME) will interpret any missing values in the Flagpole Heights column 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.

Note: 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.

- 

Press this button to import or export the discrete Cartesian receptor parameters. ISC-AERMOD View can import text files containing the x coordinate, y coordinate, terrain elevations and/or flagpole heights. Data must be provided in columns separated by one or more blank spaces.



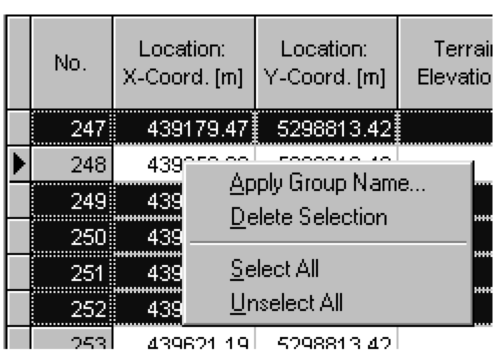
- 

Press this button if you want to import terrain elevations for all receptors already defined in this window. See the *Terrain Elevations* section at the end of this Chapter for more information on types of files supported.

- 

Press this button to turn on or off the Select Mode. The Select Mode allows you to select more than one receptor at a time, right click with the mouse, and perform one of the following operations:

1. Apply Group Name
2. Delete Selection
3. Select All
4. Unselected All



When you finish, press the **Unset Select Mode** button to toggle off the Select Mode.

- 

Deletes all receptors already defined in this window.

- 

Removes the selected (highlighted) receptors.

- 

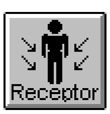
The top panel displays the entry number for the current record (indicated by an arrow ). The bottom panel displays the total number of receptors already defined in this window.

- 

Adds a new row to the table, allowing you to define a new receptor.

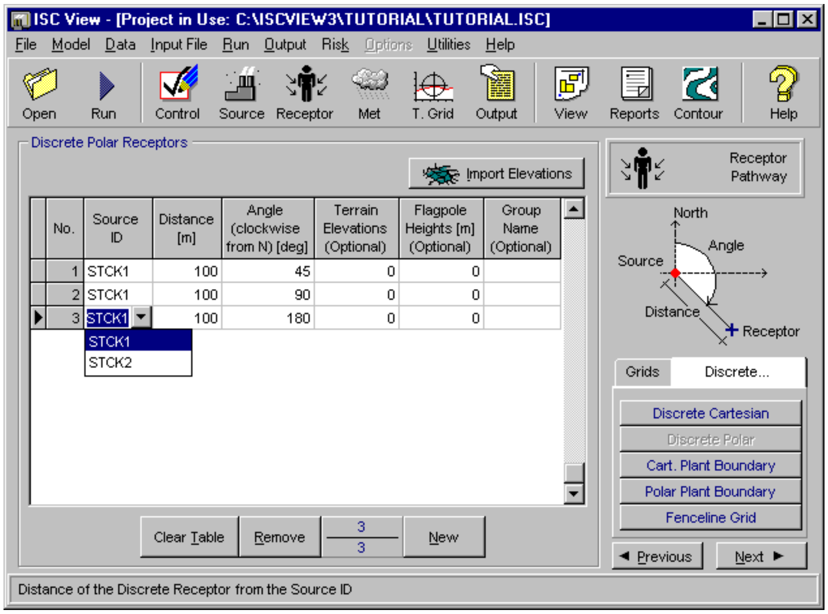
RE - Discrete Polar Window

In the **RE-Discrete Polar** window, you can define one or more discrete polar receptors. You have access to the **RE-Discrete Polar** window by pressing the **Receptor** button located on the menu toolbar and then the **Discrete Polar** button located on the lower right side of any Receptor (RE) Pathway window. From the menu select **Data | Receptor | Discrete Locations | Discrete Polar**.



+ 

= **RE–Discrete Polar** window.



RE-Discrete Polar window

Defining Discrete Polar Receptors

The following parameters are necessary to define a discrete polar receptor:

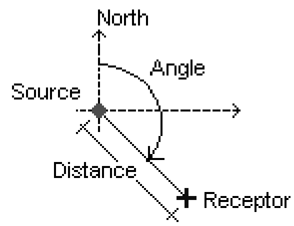
No.	Source ID	Distance [m]	Angle (clockwise from N) [deg]	Terrain Elevations (Optional)	Flagpole Heights [m] (Optional)	Group Name (Optional)
1	STCK1	100	45	0	0	
2	STCK1	100	90	0	0	
3	STCK1	100	180	0	0	
	STCK1					
	STCK2					

- **No.:** This column displays the entry number or 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. ISC-AERMOD View automatically supplies you with the list of the **Source IDs** for all sources defined on the **SO-Source Inputs** window. Click the cell twice to display the list of sources.

- **Distance [m]:** This is the distance, in meters, from the source to the discrete polar receptor.

- **Direction [degrees]:** This is the direction, in degrees, measured clockwise from north for the discrete receptor location.



- **Terrain Elevations [m or ft]:** This parameter is only requested if the **Elevated** terrain option was selected. If the **Flat** terrain option was selected, then this column is hidden and any previous inputs will be ignored. The default unit is meters but may be specified in feet if you select Feet for the **Terrain Elevations Unit** option.

- **Flagpole Heights [m] (Optional):** This is the receptor height above ground in meters. This parameter is only requested if the **Flagpole Receptors** option was selected. Otherwise, this column is hidden and any previous inputs will be ignored. Any missing values in the **Flagpole Heights** column will be interpreted by the U.S. EPA models (ISCST3, AERMOD, and ISC-PRIME) 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.

Note: If you are modeling with **Elevated** terrain and no value is defined in the **Terrain Elevations** column, then the missing terrain elevations will default to 0.0 meters.

- 

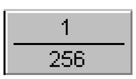
Press this button if you want to import terrain elevations for all receptors already defined in this window. See the *Terrain Elevations* section at the end of this Chapter for more information on types of files supported.

- 

Deletes all receptors already defined in this window.

- 

Removes the selected (highlighted) receptors.

- 

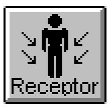
The top panel displays the entry number for the current record (indicated by an arrow ). The bottom panel displays the total number of receptors already defined in this window.

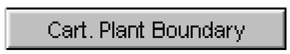
- 

Adds a new row to the table, allowing you to define a new receptor.

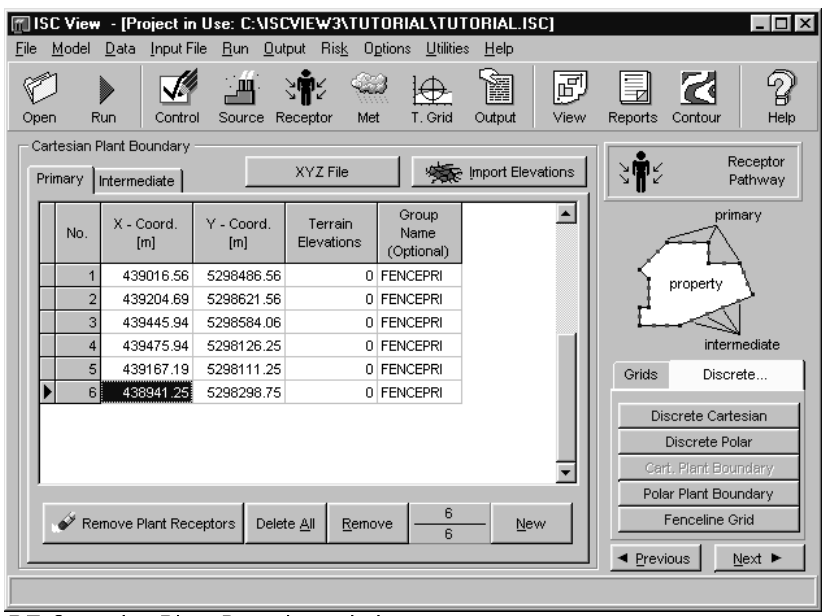
RE - Cartesian Plant Boundary Window

In the **RE-Cartesian Plant Boundary** window, you can define the boundaries of your plant (also referred as fenceline) using discrete Cartesian receptors. You have access to the **RE-Cartesian Plant Boundary** window by pressing the **Receptor** button located on the menu toolbar and then the **Cart. Plant Boundary** button located on the lower right side of any Receptor (RE) Pathway window. From the menu select **Data | Receptor | Discrete Locations | Cartesian Plant Boundary**.



+


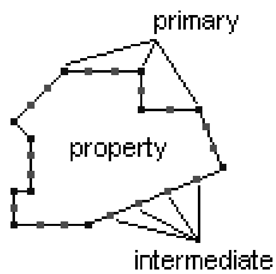
= **RE–Cartesian Plant Boundary** window.



RE-Cartesian Plant Boundary window

Defining Cartesian Plant Boundary Receptors

You define your plant boundary (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.



Primary Fenceline Receptors

The following parameters are necessary to define the primary fenceline receptors:

Primary | Intermediate

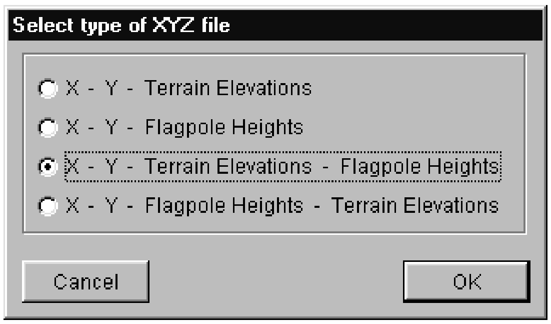
No.	X-Coord. [m]	Y-Coord. [m]	Terrain Elevations	Flagpole Heights [m] (Optional)	Group Name (Optional)
1	439032.75	5298534	0	0	FENCEPRI
2	439403.75	5298534	0	0	FENCEPRI
3	439543.5	5298187.5	0	0	FENCEPRI
4	439349	5297968.75	0	0	FENCEPRI
5	438990.25	5298011.25	0	0	FENCEPRI

- **No.:** This column displays the entry number or record number for each fenceline node (discrete Cartesian receptor) you have defined.
- **X Coord. [m]:** This is the x (east-west) coordinate, in meters, for the node location.
- **Y Coord. [m]:** This is the y (north-south) coordinate, in meters, for the node location.
- **Terrain Elevations [m or ft]:** This parameter is only requested if the **Elevated** terrain option was selected. If the **Flat** terrain option was selected, then this column is hidden and any previous inputs will be ignored. The default unit is meters but may be specified in feet if you select Feet for the **Terrain Elevations Unit** option.
- **Flagpole Heights [m] (Optional):** This is the receptor height above ground in meters. This parameter is only requested if the **Flagpole Receptors** option was selected. Otherwise, this column is hidden and any previous inputs will be ignored. Any missing values in the **Flagpole Heights** column will be interpreted by the U.S. EPA models (ISCST3, AERMOD, and ISC-PRIME) 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 (Optional):** The group name, FENCEPRI, is automatically assigned every time you add a new fenceline node. This group name can be changed at any time.

Note: If you are modeling with **Elevated** terrain and no value is defined in the **Terrain Elevations** column, then the missing terrain elevations will default to 0.0 meters.

- 

Press this button to import or export the fenceline receptor parameters. ISC-AERMOD View can import text files containing the x coordinate, y coordinate, terrain elevations and/or flagpole heights. Data must be provided in columns separated by one or more blank spaces.



- 

Press this button if you want to import terrain elevations for all the fenceline receptors already defined in this window. See the *Terrain Elevations* section at the end of this Chapter for more information on types of files supported.

- 

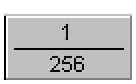
Press this button if you want to remove all discrete receptors within plant boundary. Any receptor grid networks or polar plant boundary receptors will not be deleted. For these receptors, you need to convert them to discrete receptors before being able to remove them from within plant boundary.

- 

Deletes all receptors already defined in this window.

- 

Removes the selected (highlighted) receptors.

- 

The top panel displays the entry number for the current record (indicated by an arrow ). The bottom panel displays the total number of receptors already defined in this window.

- 

Adds a new row to the table, allowing you to define a new receptor.

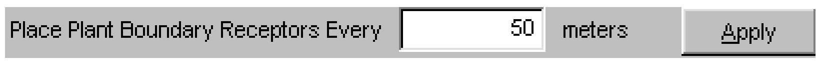
Intermediate Fenceline Receptors

Intermediate fenceline receptors can be easily assigned to your fenceline by doing the following:

Step 1: Specify the primary receptors for your fenceline.

Step 2: Click the **Intermediate** tab.

Step 3: Specify the spacing between intermediate receptors in meters and press the **Apply** button.



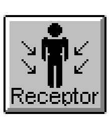
Step 4: Discrete receptors will be automatically placed between nodes, equally spaced. The distance between receptors will be rounded to an even multiple of the spacing you have specified. As an example, if between two fenceline nodes you have a distance of 100 meters and the specified spacing is 30 meters, then 3 intermediate receptors will be placed between the two nodes (corners) spaced every 25 meters.

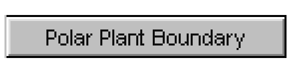
RE - Polar Plant Boundary Window

(ISCST3 and ISC-PRIME Only)

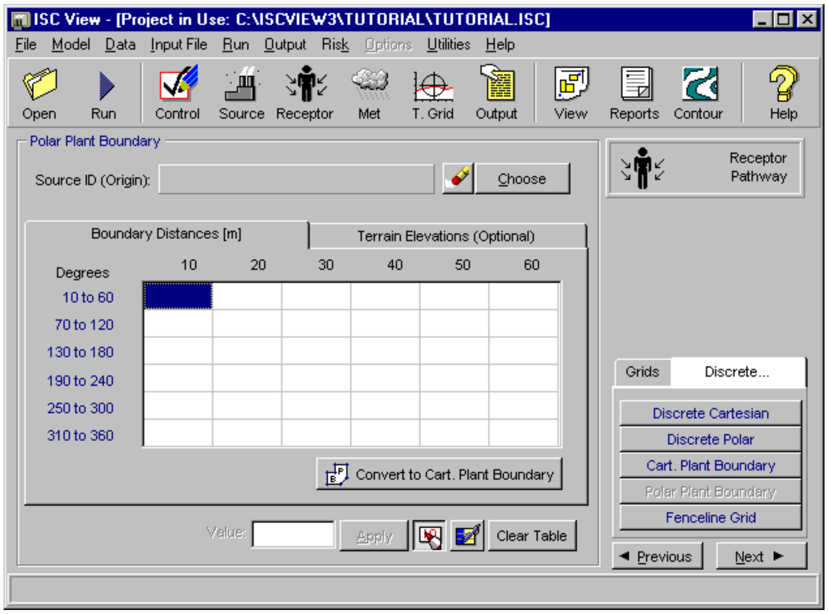
In the **RE-Polar Plant Boundary** window, you can define the boundaries of your plant (fenceline) using polar plant boundary receptors. For the polar plant boundary, you must define the distance from a specific source to the fenceline every 10 degrees for 36 directions. The Polar Plant Boundary option is not applicable to AERMOD.

You have access to the **RE-Polar Plant Boundary** window by pressing the **Receptor** button located on the menu toolbar and then the **Cart. Plant Boundary** button located on the lower right side of any Receptor (RE) Pathway window. From the menu select **Data | Receptor | Discrete Locations | Polar Plant Boundary**.



+


= **RE–Polar Plant Boundary** window.



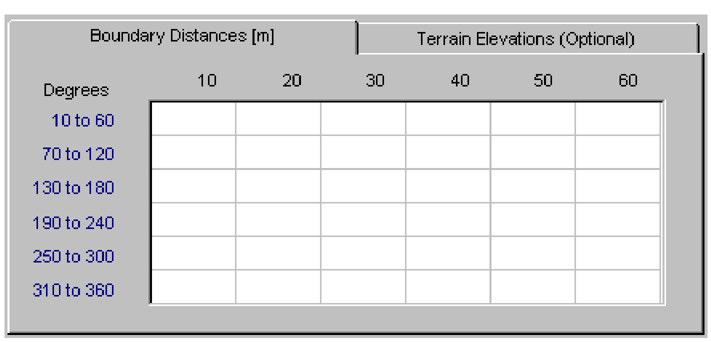
RE-Polar Plant Boundary window

Defining Plant Boundary Distances and Elevations

To define polar plant boundary distances and elevations, 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 **Choose** button to display the **Source ID** dialog box, and select one of the sources from the drop-down list box.

- **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, ISC-AERMOD View displays the degrees for each direction on the table heading.



Boundary Distances table

- **Terrain Elevations (Optional):** These are the terrain elevations for each of the 36 boundary receptor points. The default unit is meters however you can use feet, if **Feet** was defined as the **Terrain Elevations Unit**. Terrain elevations are only needed if the **Elevated** terrain option was selected.

Boundary Distances [m] | Terrain Elevations (Optional)

Degrees	10	20	30	40	50	60
10 to 60	0	0	0	0	0	0
70 to 120	0	0	0	0	0	0
130 to 180	0	0	0	0	0	0
190 to 240	0	0	0	0	0	0
250 to 300	0	0	0	0	0	0
310 to 360	0	0	0	0	0	0

Terrain Elevations table (Optional)

Note: 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.

Note: For applications where a uniform flagpole receptor height is used for all receptors, which can be specified on the **Flagpole Receptors** option, those flagpole receptor heights will also apply to any boundary receptors defined as Polar Plant Boundary Receptors.

Press this button, if you want to convert Polar Plant Boundary Receptors into Cartesian Plant Boundary Receptors. Use this option when you need to eliminate receptors within plant boundary. ISC-AERMOD View only eliminates receptors within a plant boundary that was specified as **Cartesian Plant Boundary Receptors**.

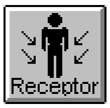
RE – Discrete Cartesian (ARC) Window

(AERMOD Only)

In the **RE-Discrete Cartesian (ARC)** window, you can define discrete Cartesian receptor locations similar to the ones defined on the **RE-Discrete Cartesian** window. The discrete Cartesian (ARC) receptors option is only applicable to AERMOD and is the only option that the grouping of receptors (Receptor Group option) is used by the U.S. EPA model.

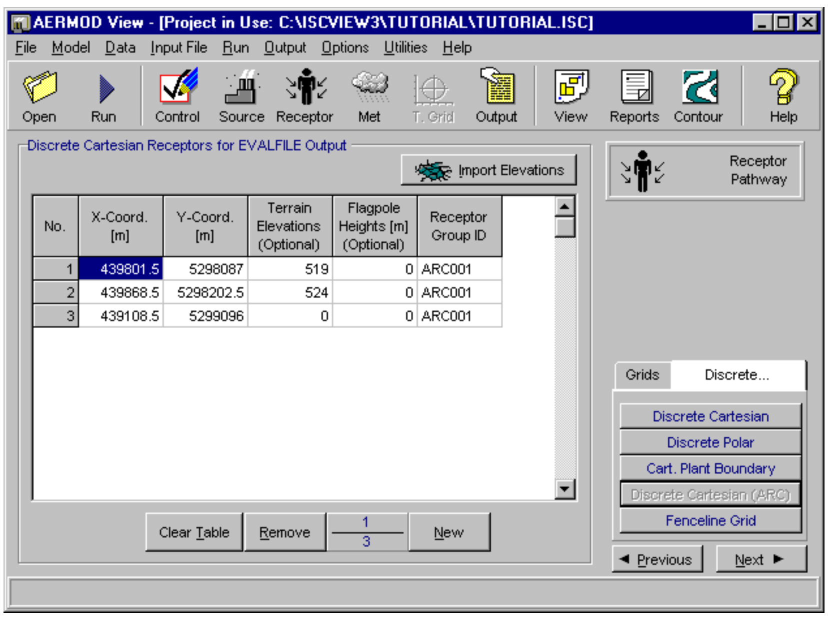
This discrete Cartesian (ARC) receptors option is designed to be used with the EVALFILE option described on the **OU-Evaluation Files** window. If the discrete Cartesian (ARC) receptors option is used without the use of the EVALFILE option, then the receptor grouping is ignored.

You have access to the **RE-Discrete Cartesian (ARC)** window by pressing the **Receptor** button located on the menu toolbar and then the **Discrete Cartesian (ARC)** button located on the lower right side of any Receptor (RE) Pathway window. From the menu select **Data | Receptor | Discrete Locations | Discrete Cartesian (ARC)**.



+

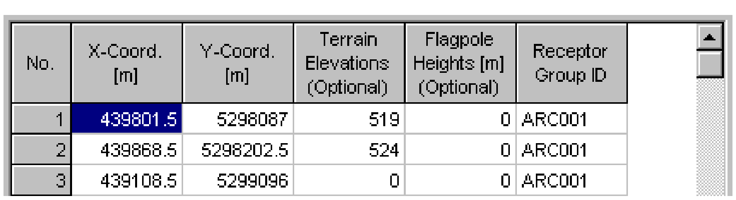

= **Discrete Cartesian (ARC)** window.



RE-Discrete Cartesian (ARC) window

Defining Discrete Cartesian (ARC) Receptors

The following parameters are necessary to define discrete Cartesian (ARC) receptors:



No.	X-Coord. [m]	Y-Coord. [m]	Terrain Elevations (Optional)	Flagpole Heights [m] (Optional)	Receptor Group ID
1	439801.5	5298087	519	0	ARC001
2	439868.5	5298202.5	524	0	ARC001
3	439108.5	5299096	0	0	ARC001

- **No.:** This column displays the entry number or record number for each discrete Cartesian (ARC) receptor you have defined.

- **X Coord. [m]:** This is the x (east-west) coordinate, in meters, for the receptor location.

- **Y Coord. [m]:** This is the y (north-south) coordinate, in meters, for the receptor location.

- **Terrain Elevations [m or ft]:** This parameter is only requested if the **Elevated** terrain option was selected. If the **Flat** terrain option was selected instead, then this column is hidden and any previous inputs will be ignored. The default unit is meters but may be specified in feet if you select Feet for the **Terrain Elevations Unit** option.

- **Flagpole Heights [m] (Optional):** This is the receptor height above ground in meters. This parameter is only requested if the **Flagpole Receptors** option was selected. Otherwise, this column is hidden and any previous inputs will be ignored. Any missing values in the **Flagpole Heights** column, will be considered to be 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:** This is the receptor grouping identification, which may be up to eight characters long. The receptor group ID can be used to group receptors by arc.

Note: If you are modeling with **Elevated** terrain and no value is defined in the **Terrain Elevations** column, then the missing terrain elevations will default to 0.0 meters.

- 

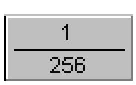
Press this button if you want to import terrain elevations for all receptors already defined in this window. See the *Terrain Elevations* section at the end of this Chapter for more information on types of files supported.

- 

Deletes all receptors already defined in this window.

- 

Removes the selected (highlighted) receptors.

- 

The top panel displays the entry number for the current record (indicated by an arrow). The bottom panel displays the total number of receptors already defined in this window.

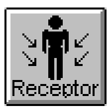
- 

Adds a new row to the table, allowing you to define a new receptor.

RE - Fenceline Grid Window

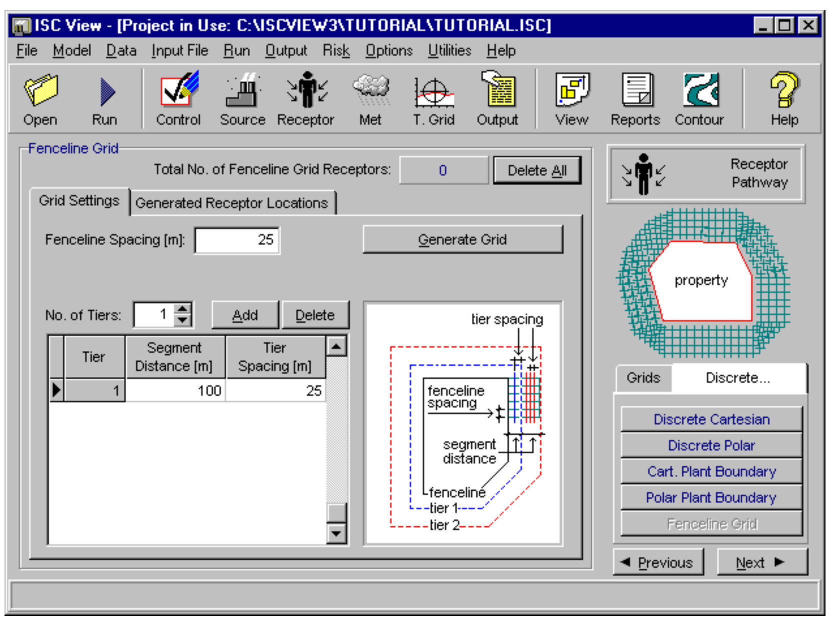
In the **RE-Fenceline Grid** window, you can define a grid around your Cartesian Plant Boundary (fenceline). The Fenceline grid can have more than one tier (segment) with different grid spacings. Fenceline grid receptors are modeled in ISCST3, AERMOD, and ISC-PRIME as Discrete Cartesian Receptors.

You have access to the **RE-Fenceline Grid** window by pressing the **Receptor** button located on the menu toolbar and then the **Fenceline Grid** button located on the lower right side of any Receptor (RE) Pathway window. From the menu select **Data | Receptor | Discrete Locations | Fenceline Grid**.



+

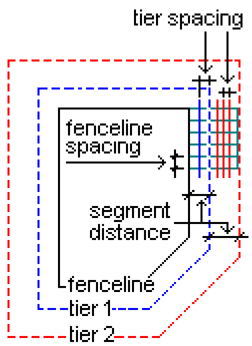

= Displays the **RE–Fenceline Grid** window.



RE-Fenceline Grid window

Defining the Fenceline Grid

The **RE-Fenceline Grid** window consists of two tabs: the **Grid Settings** tab and the **Generated Discrete Receptors** tab. These two tabs are described in the sections that follow.



Grid Settings Tab

- **Fenceline Spacing [m]:** You must define the spacing between receptors along the fenceline.

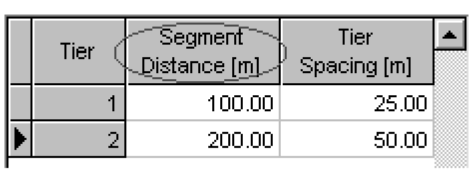
- **No. of Tiers:** Tiers are the segments with different grid spacing. You can specify the number of tiers in three ways.

1. Typing the number in the **No. of Tiers** field.
2. Clicking on the arrow up button, until the right number is displayed.
3. Pressing the **Add** button as many times as the number of tiers.

Press this button to add a new tier (segment) to the fenceline grid

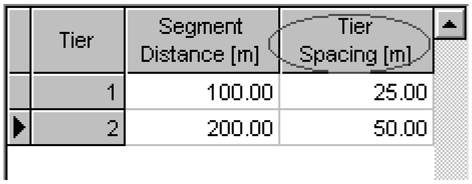
Press this button to delete the selected tier (segment)

- **Segment Distance [m]:** You must specify the distance between tiers. ISC-AERMOD View places some default values for each tier you create. These values, however, should be changed to suit your modeling needs.



Tier	Segment Distance [m]	Tier Spacing [m]
1	100.00	25.00
2	200.00	50.00

- **Tier Spacing [m]:** For each segment (tier), you must specify the spacing between receptors perpendicular to the fenceline. ISC-AERMOD View places some default values for each tier you create. These values, however, should be changed to suit your modeling needs.

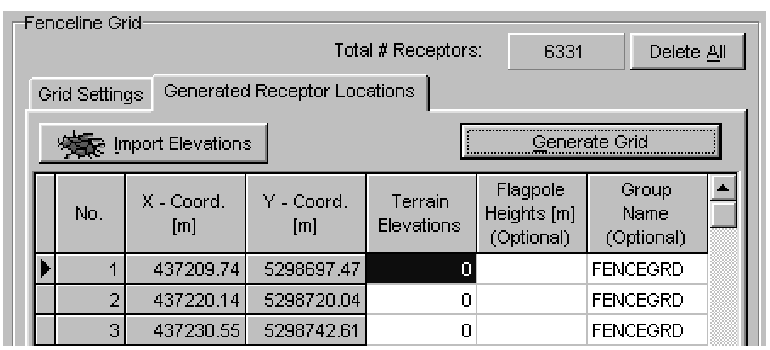


- 

Press this button to generate the Fenceline grid. When ISC-AERMOD finishes generating the grid, the total number of receptors created is automatically displayed. Use the **Delete All** button to delete all fenceline grid receptors.

Generated Receptor Locations Tab

After generating the grid, the **Generated Receptor Locations** tab is automatically displayed. Note that the X and Y coordinates for all fenceline grid receptors are automatically calculated. Some additional parameters need to be specified depending on your modeling options. These additional parameters are described below:



- **Terrain Elevations [m or ft]:** This parameter is only requested if the **Elevated** terrain option was selected. If the **Flat** terrain option was selected, then this column is hidden and any previous inputs will be ignored. The default unit is meters but may be specified in feet if you select Feet for the **Terrain Elevations Unit** option.

- **Flagpole Heights [m] (Optional):** This is the receptor height above ground in meters. This parameter is only requested if the **Flagpole Receptors** option was selected. Otherwise, this column is hidden and any previous inputs will be ignored. Any missing values in the **Flagpole Heights** column, will be interpreted by the U.S. EPA models (ISCST3, AERMOD, and ISC-PRIME) 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 (Optional):** The group name, FENCEGRD, is automatically assigned to every fenceline grid receptor. This group name, however, can be changed at any time.

Note: 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.

- 

Press this button if you want to import terrain elevations for all the fenceline grid receptors. See the *Terrain Elevations* section for more information on types of files supported.

Terrain Elevations

ISC-AERMOD View imports terrain elevations from various digital elevation file formats for two purposes:

1. To extract terrain elevations for receptor locations, and
2. To generate the Terrain Grid (TG) file to be used in the TG Pathway.

The following file formats are supported:

- USGS Digital Elevation Models (DEMs)
- UK DTM
- UK NTF
- XYZ files

See description of each one of these formats on the sections that follow.

USGS Digital Elevation Models (DEMs)

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:** DEMs produced by the USGS that correspond in coverage to standard 1:24,000-scale 7.5- x 7.5-minute quadrangles.

- **1-Degree DEM:** DEMs produced by the Defense Mapping Agency (DMA) that corresponds in coverage to 1° x 1° blocks (one half of standard 1:250,000-scale 1° x 2° quadrangles).

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; DMA 1-degree DEM data are available for most of the United States.

The 7.5-Minute DEM Data

A 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 array have been produced to date.

- The unit of coverage is the 7.5-minute quadrangle. Over-edge 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 contours 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 30-m sampling interval.

The 1-Degree DEM Data

A 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 neat-lines 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, ridgelines, 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.

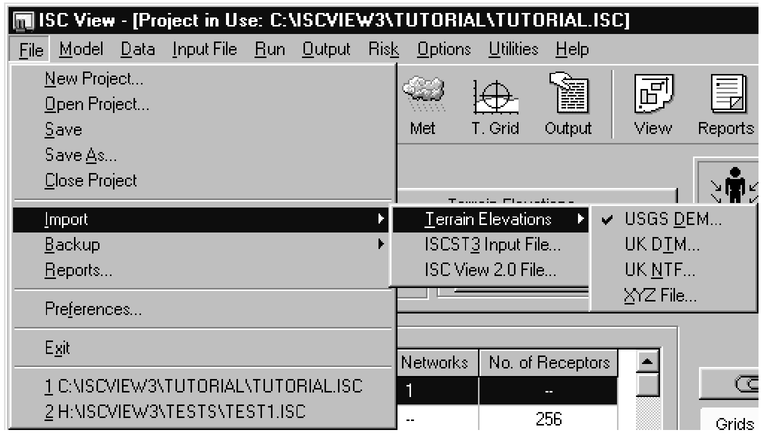
Specifying the Terrain Elevation File to Import

Before importing terrain elevations, you need to specify what type of digital terrain file you have available for your modeling area. ISC-AERMOD View will then pre-process this data for use later on. The following options are available:

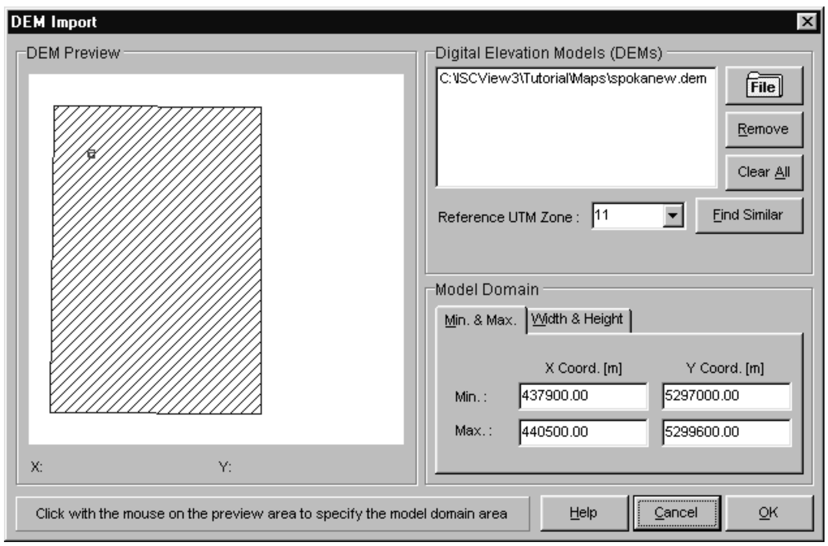
- USGS DEM
- UK DTM
- UK NTF
- XYZ File

➤ How to Specify USGS DEM Files to Import:

Step 1: Select **File | Import | Terrain Elevations** | **USGS DEM** from the menu.



Step 2: The **DEM Import** dialog box is displayed.



DEM Import dialog box

Step 3: Press the **Specify File** button () and select one or more DEM files (*.DEM). The selected DEM files are then displayed on the list. The UTM Zone text box displays the UTM zone for the selected DEM files.

Step 4: You can open more than one DEM file for the same UTM zone. You can do this in two ways, manually, by selecting the file using the **File** button, or automatically, by pressing the **Find Similar** button. The **Find Similar** button will automatically search, in the directory previously specified, all DEM files within the same UTM Zone.

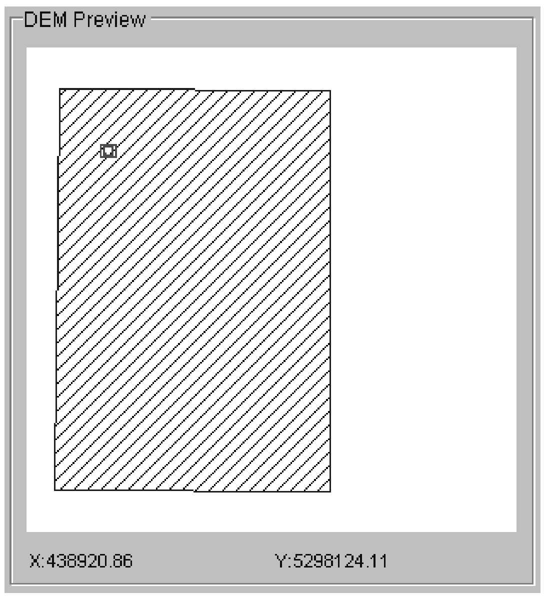
Step 5: All DEM files are displayed on the list and its extents can be previewed on the DEM Preview area.

Step 6: To complete the DEM import, you still need to specify your modeling domain, so ISC-AERMOD View can create a file containing only the terrain elevations for your domain area. You can specify your modeling domain area in three ways:

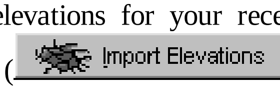
1. Specifying the X and Y coordinates for the SW corner (**Min.**) and for the NE corner (**Max.**) of your modeling domain area.

2. Specifying the X and Y coordinates for the SW corner (**Min.**), the **Width** (X dimension), and the **Height** (Y dimension) of your modeling domain area.

3. Clicking with the mouse on the **DEM Preview** area and dragging to draw a rectangle around your modeling area. Note that the X and Y coordinates for the mouse position are displayed on the bottom of the DEM Preview area.

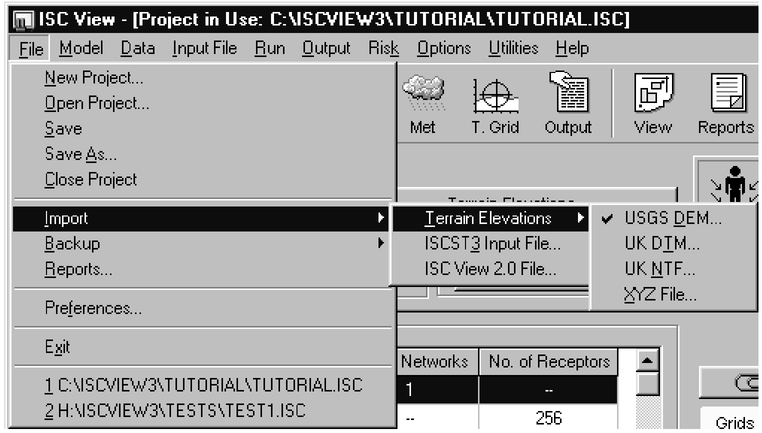


DEM Preview area

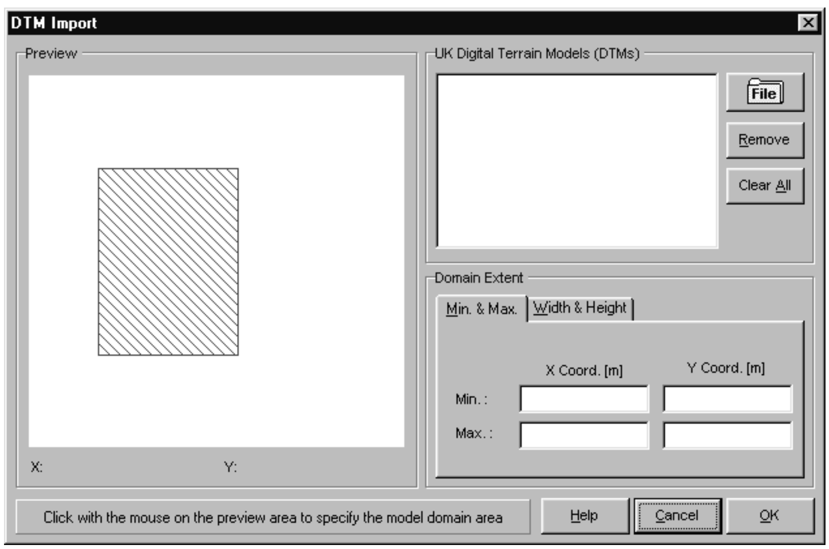
Step 7: Press the **OK** button. ISC-AERMOD View will then pre-process your DEM data and create an intermediate file containing elevations for the selected modeling domain. This intermediate file will be used by ISC-AERMOD View to extract terrain elevations for your receptors when you press any **Import Elevations** button (
) located in the Receptor Pathway windows.

➤ How to Specify UK DTM and UK NTF Files to Import:

Step 1: Select **File | Import | Terrain Elevations** | **UK DTM or UK NTF** from the menu.



Step 2: If you selected the **UK DTM** option, the **DTM Import** dialog box is displayed. If you selected the **UK NTF** option, the **NTF Import** dialog box is displayed.



DTM Import dialog box

Step 3: Press the **Specify File** button () and select one or more files to import. The selected files are then displayed on the list.

Step 4: All files are displayed on the list and their extents can be previewed on the Preview area.

Step 5: To complete the import, you still need to specify your modeling domain, so ISC-AERMOD View can create a file containing only the terrain elevations for your domain area. You can specify your modeling domain area in three ways:

1. Specifying the X and Y coordinates for the SW corner (**Min.**) and for the NE corner (**Max.**) of your modeling domain area.

2. Specifying the X and Y coordinates for the SW corner (**Min.**), the **Width** (X dimension), and the **Height** (Y dimension) of your modeling domain area.

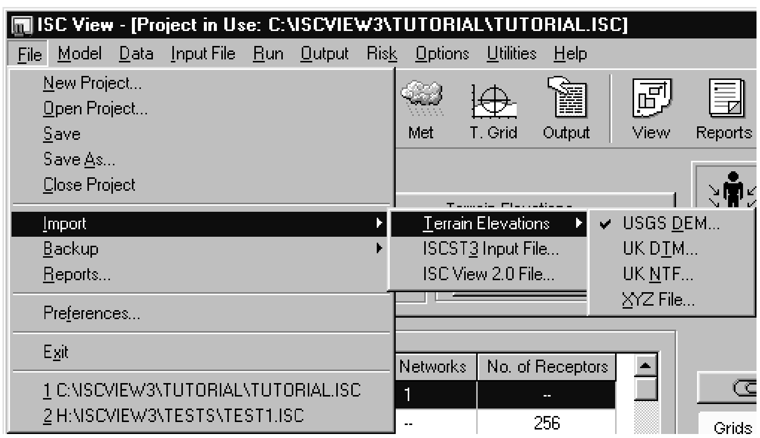
3. Clicking with the mouse on the **Preview** area and dragging to draw a rectangle around your modeling area. Note that the X and Y coordinates for the mouse position are displayed on the bottom of the Preview area.

Step 6: Press the **OK** button. ISC-AERMOD View will then pre-process your data and create an intermediate file containing elevations for the selected modeling domain. This intermediate file will be used by ISC-AERMOD View to extract terrain elevations for your receptors when you press any **Import Elevations** button (

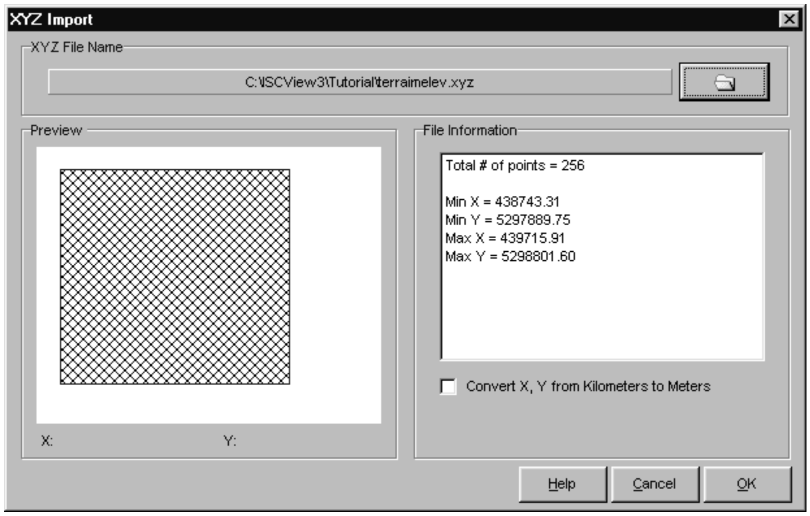
) located in the Receptor Pathway windows.

➤ **How to Specify a XYZ File to Import:**

Step 1: Select **File | Import | Terrain Elevations** | **XYZ File** from the menu.

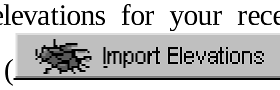


Step 2: The **XYZ Import** dialog box is displayed.



XYZ Import dialog box

Step 3: Press the **Specify File** button () and select one the XYZ file. The total number of points provided in the file and its extents are displayed for your information. If your XYZ file contains the x and y coordinates in kilometers, then you need to check the **Convert X, Y from Kilometers to Meters** box.

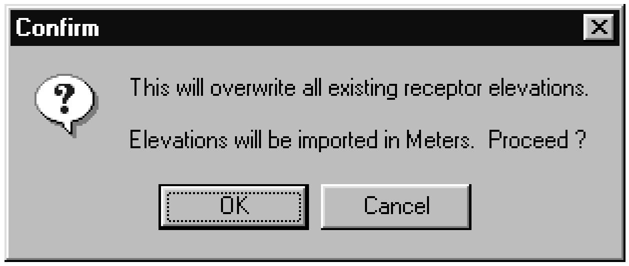
Step 4: Press the **OK** button. ISC-AERMOD View will then pre-process your DEM data and create an intermediate file containing elevations for the selected modeling domain. This intermediate file will be used by ISC-AERMOD View to extract terrain elevations for your receptors when you press any **Import Elevations** button () located in the Receptor Pathway windows.

Importing Terrain Elevations

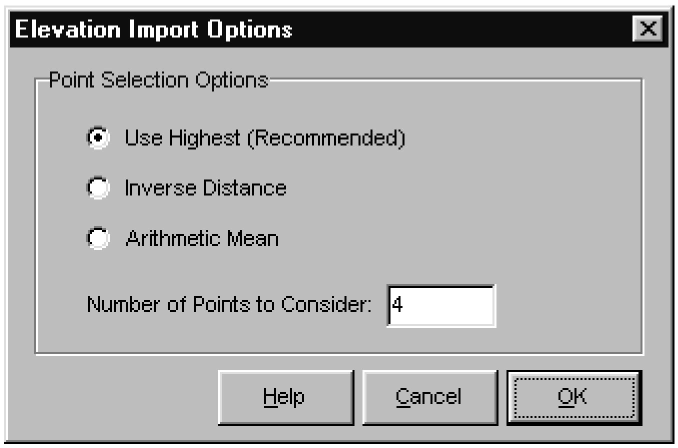
➤ **How to Import Terrain Elevations for All Receptors:**

Step 1: Make sure you followed the steps above on how to specify the digital terrain file to be imported.

Step 2: Go to the **RE-Receptor Summary** window. Press the any **Import Elevations** button (). The following message is displayed:



Step 3: Press the OK button. The **Elevation Import Options** dialog box is then displayed.



Step 4: From the **Elevation Import Options** dialog, select a terrain height interpolation option. It is unlikely that the points you need terrain elevations will exactly match the ones in the Digital Elevation Models. To proceed, you have to choose an interpolation option, as shown below:

- **Use Highest:** Uses the highest of the neighboring points as the required elevation.
- **Inverse Distance:** Interpolates the neighboring points using inverse distance to obtain the elevation at desired point.
- **Arithmetic Mean:** Uses a simple arithmetic mean to obtain elevation at the desired point.
- **Number of Points to Consider:** Number of points surrounding the desired location, to be used in the interpolation scheme.

Step 5: Press the OK button. ISC-AERMOD View will then import terrain elevations for all the receptors already specified in your project.

Note: Please note that the **Import Elevations** button (

) located on each Receptor Pathway window other than the **RE-Receptor Summary** window, will import only the terrain elevations for the receptors specified on that particular window.

UTM Zone Locations and Central Meridians

The following table indicates the degrees of longitude defining the 60 UTM zones around the globe.

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
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-162E
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-180W

The Risk Model

This Risk Mode option is used to guide you on preparing the ISCST3 input files according to the U.S. EPA OSW Human Health Risk Assessment (HHRAP) and Screening Level Ecological Risk Assessment (SLERAP) Protocols.

The Risk Mode was implemented in ISC-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 use 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.

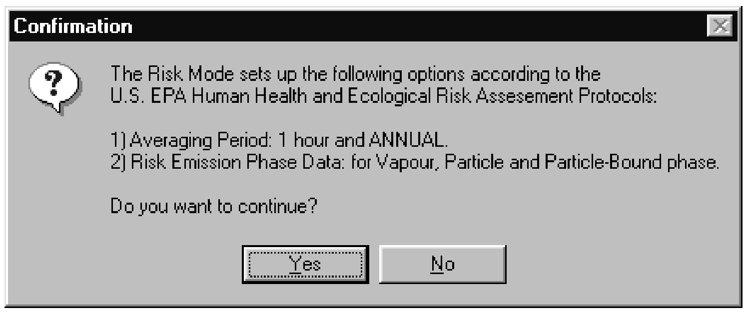
➤ **How to use the Risk Mode:**

Step 1: Select **Risk | Risk Mode (ON)** from the menu.

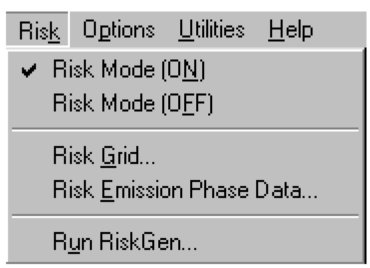
Step 2: A message is displayed asking confirmation to continue. Click **Yes** to continue. The following options will be automatically setup in your ISC-AERMOD View project:

1) Averaging Period: 1 HOUR and ANNUAL

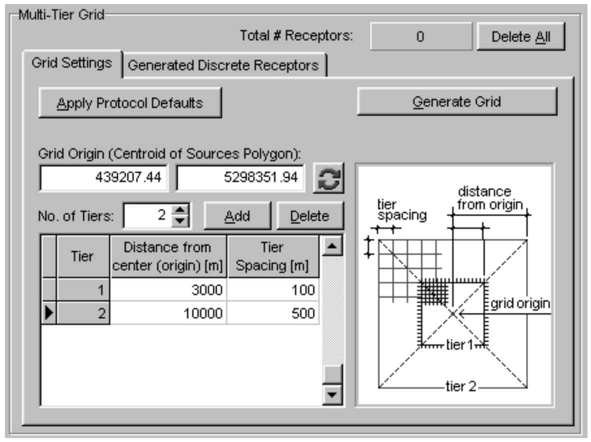
2) The **SO-Gas & Particle Data** window will require data for all three-phases (vapor, particle, and particle-bound) independent on the **Output Types** options you selected in the Control Pathway (Concentration, Deposition, etc…).



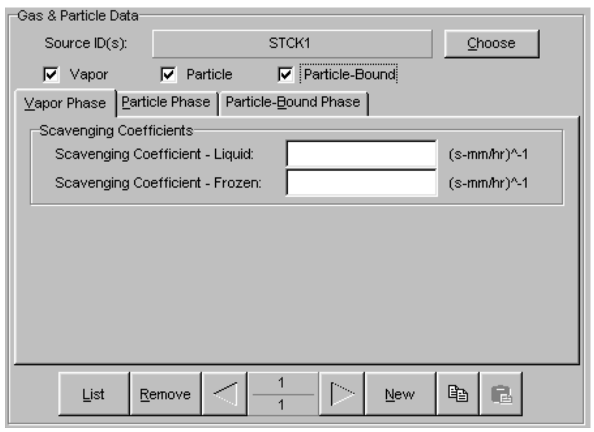
Step 3: Note that the three last options on the Risk menu are now enabled.



Step 4: Select from the **Risk** menu the **Risk Grid** option. This will display the **RE-Multi-Tier Grid** window. From this window, you can automatically setup the risk grid according to the U.S. EPA OSW protocols by clicking on the **Apply Protocol Defaults** button. To generate the grid, click on the **Generate Grid** button.



Step 5: Select from the **Risk** menu the **Risk Emission Phase Data** option. This will display the **SO-Gas & Particle Data** window. From this window, you should specify Vapor Phase, Particle Phase, and Particle-Bound Phase data for all your sources.

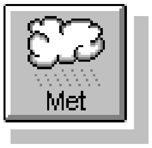


Step 6: Select from the **Risk** menu the **Run RiskGen** option. This will display the **ISC Risk Generator (RiskGen)** utility. This utility allows you to setup all ISCST3 input files you will need for your risk project.

Note: More information on the ISC Risk Generator utility can be found on RiskGen's help file.

CHAPTER 6

Meteorology Pathway



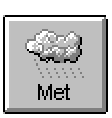
The Meteorology Pathway is where you define the meteorological data to be used for a particular model run. The contents of all the Meteorology Pathway windows will be explained in detail in this chapter.

Contents

ME - Met Input Data Window

The **ME - Met Input Data** window is the first window displayed when you press the **Met** button located on the menu toolbar. From the menu select **Data | Meteorology | Met Input Data**.

In the **ME-Met Input Data** window, you specify the meteorological data file and information on the meteorological stations. ISCST3 and ISC-PRIME use the same format for the pre-processed meteorological data file. AERMOD requires two different types of meteorological data. The requirements for all three models will be described below.

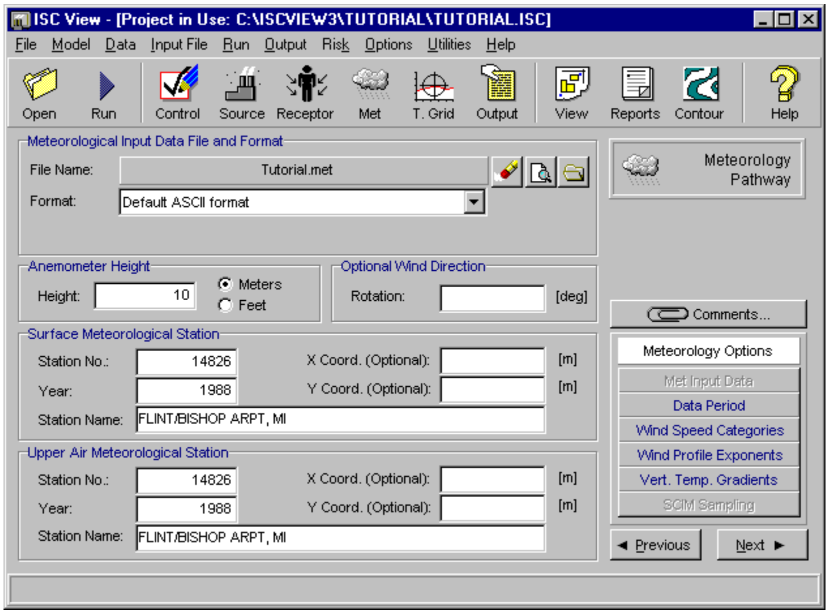


+ 

= Displays the **ME-Met Input Data** window.

Defining Met Input Data for ISCST3 and ISC-PRIME

See below, the parameters that you should define in the **ME-Met Input Data** window for ISCST3 and ISC-PRIME.



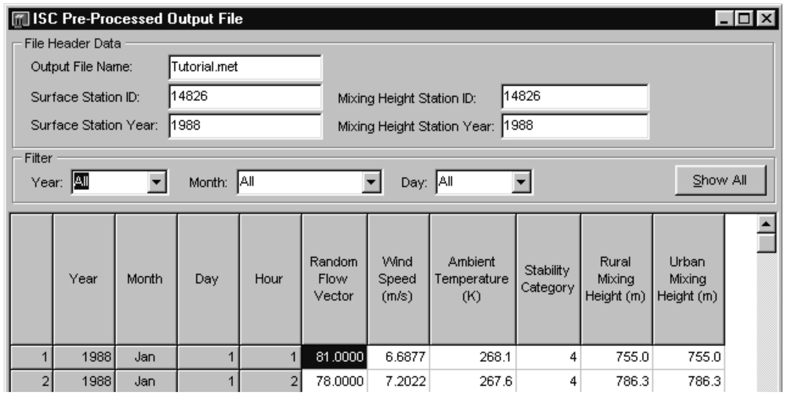
ME-Met Input Data window (ISCST3 and ISC-PRIME)

Meteorology Input Data File and Format

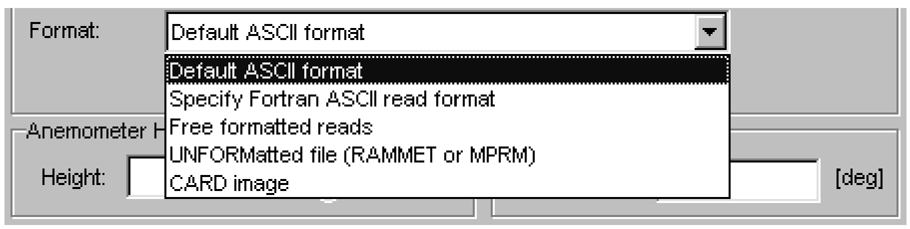
- **File Name:** ISCST3 and ISC-PRIME use hourly meteorological data as one of the basic model inputs. The meteorological data is read into the models from a separate data file. Press the **Specify File** button () to locate the met file. The full path of the specified met data file will be displayed 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.

 + 

Press the **Preview** button and then select **Text** or **Grid** from the pop-up menu to preview the specified met data in text or in a grid format. The **Grid** option displays the selected met data in a grid with a short description of each variable.



- **Format:** Specifies the format of the meteorological data file. Select one of the following format options from the drop-down list box:



1. **Default ASCII format:** Default ASCII format for a sequential hourly file. This is the default option.

2. **Specify Fortran ASCII read format:** If this option is selected than the Fortran read format for an ASCII sequential hourly file should be specified on the **User Format** field.

3. **FREE-formatted reads:** Free-formatted reads for an ASCII sequential hourly file.

4. **UNFORMatted file (RAMMET or MPRM):** Unformatted file generated by the RAMMET or MPRM preprocessors. This option cannot be used for dry or wet deposition applications.

5. **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.

- **User Format:** This option is only available if the option **Specify Fortran ASCII Read format** was selected from the **Format** drop-down list box. 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.

Note: If **Dry Deposition** and/or **Wet Deposition** options were selected on the **CO-Dispersion Options** window, then the **Format** option **4 (Unformatted file**

(RAMMET or MPRM)) cannot be used. This is due to the fact that the deposition algorithms require additional meteorological variables.

U.S. EPA's Utility Programs for Meteorological Data

- **PCRAMMET:** The PCRAMMET program is a meteorological preprocessor used for preparing National Weather Service (NWS) data for use in the ISCST3 model. **Rammet View** is the Lakes Environmental interface for PCRAMMET and is part of the ISC-AERMOD View Package.

- **BINTOASC:** The BINTOASC.EXE program is a utility program that converts unformatted (binary) meteorological data files generated by the RAMMET or MPRM preprocessor programs to the default ASCII format for applications that do not involve dry deposition. The ASCII data file consists of sequential hourly records. The BINTOASC program will convert unformatted data files generated by the Microsoft compiled version of PCRAMMET, and files generated by versions of Rammet, PCRAMMET or MPRM compiled with either the Lahey or the Ryan-McFarland FORTRAN compilers. The BINTOASC utility is available in **Rammet View**.

- **DEPMET:** DEPMET is a utility program that creates an ASCII file with the additional variables required for use of the dry and wet deposition algorithms in ISCST. DEPMET requires: a RAMMET file, the surface meteorological file (CD144 format) used to prepare the RAMMET file, and (for wet deposition only) an hourly precipitation file. Note that the RAMMET file (unformatted file) cannot be used for dry or wet deposition applications.

- **METLIST:** The METLIST.EXE program is a utility program that creates a listing file of meteorological data for a specified day or range of days, which can be sent to a printer. The program lists one day of data per page, with appropriate column headers for the meteorological variables.

Anemometer Height

The height above ground at which the wind speed data was collected must be specified 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.

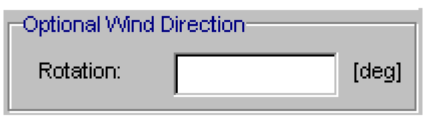
- **Height:** This is the height of the anemometer measurement above ground.

- **Unit (Meters or Feet):** This is the unit used to specify the anemometer height, meters or feet.

Note: For National Weather Service (NWS) data, you should check records, e.g. 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 (Optional)

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.

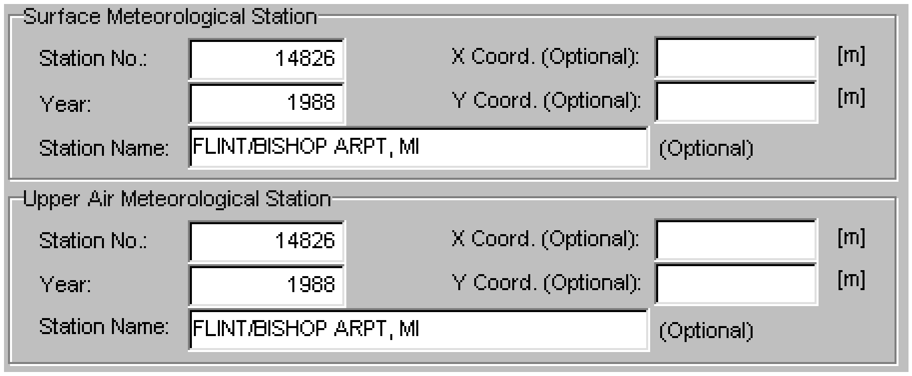


- **Rotation [deg]:** The user may specify the angle in degrees to rotate the input wind direction measurements. The specified value will be subtracted from the wind direction measurements. This option may be used to correct for known (and documented) calibration errors, or to adjust for the alignment 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 and Upper Air Meteorological Stations

The following information is requested for both, the surface meteorological station and the upper air meteorological station:

- **Station No.:** This is the station number, e.g. the 5-digit WBAN (Weather Bureau Army Navy) number for National Weather Service (NWS) stations.
- **Year:** This is the year of 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 (Optional):** This is an optional parameter specifying the name of the station. The model accepts a name of up to 40 characters. If blanks ( ) are used, these will be substituted by underscores (_) in the input file.



- **X Coord. and Y Coord. (Optional):** The x and y coordinates for the location of the stations is optional. At the present time, the U.S. EPA models are not using the station locations.

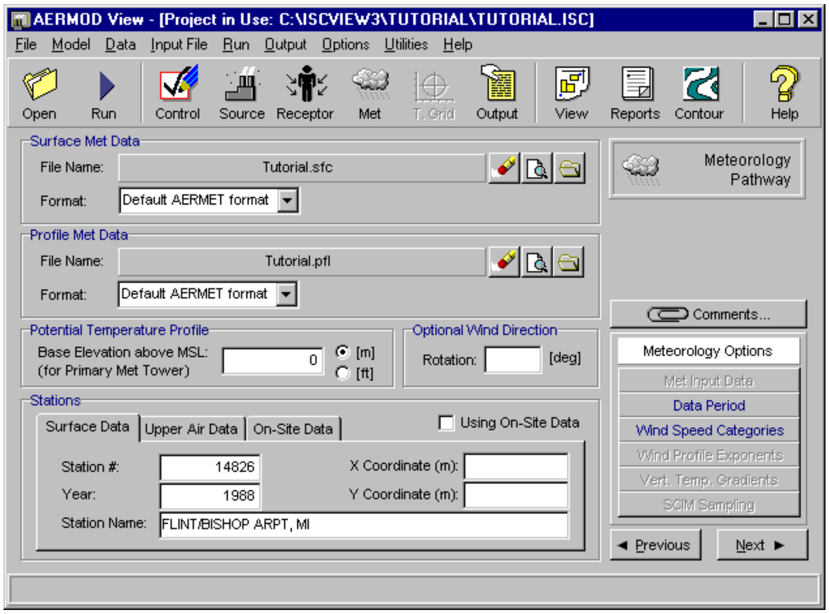
Note: It is important that the **Station No.** and met data **Year** be provided correctly since the EPA model will compare these parameters with the ones provided in the meteorological data files. If a mismatch is found, than non-fatal warning messages will be issued.

Comments (Optional)

This option allows you to add any extra comments you feel are necessary to the Meteorology Pathway portion of the input file.

Defining Met Input Data for AERMOD

See below, the parameters that you should define in the **ME-Met Input Data** window for AERMOD.

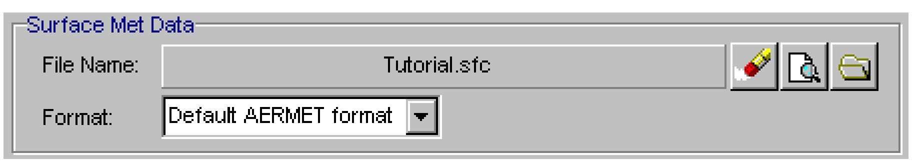


ME-Met Input Data window (AERMOD)

Surface Met Data

- **File Name:** AERMOD uses hourly meteorological data from separate surface and profile data files. The surface met data file is read into the model from a separate data file and contains observed and calculated surface variables, one record per hour.

  Press the **Specify File** button () to locate the surface met file. 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.

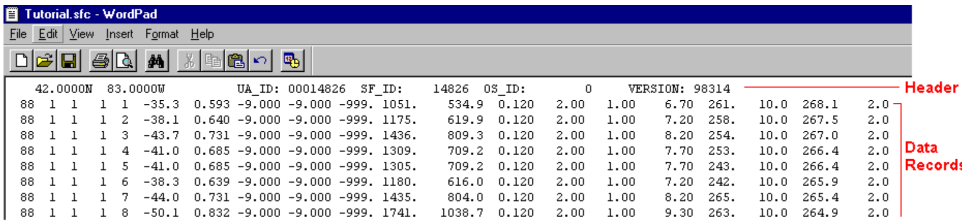
   + 

  Press the **Preview** button and then select **Text** or **Grid** from the pop-up menu to preview the specified met data in text or in a grid format. The **Grid** option displays the selected met data in a grid with a short description of each variable.
- **Format:** Specifies the format of the surface meteorological data file. The default format for the surface data file corresponds with the format of the files generated by

the AERMET meteorological preprocessor program. You also have the option of specifying the Fortran read format for the surface met data file.

Description of the Surface Met Data File Format

The surface met data file consists of a header record and one record for each hour of data. The data is delimited by at least one space between each element (the data may be read as free format). The contents of the surface met data file are as follows:



Header Record:

- Latitude for the application site
- Longitude for the application site
- Upper Air Station Identifier
- Surface Station Identifier
- On-Site Station Identifier
- Version date of AERMET used to generate the file

Data Records:

- Year
- Month (1-12)
- Day (1-31)
- Julian Day (1-366)
- Hour (1-24)
- Sensible Heat Flux (W/m^2) [not used by AERMOD, but read for information purposes only]
- Surface Friction Velocity (m/s)
- Convective Velocity Scale (m/s)
- Vertical potential temperature gradient in the 500 m layer above the planetary boundary layer
- Height of the convectively-generated boundary layer (m)
- Height of the mechanically-generated boundary layer (m)
- Monin-Obukhov length (m)
- Surface roughness length (m)
- Bowen ratio [not used by AERMOD, but read for information purposes only]
- Albedo [not used by AERMOD, but read for information purposes only]
- Wind speed (m/s) used in the computations
- Wind direction (degrees) corresponding to the wind speed above
- Height at which the wind above was measured (m)
- Temperature (K) used in the computations

- Height at which the temperature above was measured (m)

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. This user's guide can be found on your installation CD.

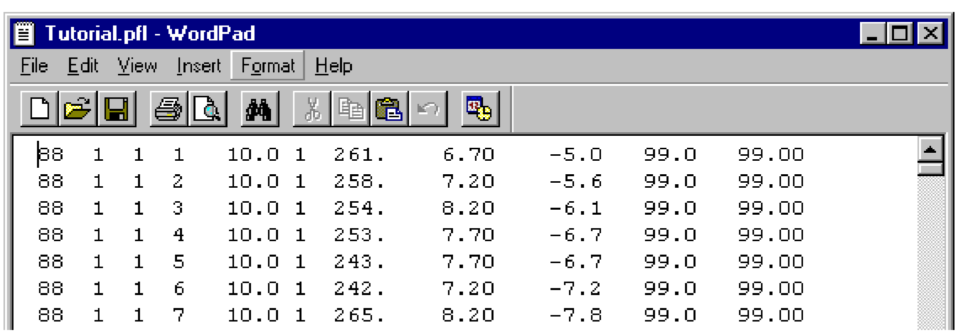
Profile Met Data

- **File Name:** The profile met data file is read into the model from a separate data file and contains the observations made at each level of an on-site tower, or the one level observations taken from NWS data. Press the **Specify File** button () to locate the profile met file. 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:** Specifies the format of the profile meteorological data file. The default format for the profile data file corresponds with the format of the files generated by the AERMET meteorological preprocessor program. You also have the option of specifying the Fortran read format for the profile met data file.

Description of the Profile Met Data File Format

The profile met data file consists of one or more records for each hour of data. The data is delimited by at least one space between each element and may be read as Fortran free format. The contents of the profile meteorological data file are as follows:



88	1	1	1	10.0	1	261.	6.70	-5.0	99.0	99.00
88	1	1	2	10.0	1	258.	7.20	-5.6	99.0	99.00
88	1	1	3	10.0	1	254.	8.20	-6.1	99.0	99.00
88	1	1	4	10.0	1	253.	7.70	-6.7	99.0	99.00
88	1	1	5	10.0	1	243.	7.70	-6.7	99.0	99.00
88	1	1	6	10.0	1	242.	7.20	-7.2	99.0	99.00
88	1	1	7	10.0	1	265.	8.20	-7.8	99.0	99.00

- Year
- Month (1 - 12)
- Day (1 -31)
- Hour (1 - 24)

- Measurement height (m)
- Top flag = 1, if this is the last (highest) level for this hour, or 0, otherwise
- Wind direction for the current level (degrees)
- Wind speed for the current level (m/s)
- Temperature at the current level (K)
- Standard deviation of the wind direction (degrees)
- Standard deviation of the vertical wind speed (m/s)

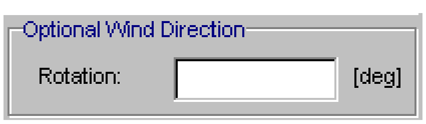
Potential Temperature Profile

The AERMOD model generates a gridded vertical profile of potential temperatures for use in the plume rise calculations. Potential temperature is dependent on the elevation above mean sea level (MSL), therefore you must specify the base elevation for the profile.

- **Base Elevation above MSL [m or ft]:** You must specify the base elevation above mean sea level for the potential temperature profile. This elevation should be the base elevation of the primary meteorological tower. You can specify the base elevation in meters or feet.

Optional Wind Direction (Optional)

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 [deg]:** The user may specify 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, and On-Site Meteorological Stations

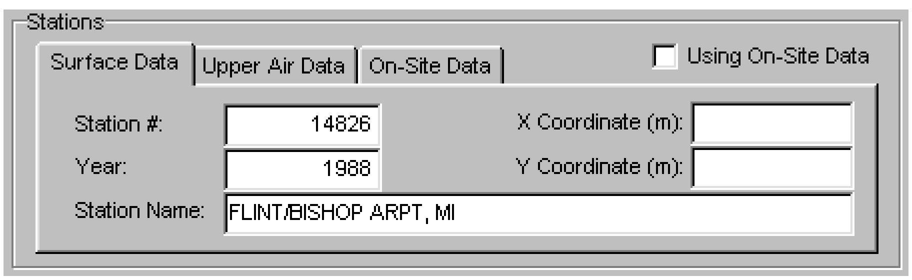
The following information is requested for the surface meteorological station, the upper air meteorological station, and the on-site meteorological station:

- **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 below.

- **Station No.:** This is the station number, e.g. the 5-digit WBAN (Weather Bureau Army Navy) number for National Weather Service (NWS) stations.

- **Year:** This is the year of 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 (Optional):** This is an optional parameter specifying the name of the station. The model accepts a name of up to 40 characters. If blanks ( ) are used, these will be substituted by underscores (_) in the input file.



- **X Coord. and Y Coord. (Optional):** The x and y coordinates for the location of the stations are optional. At the present time, the U.S. EPA model is not using the station locations.

Note: 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, than non-fatal warning messages will be issued.

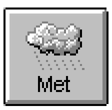
Comments (Optional)

This option allows you to add any extra comments you feel are necessary to the Meteorology Pathway portion of the input file.

ME - Data Period Window

In the **ME-Data Period** window, you specify which days within the meteorological data file are to be read by the model. The default is to read the entire meteorological data file and to process all days within that period. However, you have the choice of specifying a data period or specifying particular days or ranges of days to process.

You have access to the **ME-Data Period** window by pressing the **Met** button located on the menu toolbar and then the **Data Period** button located on the lower right side of any Meteorology (ME) Pathway window. From the menu select **Data | Meteorology | Data Period**.

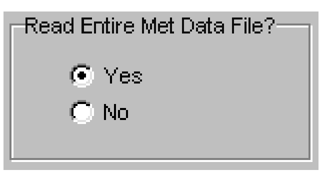


+


= Displays the **ME-Data Period** window.

Read Entire Met Data File?

By default, the model (ISCST3, AERMOD, or ISC-PRIME) will read the entire meteorological data file and will process all days within that period. However, the model also allows you to specify particular days or ranges of days to process from the sequential meteorological file input. See below how to select any one of these options.



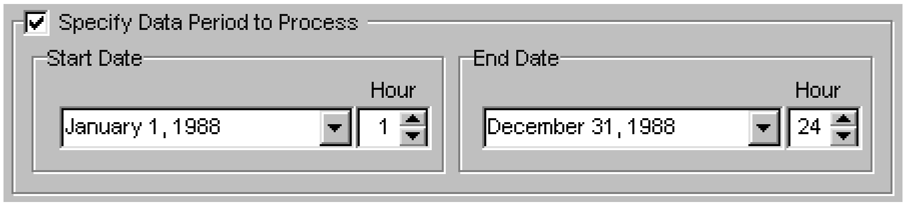
- **Yes:** This is the default option. If this option is selected, then the model will read the entire meteorological data file.

- **No:** This option tells the model not to read the entire meteorological data file but to read the specified days or ranges of days. If you select this option, then you can specify the days to be read by the model using one of the following options:

  1. Specify Data Period to Process
  2. Specify Particular Days and/or Ranges of Days to Process

Specifying Data Period to Process

This option controls which period within the meteorological data file the model reads. The following are the inputs necessary if this option is selected:



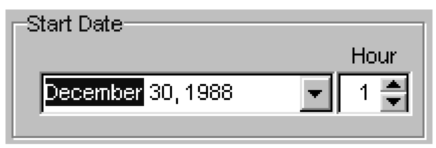
Start Date / End Date

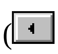
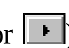
- **Start Date:** You must specify the year, month, and day of the first record to be read from the meteorological data file.

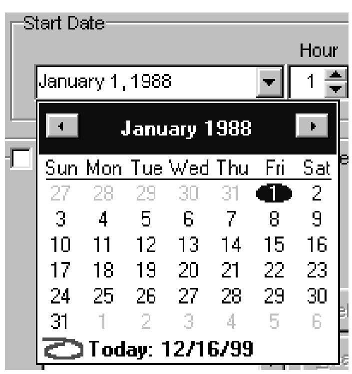
- **End Date:** You must specify the year, month, and day of the last record to be read from the meteorological data file.

➤ **How to Specify the Start and End Dates:**

Step 1: Select the text you want to change and type in the new text.

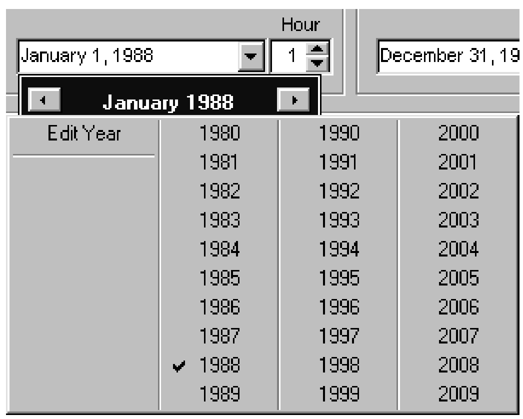


Step 2: Click on the down arrow to display the calendar. Use the arrows ( or ) to go to the previous or to the next month.



Step 3: Click on the month label (January) to display a pop up menu from where you can select the month.

Step 4: Click on the year label (1988) to display a pop up menu from where you can select the year. Click on the Edit Year label (Edit Year) to specify years other than those listed on the pop up menu.

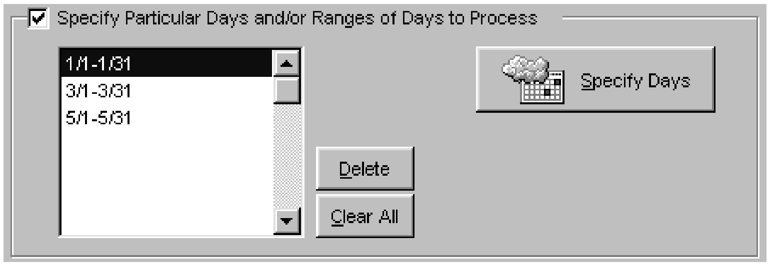


- **Hour (Optional):** These are optional parameters that may be used to specify the start and end hours for the data period to be read. If either one is to be specified, then both must be specified. If the start and end hours are not specified, then processing begins with hour 1 of the start date, and ends with hour 24 of the end date.

Note: Any records in the data file that occur before the **Start Date** are ignored. Records in the data file that occur after the **End Date** are also ignored.

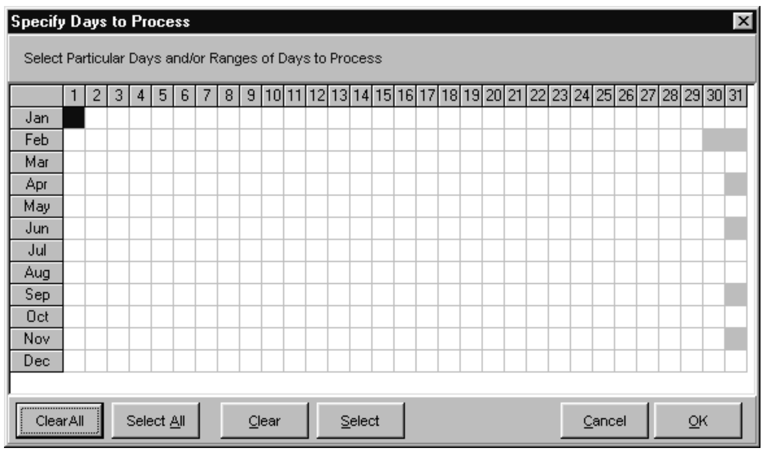
Specifying Particular Days and/or Ranges of Days to Process

This option controls which days or range of days should be read by the model.

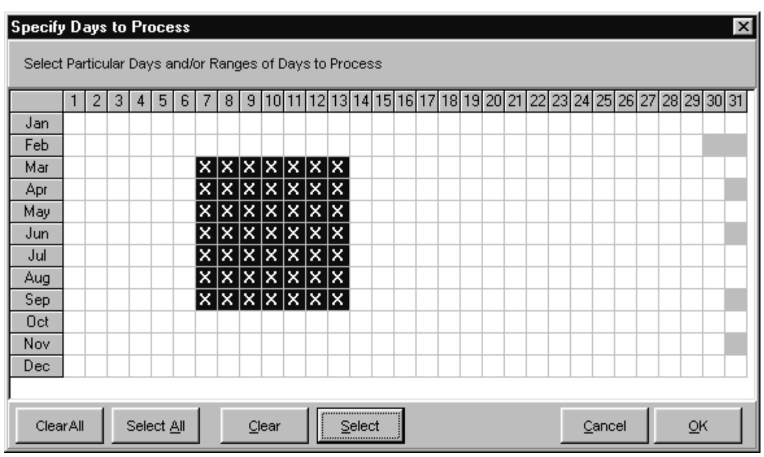


➤ **How to Specify Particular Days and Ranges of Days to Process:**

Step 1: Press the **Specify Days** button to display the **Specify Days to Process** dialog box.

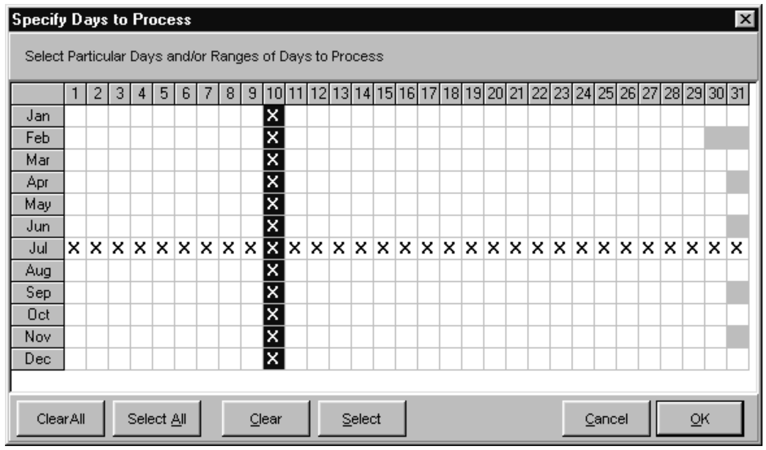


Step 2: To select a range of days, click with the left mouse button on the cell for the first day in the range and drag to select all the days in the range. Press the **Select** button to mark these cells.

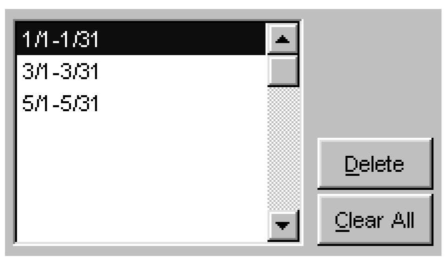


Step 3: To select a full month, click on the row header. To select the same day for all months, click on the column header. After each selection, do not forget to

press the **Select** button to mark your selection. Use the **Clear** button to clear marked cells that are currently selected. Use the **Clear All** button to clear all marked cells. Use the **Select All** button to mark all days of the year.



Step 4: After all selections are done, press the **OK** button to close the dialog. Note that the days and/or range of days that you selected are displayed in the list. Use the **Delete** button to delete an entry from the list. Use the **Clear All** button to clear the entire list.



Note: 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** averages on the **CO-Pollutant/Avg Time/...** window, and then specify the six-month period.

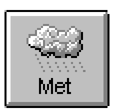
ME - Wind Speed Categories Window

Some of the parameters that may be input to the models are allowed to vary by wind speed category. Examples of such inputs are:

- Wind Speed Profile Exponents,
- Vertical Potential Temperature Gradients, and
- Variable Emission Rate Factors.

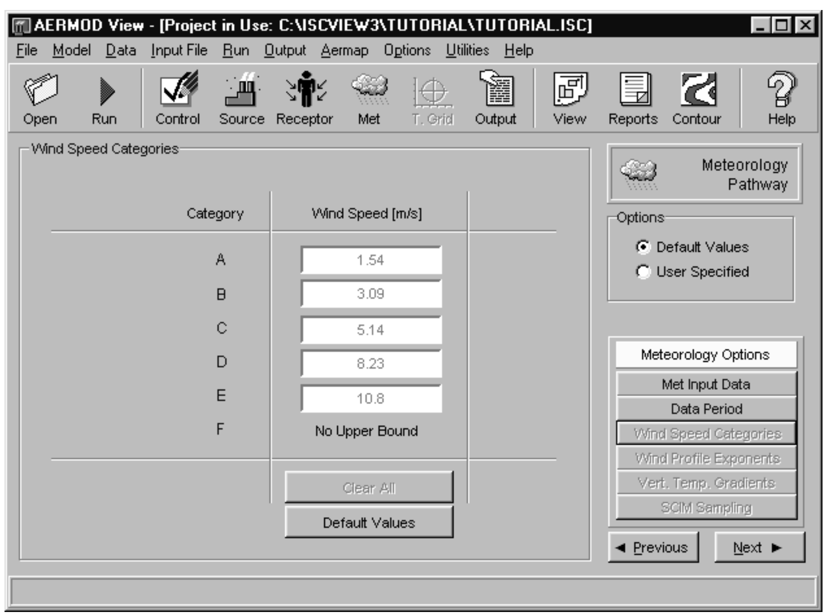
The models use six wind speed categories and these are defined by the upper bound wind speed, which must be defined for the first five categories. The sixth category is assumed to have no upper bound. The models use default values for the wind speed categories, but also allow you to define different category boundaries. In ISC-AERMOD View, you define which wind speed category values are to be used by the model on the **ME-Wind Speed Categories** window.

You have access to the **ME-Wind Speed Categories** window by pressing the **Met** button located on the menu toolbar and then the **Wind Speed Categories** button located on the lower right side of any Meteorology (ME) Pathway window. From the menu select **Data | Meteorology | Wind Speed Categories**.



+ 

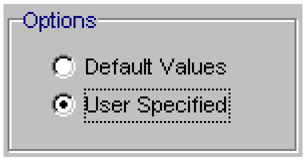
= Displays the **ME-Wind Speed Categories** window.



ME - Wind Speed Categories window

Defining User-Specified Wind Speed Categories

Two options are available on the **ME-Wind Speed Categories** window: **Default Values** or **User Specified**. Each one of these options is described below:



- **Default Values:** If this option is selected, then ISC-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

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.

- **User Specified:** This option allows you to specify wind speed values other then the default values. Use the **Default Values** button to place the default wind speed values in the fields, allowing you to change one or all of these values. Use the **Clear All** button if you want to clear all the values in the text boxes.

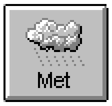
ME - Wind Profile Exponents Window

(ISCST3 and ISC-PRIME Only)

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 models use default wind profile exponents if the **Regulatory Default** option is selected in the Control Pathway.

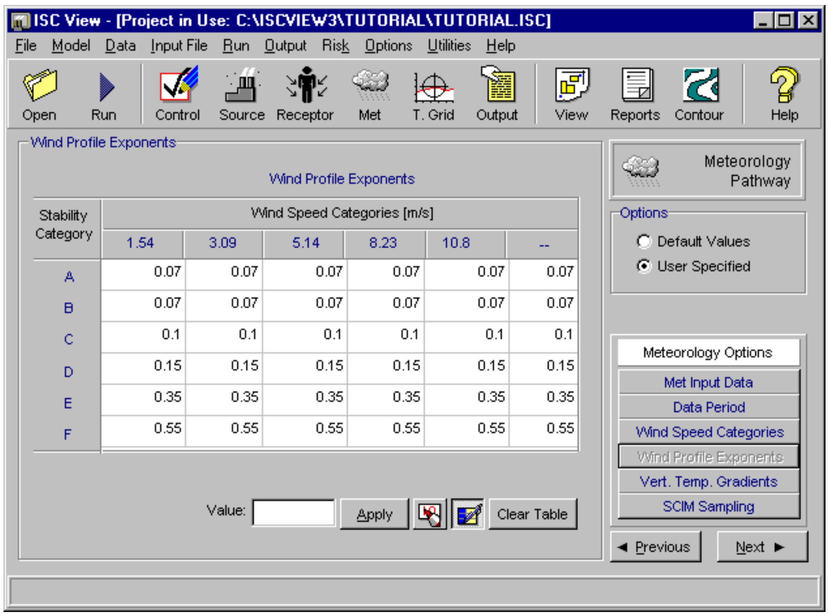
If the **Non-Default Options** is selected, then you have the choice of specifying wind profile exponent values other than the default. In ISC-AERMOD View, you define which wind profile exponent values are to be used by the model on the **ME-Wind Profile Exponents** window.

You have access to the **ME-Wind Profile Exponents** window by pressing the **Met** button located on the menu toolbar and then the **Wind Profile Exponents** button located on the lower right side of any Meteorology (ME) Pathway window. From the menu select **Data | Meteorology | Wind Profile Exponents**.



+

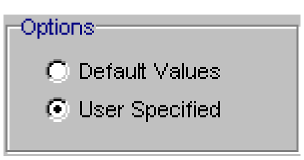

= Displays the **ME- Wind Profile Exponents** window.



ME - Wind Profile Exponents window

Defining Wind Profile Exponents

You can only have access to the **ME - Wind Profile Exponents** window if the **Non-Default Options** was selected in the Control Pathway. Two options are available on the **ME-Wind Profile Exponents** window, **Default Values** or **User Specified**. Each one of these options is described below:



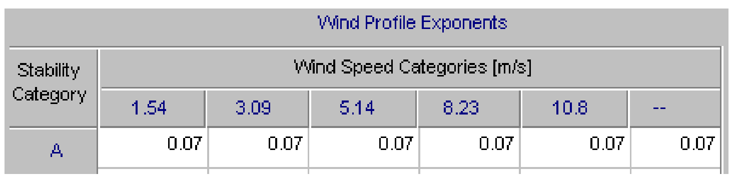
- **Default Values:** If this option is selected, then ISC-AERMOD View automatically inputs the default values for wind profile exponents in the table and does not allow you to change these values. The default values for wind profile exponents are listed in Table 6-1 below. These values are different for **Rural** and **Urban** dispersion coefficients.

Table 6-1: 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

- **User Specified:** This option allows you to specify wind profile exponent values other than the default values. Use the **Default Values** button to place the default wind profile exponent values in the fields, allowing you to change one or all of these values. Use the **Clear All** button if you want to clear all values from the table.

Note: The heading of each column will display the values for the six (6) wind speed categories. These values are either the default values used by the model (1.54, 3.09, 5.14, 8.23, and 10.8 m/s) or the user-specified values.

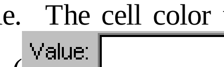


Note: If the **Regulatory Default** option was selected in the Control Pathway, then any input on this window will be ignored.

Inputting Data on the Wind Profile Exponents Table

➤ **How to Input Data on the Table:**

Step 1: With the **Edit** button () pressed down, type the values directly in each cell, or

Step 2: Click the **Mark** button () and pressing down the Shift key, select one or more rows from the table. The cell color will be changed to blue. Type the value on the **Value** field (

) and press the **Enter** key or click the **Apply** button (

). The value will be placed on the marked cells. The **Clear All** button is used to clear all values from the table.

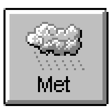
ME - Vertical Temperature Gradients Window

(ISCST3 and ISC-PRIME Only)

This 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.

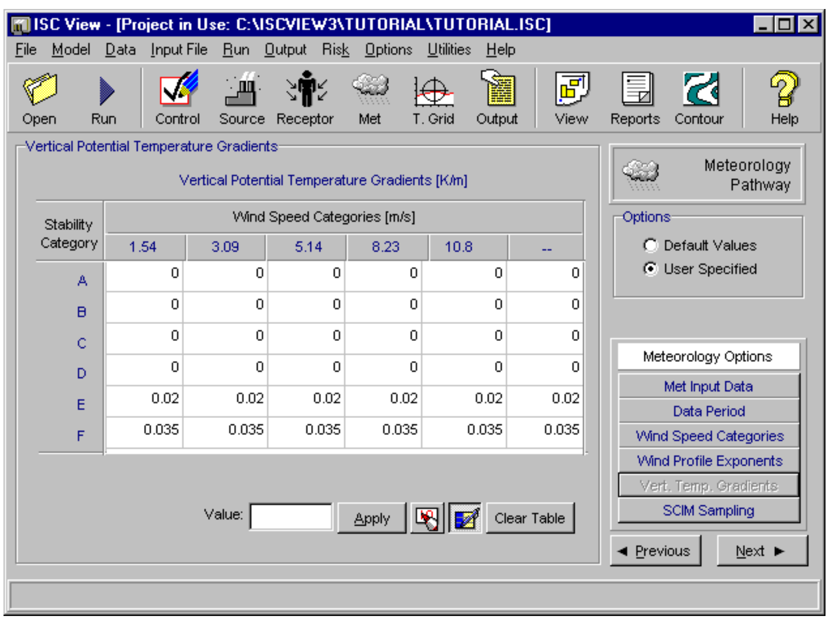
The models use default vertical potential temperature gradients if the **Regulatory Default** option was selected in the Control Pathway. If the **Non-Default Options** was selected instead, then you have the choice of specifying wind profile exponent values other than the default. In ISC-AERMOD View, you define which wind profile exponent values are to be used by the model on the **ME-Vertical Temperature Gradients** window.

You have access to the **ME-Vertical Temperature Gradients** window by pressing the **Met** button located on the menu toolbar and then the **Vert. Temp. Gradients** button located on the lower right side of any Meteorology (ME) Pathway window. From the menu select **<u>D</u>ata | <u>M</u>eteorology | Vert. <u>T</u>emp. Gradients**.



+

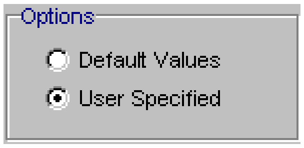

= Displays the **ME-Vertical Temperature Gradients** window.



ME - Vertical Temperature Gradients window

Defining Vertical Potential Temperature Gradients

You can only have access to the **ME–Vertical Temperature Gradients** window if the **Non-Default Options** was selected in the Control Pathway. Two options are available on the **ME–Vertical Temperature Gradients** window, **Default Values** or **User Specified**. Each one of these options is described below:



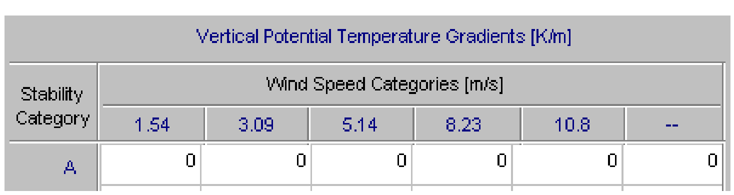
- **Default Values:** If this option is selected, then ISC-AERMOD View automatically inputs the default values for vertical potential temperature gradients in the table and does not allow you to change these values. The default values for vertical potential temperature gradients are listed in Table 6-2 below. These values are different for **Rural** and **Urban** dispersion coefficients.

Table 6-2: Default Values for Vertical Potential Temperature Gradients

Pasquill Stability Category	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	0.020	0.020
F	0.035	0.035

- **User Specified:** This option allows you to specify vertical potential temperature gradients other then the default values. Use the **Default Values** button to place the default values for vertical potential temperature gradients in the fields, allowing you to change one or all of these values. Use the **Clear All** button if you want to clear all values from the table.

Note: The heading of each column will display the values for the six (6) wind speed categories. These values are either the default values used by the model (1.54, 3.09, 5.14, 8.23, and 10.8 m/s) or the user-specified values.



Note: If the **Regulatory Default** option was selected in the Control Pathway, then any input on this window will be ignored.

Inputting Data on the Vertical Temperature Gradients Table

➤ **How to Input Data on the Table:**

Step 1: With the **Edit** button ( ) pressed down, type the values directly in each cell, or

Step 2: Click the **Mark** button ( ) and pressing down the Shift key, select one or more rows from the table. The cell color will be changed to blue. Type the value on the **Value** field ( ) and press the **Enter** key or click the **Apply** button ( ). The value will be placed on the marked cells. The **Clear All** button is used to clear all values from the table.

ME – SCIM Sampling Window

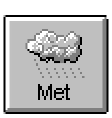
(ISCST3 Only)

The **ME-SCIM Sampling** window is only applicable to the ISCST3 model and is available if all of the following options were selected in the Control Pathway:

- **Non-Default Options**
- **Air Toxics Options**
- **Sampled Chronological Input Model (SCIM)**

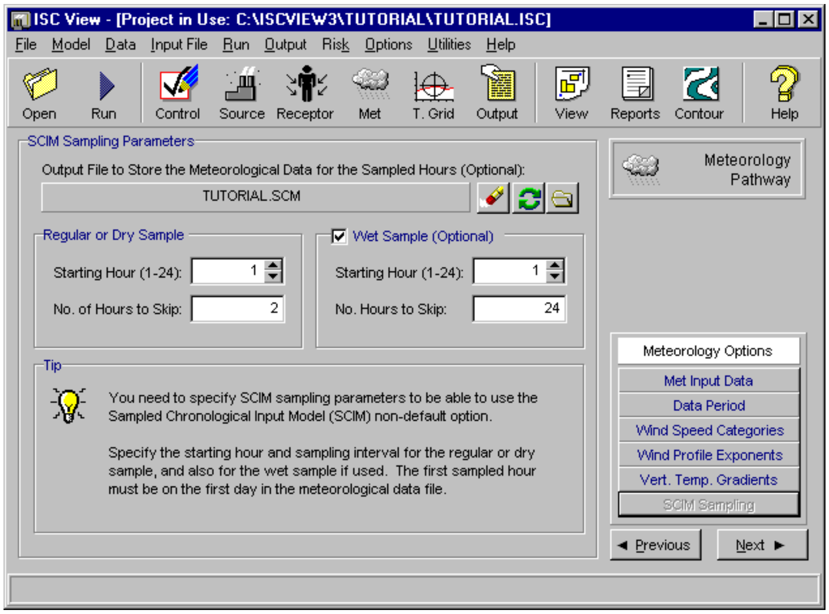
The non-default **SCIM** option is used to reduce model runtime and is primarily applicable to multi-year model simulations. The SCIM option can only be used with the ANNUAL averaging period. The SCIM option samples the meteorological data at a user-specified regular interval to approximate the long-term (i.e., ANNUAL) average impacts. The SCIM option may also be used to reduce model runtime.

You have access to the **ME-SCIM Sampling** window by pressing the **Met** button located on the menu toolbar and then the **SCIM Sampling** button located on the lower right side of any Meteorology (ME) Pathway window. From the menu select **Data | Meteorology | SCIM Sampling**.



+


= Displays the **ME-SCIM Sampling** window.



ME–SCIM Sampling window

Defining SCIM Sampling Parameters

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.

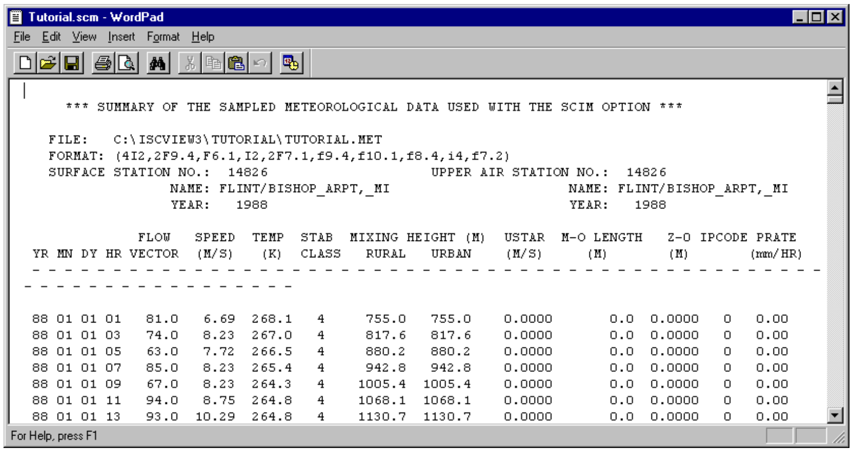
See below, the parameters that you should define if the SCIM option was selected:

- **Output File to Store the Meteorological Data for the Sampled Hours (Optional):** This option allows you to specify the file name and location of the output file to be created when running the model. This output file will contain the meteorological data for the sampled hours. To specify the file name use the following buttons:

   Press this button to get the ISC-AERMOD View default file name for this option.

   Press this button to clear the file name panel, in case you do not want the output file to be created.

   Press this button to specify a file name and location other than the default.




```
     *** SUMMARY OF THE SAMPLED METEOROLOGICAL DATA USED WITH THE SCIM OPTION ***

   FILE:   C:\ISCVIEW3\TUTORIAL\TUTORIAL.MET
   FORMAT: (4I2,2F9.4,F6.1,I2,2F7.1,f9.4,f10.1,f8.4,i4,f7.2)
   SURFACE STATION NO.:  14826                        UPPER AIR STATION NO.:  14826
                  NAME: FLINT/BISHOP_ARPT,_MI                          NAME: FLINT/BISHOP_ARPT,_MI
                  YEAR:   1988                                         YEAR:   1988

              FLOW   SPEED  TEMP  STAB  MIXING HEIGHT (M)  USTAR  M-O LENGTH   Z-O IPCODE PRATE
 YR MN DY HR VECTOR  (M/S)   (K)  CLASS   RURAL   URBAN    (M/S)     (M)       (M)        (mm/HR)
 - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
- - - - - - - - - - - - - - - - -

 88 01 01 01   81.0   6.69  268.1   4     755.0   755.0    0.0000       0.0  0.0000   0    0.00
 88 01 01 03   74.0   8.23  267.0   4     817.6   817.6    0.0000       0.0  0.0000   0    0.00
 88 01 01 05   63.0   7.72  266.5   4     880.2   880.2    0.0000       0.0  0.0000   0    0.00
 88 01 01 07   85.0   8.23  265.4   4     942.8   942.8    0.0000       0.0  0.0000   0    0.00
 88 01 01 09   67.0   8.23  264.3   4    1005.4  1005.4    0.0000       0.0  0.0000   0    0.00
 88 01 01 11   94.0   8.75  264.8   4    1068.1  1068.1    0.0000       0.0  0.0000   0    0.00
 88 01 01 13   93.0  10.29  264.8   4    1130.7  1130.7    0.0000       0.0  0.0000   0    0.00
```

Sample of the SCIM Output File

- **Regular or Dry Sample:** You must specify the starting hour and the sampling interval for the regular or dry sample. The starting hour can range from 1 to 24. The first sampled hour must be on the first day in the meteorological data file. The sampling interval (No. of Hours to Skip) must be greater than 1.

- **Wet Sample (Optional):** Check this option if you want to specify the starting hour and the sampling interval for the wet sample (if used). The only restriction for the wet sample option is that the Start Hour cannot be greater than the No. of Days to Skip. Since wet deposition does not occur at regular intervals, the user can also specify the wet sampling interval to reduce uncertainty introduced by sampling for wet deposition.

The ISCST3 model uses the following approach depending on the type of sampling parameters specified:

Only Regular Sampling Selected

All hourly impacts (concentration, dry deposition flux, and wet deposition flux) are calculated in the normal fashion for each sampled hour. See below:

- **Annual Average Concentration:** Calculated by dividing the cumulative concentration for the sampled hours by the number of hours sampled (arithmetic average).

$$\mathbf{C = C_s / N_s}$$

Where:

C = Calculated Concentration

C_s = Cumulative impacts for the sampled hours
N_s = Number of sampled hours

- **Annual Dry and Wet Deposition Fluxes:** Calculated by scaling the respective cumulative fluxes for the sampled hours by the ratio of the total hours to the sampled hours.

$$\mathbf{D = D_s \ (N_t/N_s)}$$
$$\mathbf{W = W_s \ (N_t/N_s)}$$

Where:

D = Calculated Dry Deposition Fluxes
W = Calculated Wet Deposition Fluxes
D_s = Cumulative Dry Deposition impacts for the sampled hours
W_s = Cumulative Wet Deposition impacts for the sampled hours
N_s = Number of sampled hours
N_t = Total number of hours in the data period

Regular Sampling + Wet Sampling Selected

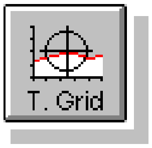
When the wet hour sampling is also selected along with the regular (or dry) sampling, then the following approach is followed:

- **Annual Average Concentration and Dry Deposition Fluxes:** These are based on the weighted contributions from regular samples, modeled as dry hours and wet hour samples. The regular samples consist of all the hours based on regular sampling interval, but the effects of precipitation are ignored so that their contribution represents only dry conditions, while the contribution from the wet hour samples represents only wet conditions.

- **Annual Wet Deposition Fluxes:** These are only based on the wet hour samples.

CHAPTER 7

Terrain Grid Pathway



The Terrain Grid Pathway (**TG**) is an optional pathway and is only applicable to the ISCST3 and ISC-PRIME models. The AERMOD model does make use of this option. In this pathway you can define the input terrain grid data to be used in calculating dry depletion in elevated or complex terrain. In ISC-AERMOD View, the Terrain Grid pathway options are available in the **TG-Terrain Grid** window. The contents of this window will be explained in detail in the section that follows.

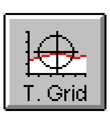
Contents

TG - Terrain Grid Window

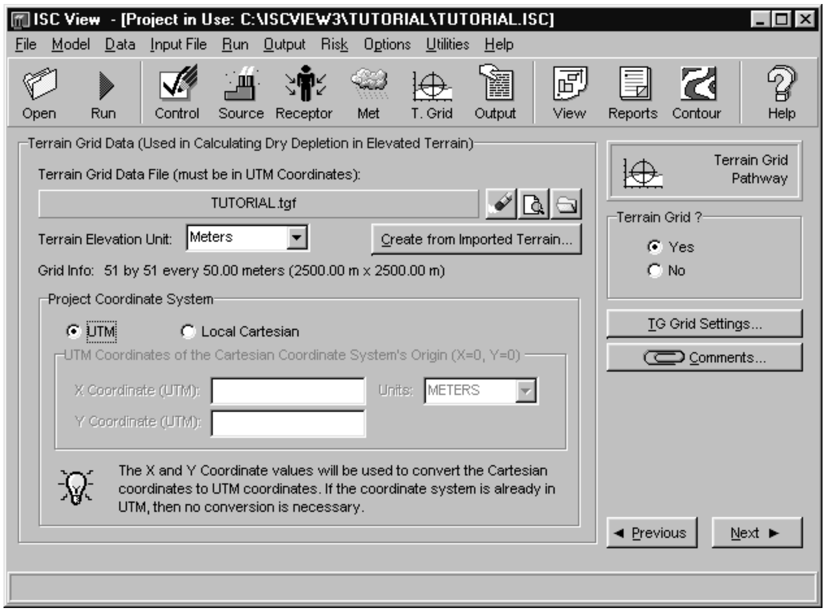
The **Terrain Grid** option defines the input terrain grid data used in calculating dry depletion in elevated or complex terrain. If the **Flat** option was selected in the Control Pathway, then the **Terrain Grid** option is not available for use.

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.

The **TG-Terrain Grid** window is the first window displayed when you press the **T. Grid** button located on the menu toolbar. From the menu select **Data | Terrain Grid**.



= Displays the **TG-Terrain Grid** window.

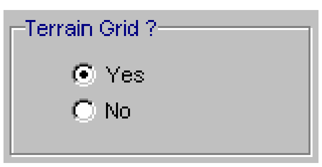


TG-Terrain Grid window

Defining Terrain Grid Data

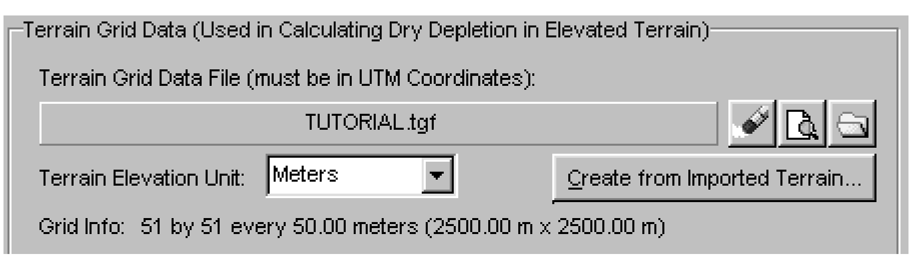
The following parameters should be defined for the terrain grid option:

- **Terrain Grid ?** Two options are available. See description below.



1. **Yes**: This option allows you to specify a terrain grid data input file.
2. **No**: This option tells the model not to use the terrain grid data input file for the current run.

- **Terrain Grid Data File (must be in UTM Coordinates)**: If you are using the **Terrain Grid** option, then you must specify the name and location of the file containing the terrain grid data. Note that the terrain grid data file must be in UTM coordinates.



To specify the file name use the following buttons:

 Press this button to specify the file name and location of your terrain grid file.

 Press this button to preview the specified terrain grid file.

 Press this button to clear the file name panel.

- **Terrain Elevation Unit:** The default units for terrain elevations in the terrain grid file are meters above MSL (Mean Sea Level). If the elevations in your terrain grid file are defined in feet, than you must select **Feet** from the drop-down list.

  Note: If you are using the **Create from Imported Terrain** option, then the elevation unit will always be **Meters**.

- **Create from Imported Terrain:** If you have one or more DEM (Digital Elevation Model) files for the area you are modeling, then you can use them to automatically generate a terrain grid file. When you press the **Create from Imported Terrain** button, ISC-AERMOD View will read the database table containing the pre-processed terrain elevation and will resample the data according to the spacing defined in the TG Grid Settings dialog box. By default, ISC-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 by pressing the **TG Grid Settings** button located in the TG-Terrain Grid window.

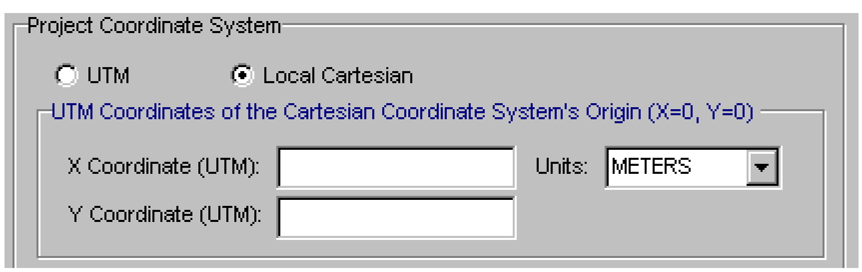
- **TG Grid Settings:** In the **TG Grid Settings** dialog box, you can change the default TG grid spacing of 50 meters. After specifying the new spacing, you should press the **Calculate No. of Points** button. By pressing this button, ISC-AERMOD View calculates the number of data points in the X and Y directions that will be created when the TG File is generated. It is important that you verify the calculated No. 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 TG 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 TG File.

WARNING: The creation of the TG 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 TG File.

- **Project Coordinate System:** If the location of your sources and receptors was not given in UTM coordinates then you need to specify the relationship between the coordinate system you used to define source and receptor locations and the UTM coordinate system.

  1. **UTM:** Select UTM if you defined source and receptor locations in UTM

  2. **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 for the origin of your project coordinate system.



- **UTM Coordinates of the Cartesian Coordinate Systems's Origin:** If your project's coordinate system was not defined in UTM then you need to define the UTM coordinates for the origin of your project coordinate system 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:** This is the unit used for the value input in the **X Coordinate** and **Y Coordinate** text boxes. The units may be specified in **METERS**, **FEET**, and **KM**. The default unit is **METERS**.

Format of the Terrain Grid Data File

The terrain grid file contains 1 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

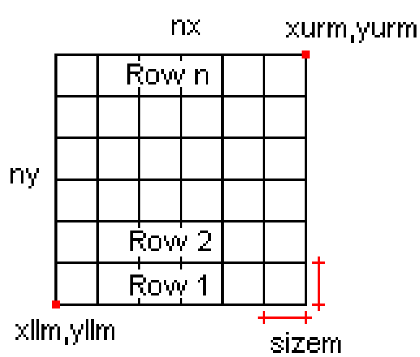
Where:

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 parameters MXTX and MXTY in the U.S. EPA models. The original ISC-PRIME model, for example, allows a maximum of 201 points in the x and y directions. AERMOD allows a maximum of 601 points in both directions. 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 TG File.

See in **Figure 7-2** below, a terrain grid data file that contains terrain elevations for a 101x101 grid.

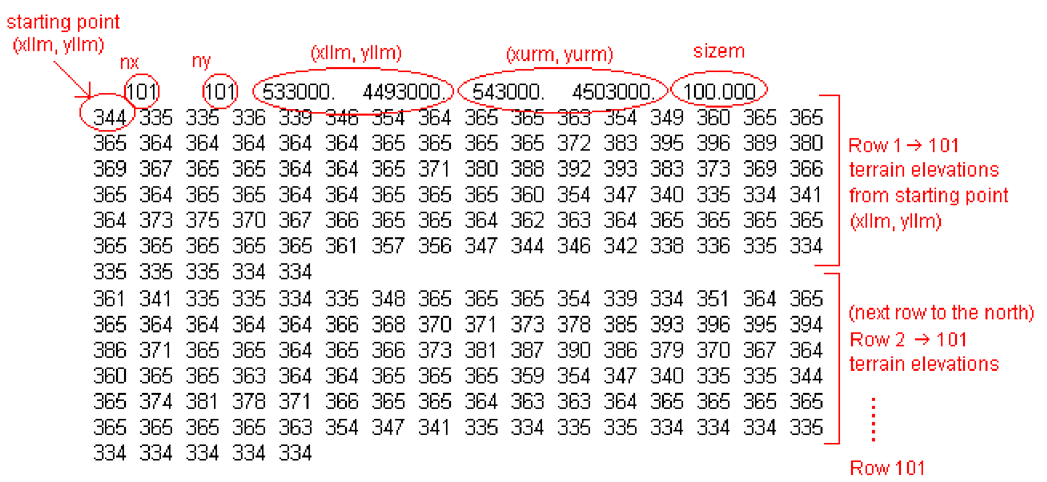

```
101  101  533000.  4493000.  543000.  4503000.  100.000
344 335 335 336 339 346 354 364 365 365 363 354 349 360 365 365
365 364 364 364 364 364 365 365 365 365 372 383 395 396 389 380
369 367 365 365 364 364 365 371 380 388 392 393 383 373 369 366
365 364 365 365 364 364 365 365 365 360 354 347 340 335 334 341
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
```

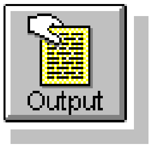

Figure 7-2. Terrain Grid Data File Format

Comments (Optional)

This option allows you to add any extra comments you feel are necessary to the Terrain Pathway portion of the input file.

CHAPTER 8

Output Pathway



The Output Pathway (**OU**) defines the output options for the model run. Several output options are available. You may select any combination of output options for a particular run.

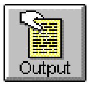
In ISC-AERMOD View, the Output Pathway inputs and options are available in several windows. The contents of these windows will be explained in detail in the sections that follows:

Contents

OU - Tabular Outputs Window

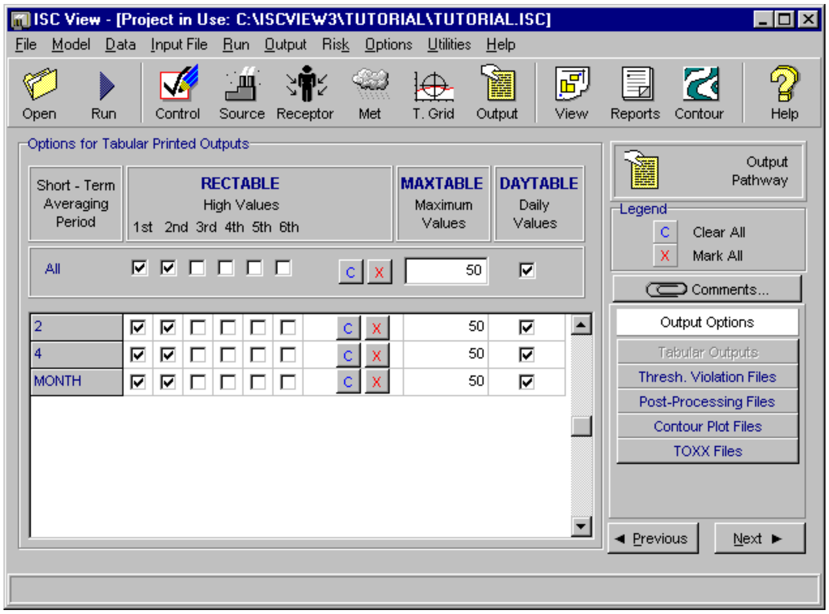
Three options of tabular output are available for the main output file. In ISC-AERMOD View, these options are available to you in the **OU-Tabular Outputs** window.

You have access to the **OU-Tabular Outputs** window by pressing the **Output** button located on the menu toolbar. If you are in another Output Pathway window, then you just need to press the **Tabular Outputs** button located on the lower right side of any Output Pathway window. From the menu select **Data | Output | Tabular Outputs**.



or 

= Displays the **OU-Tabular Outputs** window



OU-Tabular Outputs window

Defining Tabular Printed Outputs

Three tabular printed output options are available on the **OU-Tabular Outputs** window. They control the output option for the following:

- **RECTABLE - High Values**............Controls output options for high value summary tables by receptor.
- **MAXTABLE - Maximum Values**....Controls output options for overall maximum value summary tables.
- **DAYTABLE - Daily Values**...........Controls output options for tables of concurrent values summarized by receptor for each day processed.

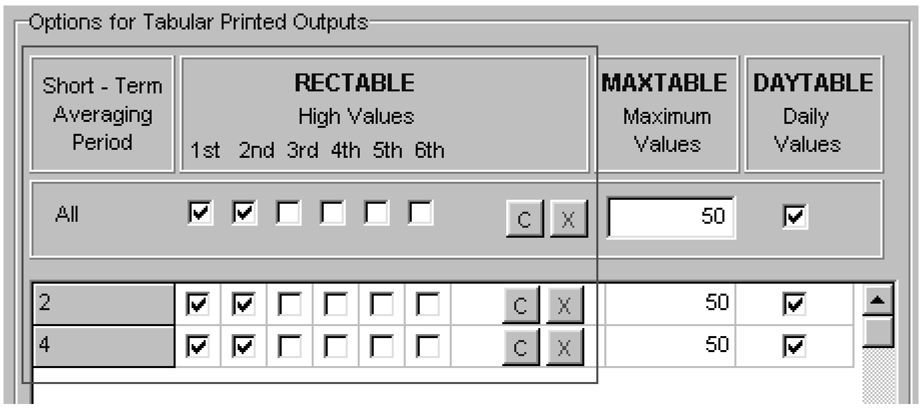
You can define tabular output options for each short-term averaging period (e.g., 1, 2, 3, 4, 6, 8, 12, 24, or Month) selected in the Control Pathway. The tabular output options do not apply to the **Period** or **Annual** averaging time.

The option **ALL** indicates that all short-term averaging periods being modeled are being used.

Note: If no short-term averaging period was selected in Control Pathway (e.g., 1, 2, 3, 4, 6, 8, 12, 24, or Month), then the **RECTABLE**, the **MAXTABLE**, and the **DAYTABLE** options are not available for use. ISC-AERMOD View displays a message advising you that no short-term averaging periods were selected for the current run.

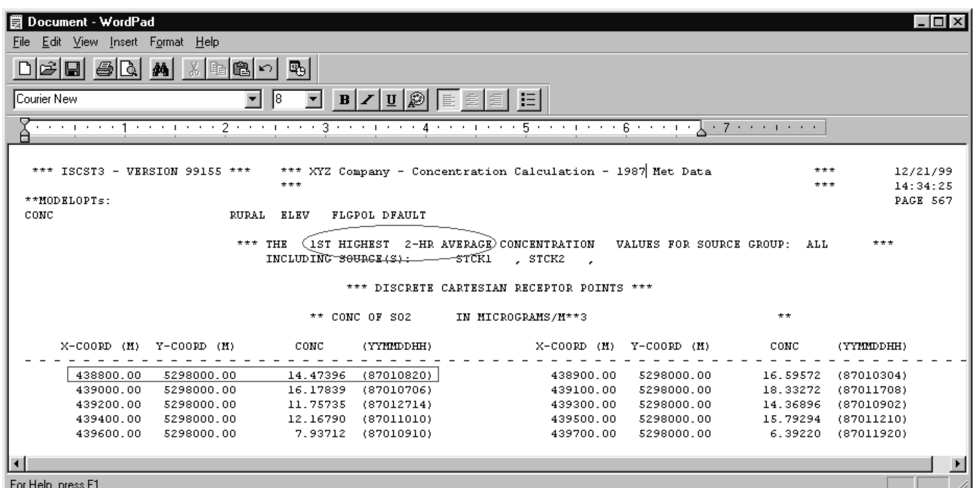
Defining the RECTABLE - High Values Option

The RECTABLE 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 (e.g., 1st, 2nd, etc.) that are applicable to your modeling.



RECTABLE Option

The table is printed in the main output file in the following format:



RECTABLE option - Example of the table produced for the 1st High Value for 2-Hr Averaging Period

The above example shows that the first highest concentration value at the specified receptor location (438800.00, 5298000.00) is 14.47396 MICROGRAMS/M**3 and happens on January 08, 1987 at hour 20.

X-COORD (M)	Y-COORD (M)	CONC	(YYMMDDHH)
438800.00	5298000.00	14.47396	(87010820)

To facilitate the selection of the **RECTABLE-High Values** check boxes, two buttons are provided for each row. The function of these two buttons is explained below:

 Clear all selected **High Value** check boxes for the current row.
 Checks all the **High Value** check boxes for the current row.

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.

Defining the MAXTABLE - Maximum Values Option

The **MAXTABLE** 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.

```
                              *** THE MAXIMUM    50    2-HR AVERAGE CONCENTRATION    VALUES FOR SOURCE GROUP:  ALL       ***
                                  INCLUDING SOURCE(S):     STCK1    , STCK2    ,

                                        ** CONC OF SO2       IN MICROGRAMS/M**3                          **

RANK        CONC    (YYMMDDHH) AT      RECEPTOR (XR,YR) OF TYPE    RANK        CONC    (YYMMDDHH) AT      RECEPTOR (XR,YR) OF TYPE
- - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
  1.      18.33272 (87011708) AT ( 439100.00, 5298000.00)  DC       26.      10.98081 (87012420) AT ( 439100.00, 5298000.00)  DC
  2.      17.12526 (87010624) AT ( 439100.00, 5298000.00)  DC       27.      10.86178 (87011610) AT ( 439000.00, 5298000.00)  DC
  3.      17.00970 (87010702) AT ( 439100.00, 5298000.00)  DC       28.      10.76936 (87011516) AT ( 439000.00, 5298000.00)  DC
  4.      16.59572 (87010304) AT ( 438900.00, 5298000.00)  DC       29.      10.58262 (87011718) AT ( 439100.00, 5298000.00)  DC
  5.      16.46148 (87011704) AT ( 439100.00, 5298000.00)  DC       30.      10.57138 (87011812) AT ( 439100.00, 5298000.00)  DC
  6.      16.17839 (87010706) AT ( 439000.00, 5298000.00)  DC       31.      10.50822 (87011504) AT ( 439100.00, 5298000.00)  DC
  7.      15.79294 (87011210) AT ( 439500.00, 5298000.00)  DC       32.      10.32627 (87011702) AT ( 439100.00, 5298000.00)  DC
  8.      15.40624 (87011622) AT ( 439100.00, 5298000.00)  DC       33.      10.32483 (87011618) AT ( 439100.00, 5298000.00)  DC
  9.      14.88025 (87012504) AT ( 439000.00, 5298000.00)  DC       34.      10.31335 (87011802) AT ( 439100.00, 5298000.00)  DC
 10.      14.47396 (87010820) AT ( 438800.00, 5298000.00)  DC       35.      10.03330 (87010306) AT ( 439100.00, 5298000.00)  DC
 11.      14.36896 (87010902) AT ( 439300.00, 5298000.00)  DC       36.      10.01913 (87011706) AT ( 439100.00, 5298000.00)  DC
 12.      13.64539 (87010824) AT ( 439100.00, 5298000.00)  DC       37.       9.98723 (87012422) AT ( 439100.00, 5298000.00)  DC
 13.      13.26755 (87010904) AT ( 439100.00, 5298000.00)  DC       38.       9.89989 (87010312) AT ( 439200.00, 5298000.00)  DC
 14.      13.01780 (87011806) AT ( 439000.00, 5298000.00)  DC       39.       9.80514 (87010304) AT ( 439000.00, 5298000.00)  DC
 15.      12.56437c(87012602) AT ( 439100.00, 5298000.00)  DC       40.       9.74083 (87010704) AT ( 439100.00, 5298000.00)  DC
 16.      12.34977 (87010716) AT ( 439000.00, 5298000.00)  DC       41.       9.72069 (87010314) AT ( 439500.00, 5298000.00)  DC
 17.      12.27920 (87011420) AT ( 439100.00, 5298000.00)  DC       42.       9.59559 (87010710) AT ( 439000.00, 5298000.00)  DC
 18.      12.16790 (87011010) AT ( 439400.00, 5298000.00)  DC       43.       9.43984 (87010714) AT ( 439000.00, 5298000.00)  DC
 19.      12.13942 (87010302) AT ( 439100.00, 5298000.00)  DC       44.       9.24631 (87010902) AT ( 439200.00, 5298000.00)  DC
 20.      12.12319 (87012004) AT ( 439000.00, 5298000.00)  DC       45.       9.13485 (87011624) AT ( 438800.00, 5298000.00)  DC
 21.      11.75735 (87012714) AT ( 439200.00, 5298000.00)  DC       46.       8.63143 (87010724) AT ( 438800.00, 5298000.00)  DC
 22.      11.38026 (87010718) AT ( 439000.00, 5298000.00)  DC       47.       8.51366 (87011908) AT ( 439200.00, 5298000.00)  DC
 23.      11.19733 (87010306) AT ( 439000.00, 5298000.00)  DC       48.       8.47381 (87010710) AT ( 439100.00, 5298000.00)  DC
 24.      11.17732 (87011808) AT ( 439000.00, 5298000.00)  DC       49.       8.45837 (87011616) AT ( 439100.00, 5298000.00)  DC
 25.      11.00670 (87010308) AT ( 439200.00, 5298000.00)  DC       50.       8.43046 (87013014) AT ( 439400.00, 5298000.00)  DC
 *** RECEPTOR TYPES:  GC = GRIDCART
                      GP = GRIDPOLR
                      DC = DISCCART
                      DP = DISCPOLR
                      BD = BOUNDARY
```

MAXTABLE option - Example of the table produced for the 2-Hr Averaging Period and Source Group ALL

If, for example, you specify 50 for the MAXTABLE option then the model will produce a table for each short-term averaging period and each source group containing the 50 highest values.

The MAXTABLE option will be produced in the main output file in the following format:

```
RANK        CONC     (YYMMDDHH) AT      RECEPTOR (XR,YR) OF TYPE
- - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
   1.      18.33272 (87011708) AT ( 439100.00, 5298000.00)  DC
```

- Rank Number
- Concentration and/or Deposition Value
- Date and Time it occurred
- Receptor Location (X and Y Coordinates)
- Receptor Type (GC = GRIDCART, GP = GRIDPOLR, DC = DISCCART, DP = DISCPOLR, BD = BOUNDARY)

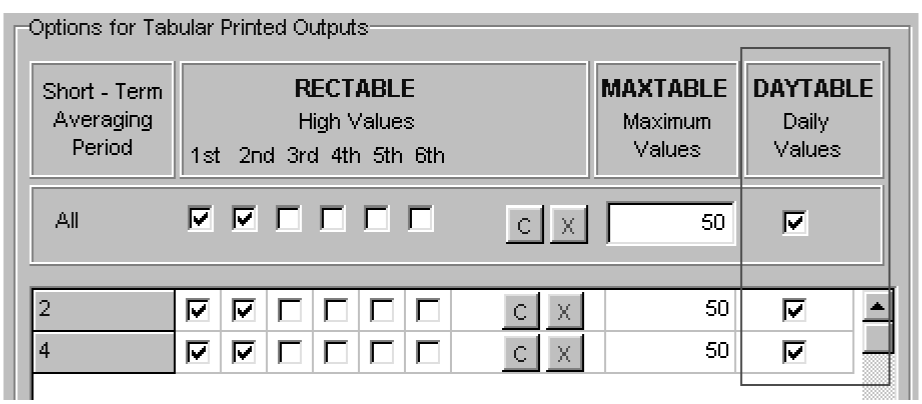
As with the **RECTABLE** option, for cases where the same **MAXTABLE** options will be applied for all short-term averaging periods being modeled, the input can be done only once for the **ALL** averaging period.

Note: The number of overall maximum values that the model can store for each averaging period and source group is controlled by the NMAX parameter in the U.S. EPA Fortran code which is initially set at 50.

Defining DAYTABLE - Daily Values Option

For each averaging period for which the DAYTABLE 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 for each source group are output.

For cases were you want to define daily tables for all the short term averaging periods being modeled, the input may be simplified by only checking the **DAYTABLE - Daily Values** check box for the **ALL** averaging period.



For example, if 2, and 4-hour averages are calculated and the DAYTABLE 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).

CAUTION! The **DAYTABLE** 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.

```
*** CONCURRENT 2-HR AVERAGE CONCENTRATION   VALUES ENDING WITH HOUR 2 FOR DAY   1 OF 1987 ***
    FOR SOURCE GROUP: ALL
    INCLUDING SOURCE(S):      STCK1    , STCK2    ,

                       *** DISCRETE CARTESIAN RECEPTOR POINTS ***

                  ** CONC OF SO2      IN MICROGRAMS/M**3                    **
```

X-COORD (M)	Y-COORD (M)	CONC	X-COORD (M)	Y-COORD (M)	CONC
438800.00	5298000.00	0.00000	438900.00	5298000.00	0.00000
439000.00	5298000.00	0.00000	439100.00	5298000.00	0.00000
439200.00	5298000.00	0.00000	439300.00	5298000.00	0.00000
439400.00	5298000.00	0.00000	439500.00	5298000.00	4.89861
439600.00	5298000.00	1.17541	439700.00	5298000.00	2.73966

DAYTABLE option - Example of the table produced for the 2-Hr Averaging Period ending with Hour 2 for Day 1 and Source Group ALL

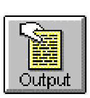
Comments (Optional)

This option allows you to add any extra comments you feel are necessary to the Output Pathway portion of the input file.

OU - Threshold Violation Files Window

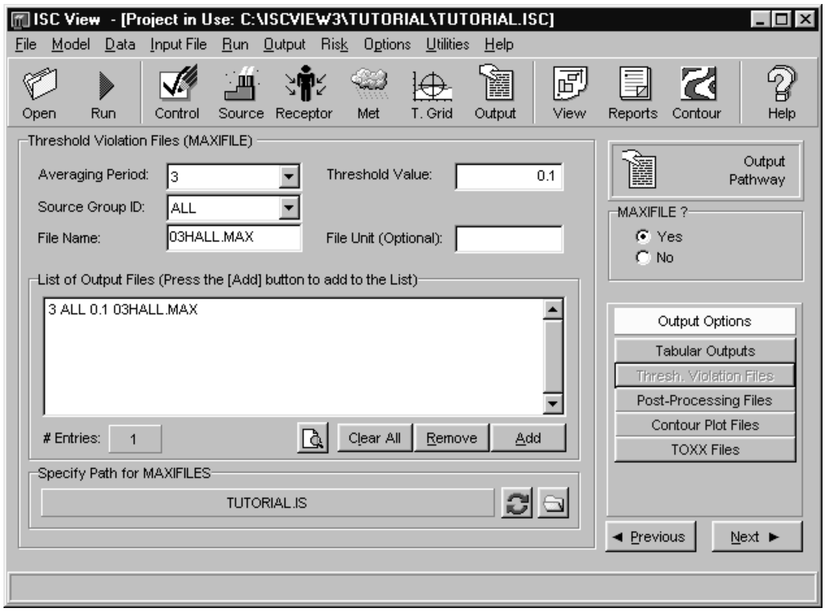
The **Threshold Violation Files (MAXIFILE)** option produces files of all occurrences of violations equal to or above the user-specified threshold values.

You have access to the **OU-Threshold Violation Files** window by pressing the **Output** button located on the menu toolbar and then pressing the **Thresh. Violation Files** button located on the lower right side of any Output Pathway window. From the menu select **Data | Output | Thresh. Violation Files**.



+

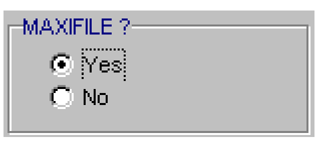

= Displays the **OU-Threshold Violation Files** window



OU-Threshold Violation Files window

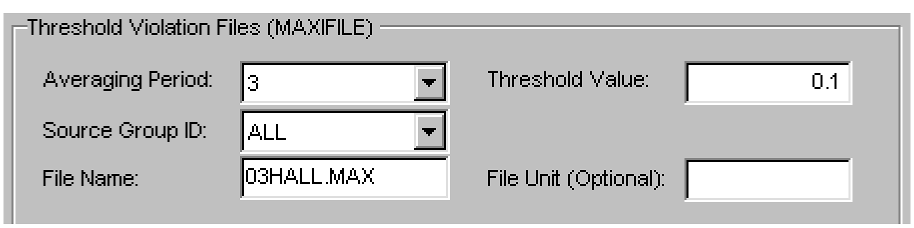
Defining Threshold Violations Files (Optional)

Since the **Threshold Violations Files (MAXIFILE)** option is optional, to select this option you have to check the **Yes** option button in the **MAXIFILE?** frame. The **No** option specifies that no **MAXIFILE** 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.



MAXIFILE? frame

To define one or more threshold violation files, you must specify the following:



Threshold Violation File parameters

- **Averaging Period:** This is one of the short-term averaging periods that were selected in the Control Pathway. To specify the **Averaging Period**, click on the arrow and select the averaging period from the drop-down list box.

- **Source Group ID:** This is the source group ID for which the **MAXIFILE** option is selected. The drop-down list box contains a list of all **Source Group IDs** created on the **SO-Source Groups** window. To specify the **Source Group ID** you should select one of the source groups IDs from the drop-down list box.

Note: Only one threshold violation file may be used for each **Averaging Period**/**Source Group** combination.

- **Threshold Value:** This is 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.

- **File Unit (Optional):** This optional parameter allows the user 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**.

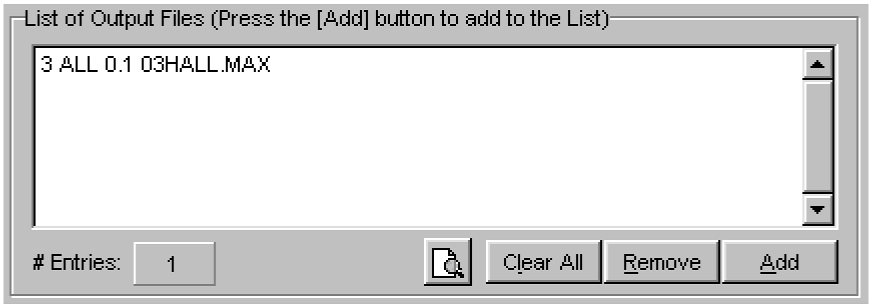
- **File Name:** This is the name of the threshold violation file to be created when you run the model. The file name can be up to 8 characters long. The default extension is *.MAX. By default, all threshold violation files will be written to the following location:

  - **ISCST3 Runs:** *project directory\ProjectName.IS*
  - **AERMOD Runs:** *project directory|ProjectName.AE*
  - **ISC-PRIME Runs:** *project directory\ProjectName.PR*

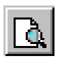
Press this button to define a different location (path) for your files.

Press this button if you want the files to be written to the ISC, AER, and PRI folders located in the current project directory.

- **List of Output Files:** After specifying all parameters, you must add them to the **List of Output Files** by clicking the **Add** button. This list is used to store all files specified. Only those combinations contained in the List of Output Files will be considered when running the model. The **# Entries** panel displays the total number of files added to the list. To store and manage the information in this list, you can use the buttons located on the bottom of the list. The purpose of these buttons is explained below:



List of Output Files

 ....Select an entry from the list and press this button to view the contents of the file. Please note that the threshold violation file is generated only after running the model.

............Clears all information from the List of Output Files.

............Removes the selected line from the List of Output Files.

............You must click this button to add the specified parameters to the List of Output Files.

Note: If more than one **Output Type** is selected in the Control Pathway, then the **MAXIFILE** 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 **MAXIFILE** output file.

CAUTION! The **MAXIFILE** 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, please be sure you have adequate space on your hard drive.

The Threshold Violation File (MAXIFILE) Format

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:

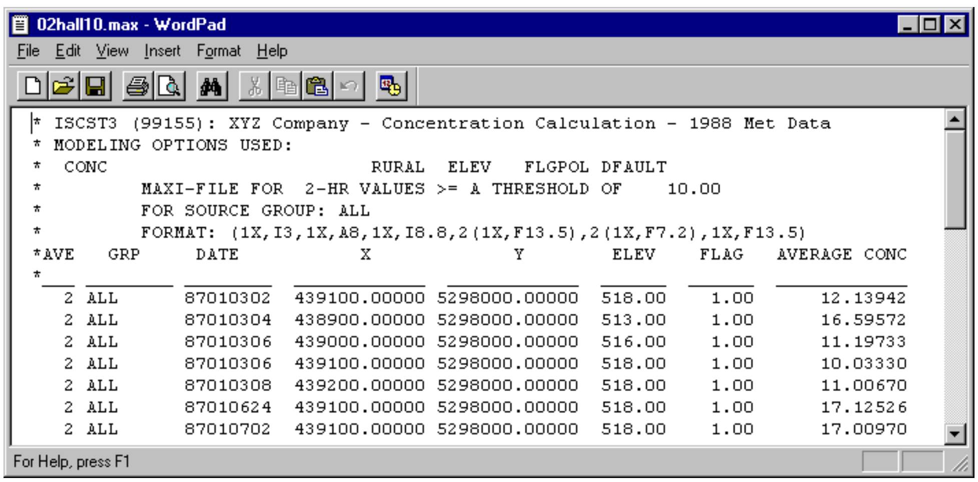
- Model Name + Model Version + First Title
- List of modeling option keywords applicable to the results

- 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



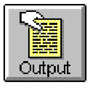

```
* ISCST3 (99155): XYZ Company - Concentration Calculation - 1988 Met Data
* MODELING OPTIONS USED:
*  CONC                         RURAL  ELEV   FLGPOL DFAULT
*        MAXI-FILE FOR  2-HR VALUES >= A THRESHOLD OF    10.00
*        FOR SOURCE GROUP: ALL
*        FORMAT: (1X,I3,1X,A8,1X,I8.8,2(1X,F13.5),2(1X,F7.2),1X,F13.5)
*AVE   GRP      DATE           X             Y         ELEV    FLAG   AVERAGE CONC
*___ ________ ________ _____________ _____________ _______ _______ _____________
   2 ALL      87010302  439100.00000 5298000.00000  518.00    1.00      12.13942
   2 ALL      87010304  438900.00000 5298000.00000  513.00    1.00      16.59572
   2 ALL      87010306  439000.00000 5298000.00000  516.00    1.00      11.19733
   2 ALL      87010306  439100.00000 5298000.00000  518.00    1.00      10.03330
   2 ALL      87010308  439200.00000 5298000.00000  518.00    1.00      11.00670
   2 ALL      87010624  439100.00000 5298000.00000  518.00    1.00      17.12526
   2 ALL      87010702  439100.00000 5298000.00000  518.00    1.00      17.00970
```

Example of a threshold violation file (MAXIFILE)

OU - Post-Processing Files Window

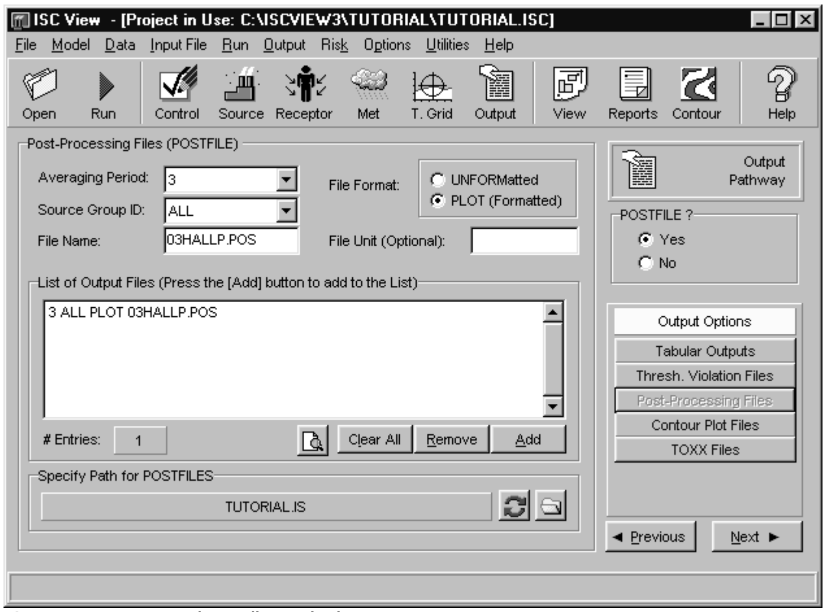
The Post-Processing File (POSTFILE) option produces files of concurrent (raw) results at each receptor suitable for post-processing. The **POSTFILE** option is available to you on the **OU-Post-Processing Files** window.

You have access to the **OU-Post-Processing Files** window by pressing the **Output** button located on the menu toolbar and then pressing **Post-Processing Files** button located on the lower right side of any Output Pathway window. From the menu select **Data | Output | Post-Processing Files**.



+ 

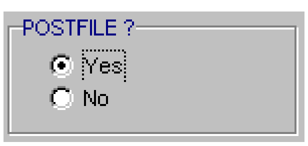
= Displays the **OU- Post-Processing Files** window



OU-Post-Processing Files window

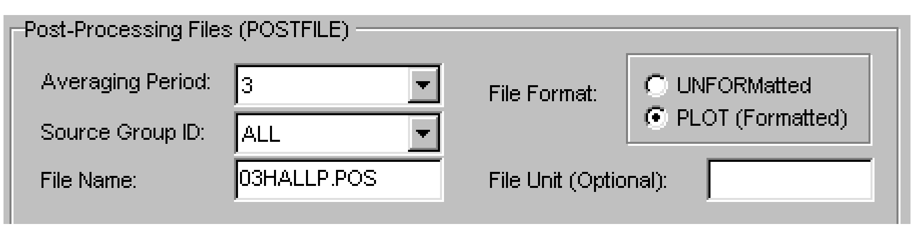
Defining Post-Processing Files (Optional)

To use the **Post-Processing Files (POSTFILE)** option you have to check the **Yes** option button in the **POSTFILE?** frame. The **No** option specifies that no **POSTFILE** files will be generated for the current run, although you still can store any **POSTFILE** information already added to the **List of Output Files** to be used in later runs.



POSTFILE? frame

To produce one or more post-processing files, you must specify the following parameters:



Post-Processing File parameters

- **Averaging Period:** This is one of the averaging periods that were selected in the Control Pathway, including **Period** or **Annual** averages. To specify the **Averaging Period**, click on the arrow and select the averaging period from the drop-down list box.

- **Source Group ID:** This is the source group ID for which the **POSTFILE** option is selected. The drop-down list box contains a list of all **Source Group IDs** created on the **SO-Source Groups** window. To specify the **Source Group ID** you should select one of the source group IDs from the drop-down list box.

- **File Format:** This parameter specifies the format of the **POSTFILE** output file, and may be either:

  1. **Unformatted:** This option tells the model to produce a POSTFILE in binary format (unformatted).

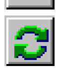
  2. **Formatted:** This option tells the model to produce a POSTFILE in ASCII format.

- **File Unit (Optional):** This optional parameter allows the user 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**.

- **File Name:** This is the name of the post-processing file to be created when you run the model. The file name can be up to 8 characters long. The default extension is *.POS. By default, all threshold violation files will be written to the following location:

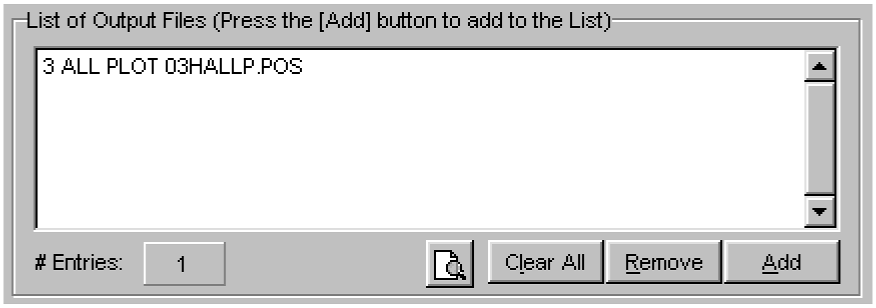
  - **ISCST3 Runs:** *project directory\ProjectName.IS*
  - **AERMOD Runs:** *project directory|ProjectName.AE*
  - **ISC-PRIME Runs:** *project directory\ProjectName.PR*

   Press this button to define a different location (path) for your files.

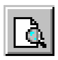
   Press this button if you want the files to be written to the ISC, AER, and PRI folders located in the current project directory.

- **List of Output Files:** After specifying all parameters, you must add them to the **List of Output Files** by clicking the **Add** button. The list is used to store all files specified. Only those combinations contained in the List of Output Files will be considered when running the model. The **# Entries** panel displays the total number of files added to the list. To store and manage the information in this list, you can use the buttons located on the bottom of the list. The purpose of these buttons is explained below:



List of Output Files

 ....Select an entry from the list and press this button to view the contents of the file. Please note that the post-processing file is generated only after running the model.

...........Clears all information from the List of Output Files.

...........Removes the selected line from the List of Output Files.

...........You must click this button to add the specified parameters to the List of Output Files.

Note: If more than one **Output Type** is selected in the Control Pathway, then the specified **Post-Processing Files (POSTFILE)** will include all of the output types selected in the following order: **Concentration**, **Total Deposition**, **Dry Deposition**, and/or **Wet Deposition**.

CAUTION! The **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:

$$\text{File Size (bytes)} = \frac{\text{(\# of Hrs/Yr)}}{\text{(\# of Hrs/Ave)}} * \text{(\# of Rec + 4)} * 4$$

# 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).

The Post-Processing File (POSTFILE) Format

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:

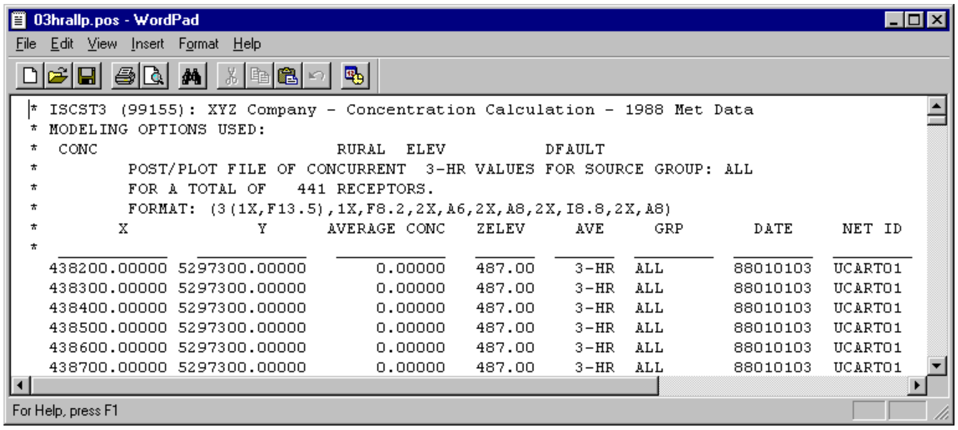
- **X:** X Coordinate of the receptor location
- **Y:** Y Coordinate of the receptor location
- **AVERAGE…:** Concentration or deposition value for that location
- **ZELEV:** Receptor terrain elevation
- **AVE:** Averaging period
- **GRP:** Source group ID
- **DATE or NUM HRS:** Date (YYMMDDHH) for the end of the averaging period for short-tem 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.

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:

$$\#\ \text{Data Records} = \frac{(\#\ \text{of Hrs/Day})}{(\#\ \text{Avg})} * \#\ \text{Days}\ (\#\ \text{of Rec})$$

$$= \frac{24}{3} * 31 * 441 = 109{,}368\ \text{data records}$$

# 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



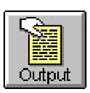

```
* ISCST3 (99155): XYZ Company - Concentration Calculation - 1988 Met Data
* MODELING OPTIONS USED:
*  CONC                         RURAL  ELEV          DFAULT
*         POST/PLOT FILE OF CONCURRENT  3-HR VALUES FOR SOURCE GROUP: ALL
*         FOR A TOTAL OF   441 RECEPTORS.
*         FORMAT: (3(1X,F13.5),1X,F8.2,2X,A6,2X,A8,2X,I8.8,2X,A8)
*        X             Y      AVERAGE CONC    ZELEV     AVE     GRP       DATE     NET ID
* ____________  ____________  ____________  ______  ______  ________  ________  ________
  438200.00000 5297300.00000       0.00000   487.00    3-HR  ALL       88010103  UCART01
  438300.00000 5297300.00000       0.00000   487.00    3-HR  ALL       88010103  UCART01
  438400.00000 5297300.00000       0.00000   487.00    3-HR  ALL       88010103  UCART01
  438500.00000 5297300.00000       0.00000   487.00    3-HR  ALL       88010103  UCART01
  438600.00000 5297300.00000       0.00000   487.00    3-HR  ALL       88010103  UCART01
  438700.00000 5297300.00000       0.00000   487.00    3-HR  ALL       88010103  UCART01
```

Example of a formatted post-processing file (POSTFILE)

OU - Contour Plot Files Window

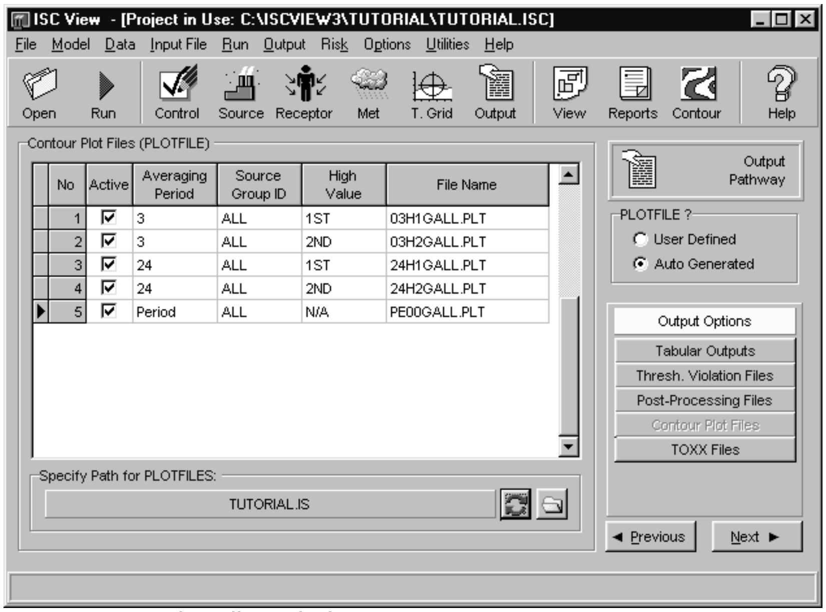
The **Contour Plot Files (PLOTFILE)** option produces files of design values that can be imported into graphics packages, such as POST View, in order to produce contour plots. The **PLOTFILE** option is available to you on the **OU - Contour Plot Files** window.

You have access to the **OU- Contour Plot Files** window by pressing the **Output** button located on the menu toolbar and then pressing the **Contour Plot Files** button located on the lower right side of any Output Pathway window. From the menu select **Data | Output | Contour Plot Files**.



+

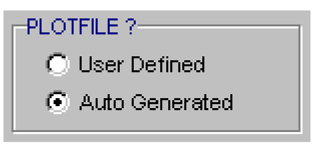

= Displays the **OU-Contour Plot Files** window



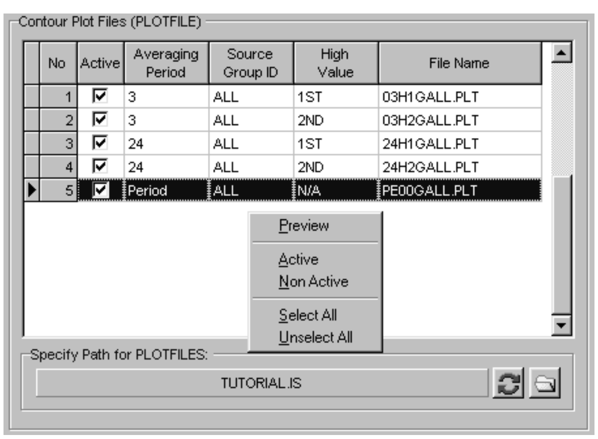
OU-Contour Plot Files window

Auto-Generated Contour Plot Files

Contour plot files can be automatically defined by ISC-AERMOD View with the **Auto Generated** option. By selecting this option, ISC-AERMOD View will automatically setup all possible combinations for generating plotfiles for each run.



By default, all the auto-generated plotfiles will be included in the Input File (**Active**). If you do not want to include one or more of these plotfiles, you should make them **Non-Active**. To make a plotfile active, just click on the **Active** check box.



To have access to the floating menu, right click anywhere on the table. The following options are available on the floating menu:

- **Preview:** Select this option to preview the contents of the selected plotfile. If more than one plotfile is selected, then only the first one will be previewed. You can select more than one plotfile by pressing down the Shift key. Please note that the contour plot file is generated only after running the model.

- **Active:** Select this option to mark as active the selected plotfiles. Only active plotfiles will be written into the input file.

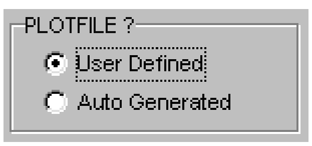
- **Non-Active:** Select this option to mark as non-active the selected plotfiles. Only active plotfiles will be written into the input file.

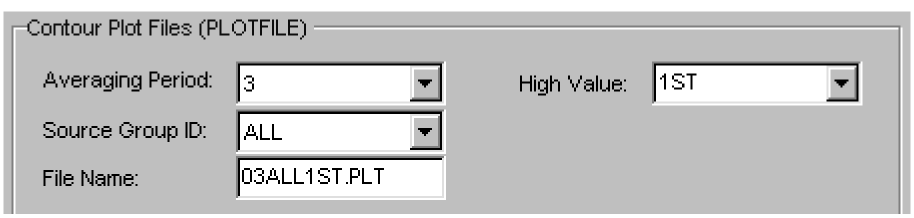
- **Select All:** Select this option if you want to select all auto-generated plotfiles.

- **Unselect All:** Select this option if you want to clear any selections.

Defining Contour Plot Files (PLOTFILES)

If you prefer to define your own plot files, then you should select the **User Defined** option and define the following parameters:





Contour Plot File parameters

- **Averaging Period:** This is one of the averaging periods that were selected in the Control Pathway, including **Period** or **Annual** averages. To specify the averaging period, click on the arrow and select the averaging period from the drop-down list box.

- **Source Group ID:** This is the source group ID for which the **PLOTFILE** option is selected. The drop-down list box contains a list of all **Source Group IDs** created on the **SO-Source Groups** window. To specify the **Source Group ID** you should select one of the source group IDs from the drop-down list box.

- **File Name:** This is the name of the contour plot file to be created when you run the model. The file name can be up to 8 characters long. The default extension is *.PLT. By default, all threshold violation files will be written to the following location:

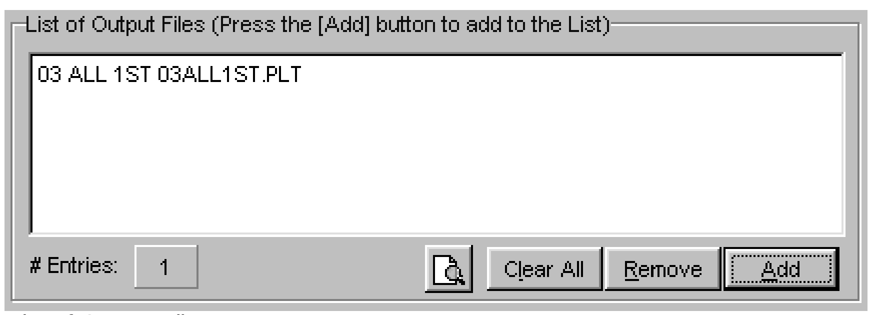
  - **ISCST3 Runs:** *project directory\ProjectName.IS*
  - **AERMOD Runs:** *project directory|ProjectName.AE*
  - **ISC-PRIME Runs:** *project directory\ProjectName.PR*

 Press this button to define a different location (path) for your files.

 Press this button if you want the files to be written to the ISC, AER, and PRI folders located in the current project directory.

- **High Value:** Specifies which short-term high values are to be output, **1ST** to **6TH** (**1ST** for the first highest at each receptor, **2ND** for the second highest at each receptor, etc…).

- **List of Output Files:** After specifying all parameters, you must add them to the **List of Output Files** by clicking the **Add** button. The list is used to store all files specified. Only those combinations contained in the List of Output Files will be considered when running the model. The **# Entries** panel displays the total number of files added to the list. To store and manage the information in this list, you can use the buttons located on the bottom of the list. The purpose of these buttons is explained below:



List of Output Files

 .... Select an entry from the list and press this button to view the contents of the file. Please note that the contour plot file is generated only after running the model.

.... Clears all information from the List of Output Files.

.... Removes the selected line from the List of Output Files.

.... You must click this button to add the specified parameters to the List of Output Files.

Note: The selected **High Value** must already have been specified on the **RECTABLE** option on the **OU-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.

Note: More than one **PLOTFILE** option can be specified for each combination of **Averaging Period**, **Source Group**, and **High Value**, and a different **File Name** should be used for each file.

Note: 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.

The Contour Plot File (PLOTFILE) Format

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:

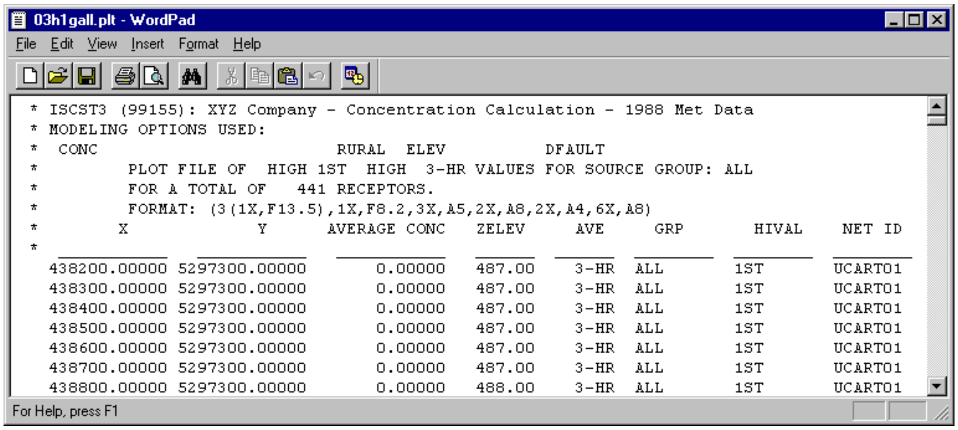
- Model Name + Model Version + First Title
- List of modeling option keywords applicable to the results
- Averaging Period + High Value + Source Group included in the file
- The total number of receptors included
- The Fortran format used for writing the data records
- Column headers for the variables 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.

The contour plot file will include the selected high value (e.g., 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).



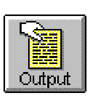

```
* ISCST3 (99155): XYZ Company - Concentration Calculation - 1988 Met Data
* MODELING OPTIONS USED:
*  CONC                         RURAL  ELEV           DFAULT
*         PLOT FILE OF  HIGH 1ST  HIGH  3-HR VALUES FOR SOURCE GROUP: ALL
*         FOR A TOTAL OF   441 RECEPTORS.
*         FORMAT: (3(1X,F13.5),1X,F8.2,3X,A5,2X,A8,2X,A4,6X,A8)
*        X             Y      AVERAGE CONC   ZELEV     AVE     GRP       HIVAL    NET ID
*  ____________ ____________  ____________ ______   ______  ________   ________  ________
  438200.00000 5297300.00000       0.00000  487.00    3-HR  ALL        1ST       UCART01
  438300.00000 5297300.00000       0.00000  487.00    3-HR  ALL        1ST       UCART01
  438400.00000 5297300.00000       0.00000  487.00    3-HR  ALL        1ST       UCART01
  438500.00000 5297300.00000       0.00000  487.00    3-HR  ALL        1ST       UCART01
  438600.00000 5297300.00000       0.00000  487.00    3-HR  ALL        1ST       UCART01
  438700.00000 5297300.00000       0.00000  487.00    3-HR  ALL        1ST       UCART01
  438800.00000 5297300.00000       0.00000  488.00    3-HR  ALL        1ST       UCART01
```

Example of a contour plot file (PLOTFILE)

OU - TOXX Input Files Window

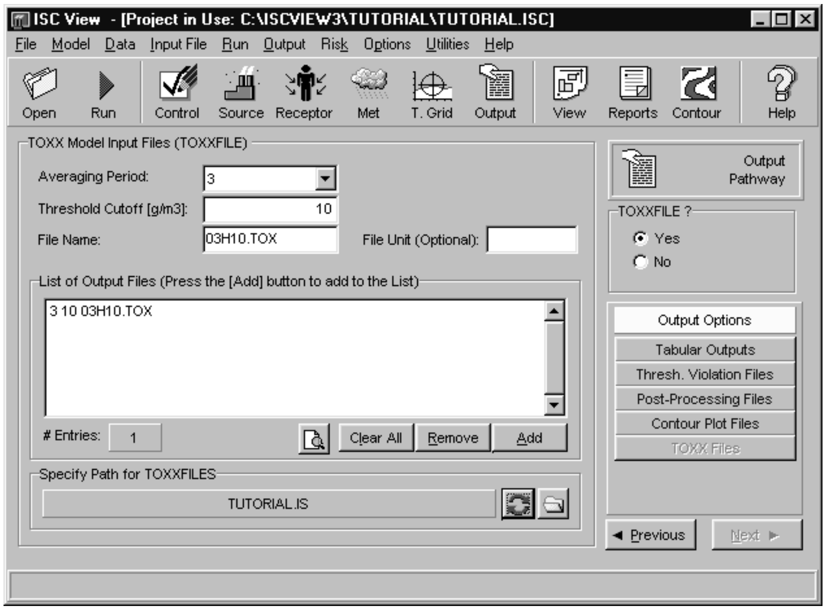
The **TOXX Input Files (TOXXFILE)** option produces unformatted (binary) files of raw results above a threshold value with a special structure for use with the TOXX model component of TOXST. The **TOXXFILE** option is available to you on the **OU-TOXX Input Files** window.

You have access to the **OU- TOXX Input Files** window by pressing the **Output** button located on the menu toolbar and then pressing **TOXX Input Files** button located on the lower right side of any Output Pathway window. From the menu select **Data | Output | TOXX Input Files**.



+ 

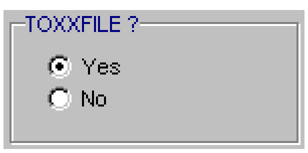
= Displays the **OU-Contour Plot Files** window



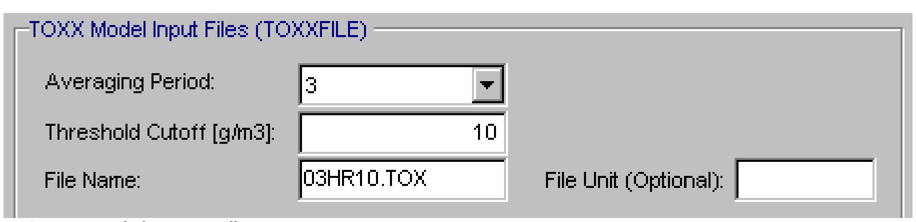
OU-TOXX Input Files window

Defining TOXX Model Input Files (Optional)

To select the **TOXX Input Files (TOXXFILE)** option, you must check the **Yes** option button in the **TOXXFILE?** frame. The **No** option specifies that no **TOXXFILE** files will be generated for the current run, although you still can store any **TOXXFILE** information already added to the **List of Output Files** to be used in later runs.



To produce one or more TOXX model input files, you must specify the following parameters:



TOXX Model Input File parameters

- **Averaging Period:** This is one of the short-term averaging periods that were selected in the Control Pathway (excluding **Period** or **Annual** averages). To specify the averaging period, click on the arrow and select the averaging period from the drop-down list box.

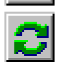
- **Threshold (g/m^3):** This is the user-specified threshold cutoff value. Note that the units of the **Threshold Cutoff** parameter are in g/m^3, regardless of the input and output units selected in the **SO - Emission Output Units** window.

- **File Unit (Optional):** This optional parameter allows the user 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**.

- **File Name:** This is the name of the TOXX model input file to be created when you run the model. The file name can be up to 8 characters long. The default extension is *.TOX. By default, all files will be written to the following location:

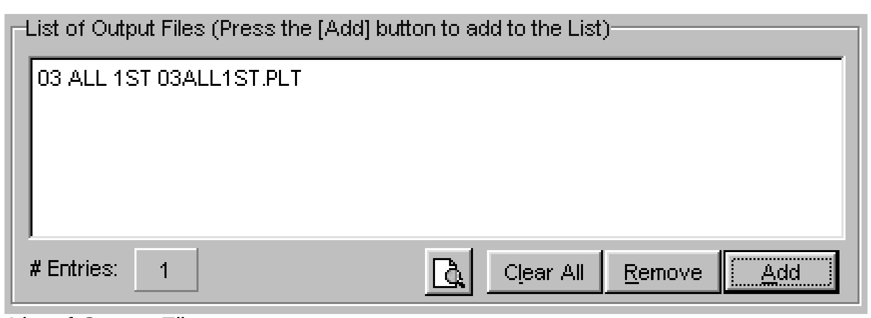
  - **ISCST3 Runs:** *project directory\ProjectName.IS*
  - **AERMOD Runs:** *project directory|ProjectName.AE*
  - **ISC-PRIME Runs:** *project directory\ProjectName.PR*

Press this button to define a different location (path) for your files.



Press this button if you want the files to be written to the ISC, AER, and PRI folders located in the current project directory.

- **List of Output Files:** After specifying all parameters, you must add them to the **List of Output Files** by clicking the **Add** button. The list is used to store all files specified. Only those combinations contained in the List of Output Files will be considered when running the model. The **# Entries** panel displays the total number of files added to the list. To store and manage the information in this list, you can use the buttons located on the bottom of the list. The purpose of these buttons is explained below:



List of Output Files

 ....Select an entry from the list and press this button to view the contents of the file. Please note that the TOXXFILE is generated only after running the model.

Clear All ....Clears all information from the List of Output Files.

Remove ....Removes the selected line from the List of Output Files.

Add ....You must click this button to add the specified parameters to the List of Output Files.

Note: Only one **TOXXFILE** file may be used for each **Averaging Period**.

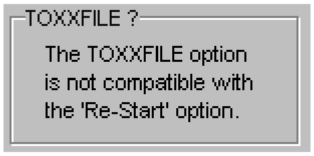
Note: The **TOXXFILE** option can be specified for each **Averaging Period**, but a different **File Name** should be used for each file. Different filenames should be specified for each file since the structure of the output file generated by the **TOXXFILE** option does not allow for a clear way to distinguish between results for different averaging periods.

Note: 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 other than the **1-hour** average is specified. This is because the TOXST model currently supports only **1-hour** averages.

Note: If more than one **Output Type** is in the Control Pathway, the **TOXXFILE** threshold 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 **TOXXFILE** output file.

Note: When using the **TOXXFILE** option, the user will normally place a single source in each source group.

Note: The **TOXXFILE** option is not compatible with the **Re-Start File** option if you have selected the **Re-Start File** option on the **CO-Optional Files** window, then ISC-AERMOD View will disable the options on the **OU-TOXX Input Files** window and will display a message on the **TOXXFILE?** frame.



CAUTION! : 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. Thus, please be sure you have adequate space on your hard drive.

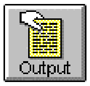
OU–Season by Hour Files Window

(ISCST3 Only)

The Season by Hour-of-Day Output Files option (SEASONHR) is a non-default TOXICS option containing average results (CONC, DEPOS, DDEP and/or WDEP) by season and hour-of-day. This option is only applicable to the ISCST3 model. To be able to define Season by Hour-of-Day Output files, you must select the following options in the Control Pathway:

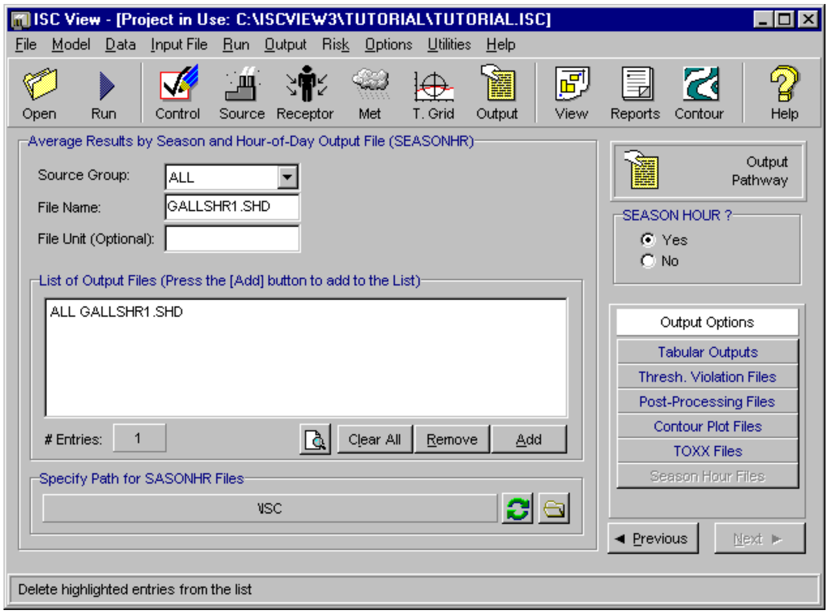
- Non-Default Options (**CO-Dispersion Options** window)
- Air Toxics Options (**CO-Dispersion Options** window)
- Season by Hour-of-Day Output Option (**CO-Air Toxics Options** window)

You have access to the **OU-Season by Hour Files** window by pressing the **Output** button located on the menu toolbar and then pressing **Season by Hour Files** button located on the lower right side of any Output Pathway window. From the menu select **Data | Output | Season by Hour Files**.



+ 

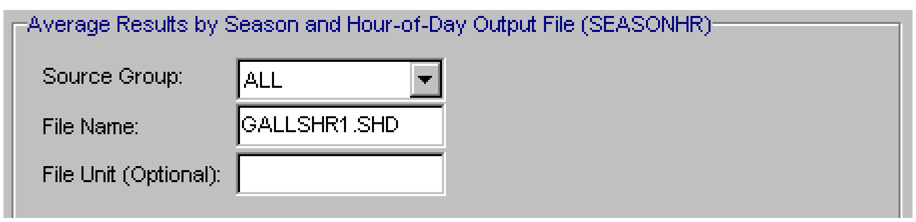
= Displays the **OU-Season by Hour Files** window



OU-Season by Hour Files window

Defining Season by Hour Output Files (SEASONHR)

To define Season by Hour-of-Day Output files, you must specify the following parameters:

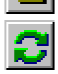


- **Source Group ID:** This is the source group ID to be output. The drop-down list box contains a list of all **Source Group IDs** created on the **SO-Source Groups** window. To specify the **Source Group ID** you should select one of the source group IDs from the drop-down list box.

- **File Name:** This is the name of the season by hour output file to be created when you run the model. The file name can be up to 8 characters long. The default extension is *.SHD. By default, all files will be written to the following location:

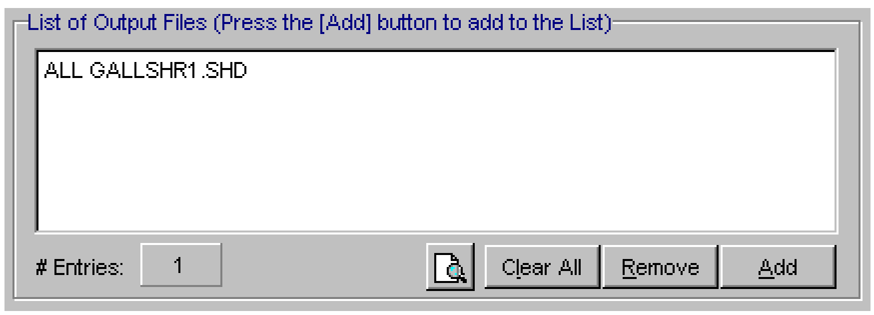
  - **ISCST3 Runs:** *project directory\ProjectName.IS*

 Press this button to define a different location (path) for your files.

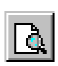
 Press this button if you want the files to be written to the ISC folder located in the current project directory.

- **File Unit (Optional):** This optional parameter allows the user 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 based on the formula 302+IGRP*10, where IGRP is the group index number.

- **List of Output Files:** After specifying all parameters, you must add them to the **List of Output Files** by clicking the **Add** button. The list is used to store all files specified. Only those combinations contained in the List of Output Files will be considered when running the model. The **# Entries** panel displays the total number of files added to the list. To store and manage the information in this list, you can use the buttons located on the bottom of the list. The purpose of these buttons is explained below:



List of Output Files

 ....Select an entry from the list and press this button to view the contents of the file. Please note that the SEASONHR is generated only after running the model.

...........Clears all information from the List of Output Files.

...........Removes the selected line from the List of Output Files.

...........You must click this button to add the specified parameters to the List of Output Files.

The Season by Hour Output File (SEASONHR) Format

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

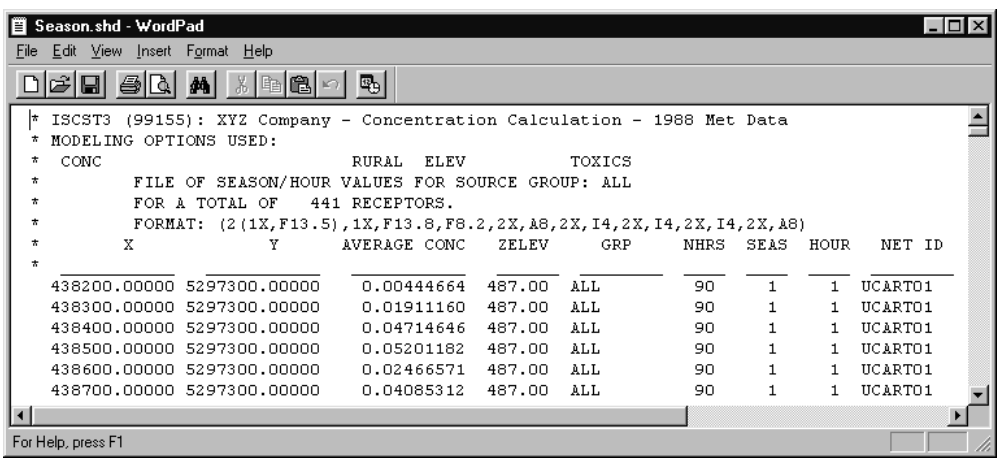
Data Record:

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.

A sample from a SEASONHR output file is shown below:



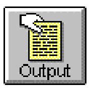

```
* ISCST3 (99155): XYZ Company - Concentration Calculation - 1988 Met Data
* MODELING OPTIONS USED:
*  CONC                          RURAL  ELEV          TOXICS
*         FILE OF SEASON/HOUR VALUES FOR SOURCE GROUP: ALL
*         FOR A TOTAL OF   441 RECEPTORS.
*         FORMAT: (2(1X,F13.5),1X,F13.8,F8.2,2X,A8,2X,I4,2X,I4,2X,I4,2X,A8)
*        X             Y      AVERAGE CONC   ZELEV     GRP     NHRS  SEAS  HOUR   NET ID
*  ____________  ____________  ____________  ______  ________  ____  ____  ____  ________
  438200.00000 5297300.00000    0.00444664  487.00  ALL         90     1     1  UCART01
  438300.00000 5297300.00000    0.01911160  487.00  ALL         90     1     1  UCART01
  438400.00000 5297300.00000    0.04714646  487.00  ALL         90     1     1  UCART01
  438500.00000 5297300.00000    0.05201182  487.00  ALL         90     1     1  UCART01
  438600.00000 5297300.00000    0.02466571  487.00  ALL         90     1     1  UCART01
  438700.00000 5297300.00000    0.04085312  487.00  ALL         90     1     1  UCART01
```

OU – Rank Files Window

(AERMOD Only)

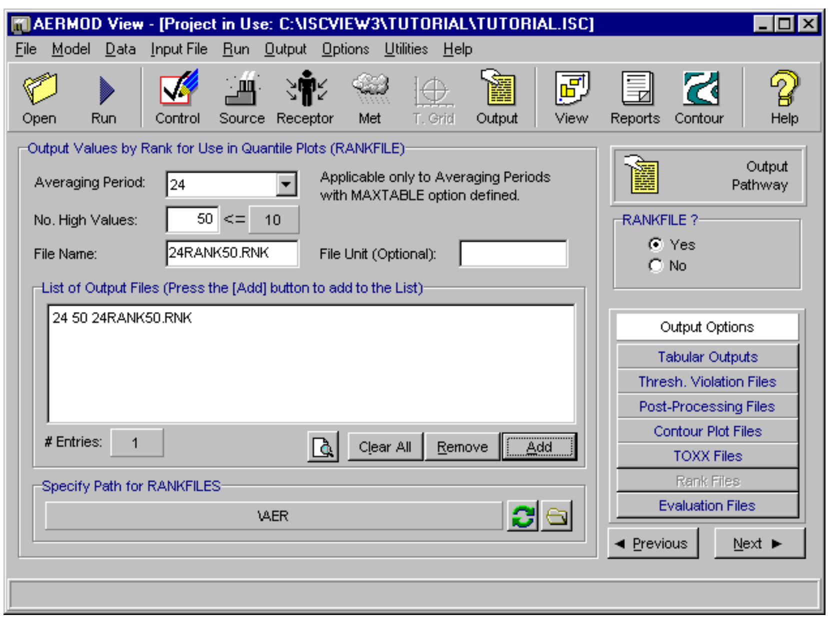
The Rank File Output option (RANKFILE) is applicable only to the AERMOD model. The Rank File Output option allows you to create output files of the maximum concentration values by rank, suitable for generating Q-Q or quantile plots. The data contained in the RANKFILE output is based on the MAXTABLE arrays, except that only one occurrence per data period is included. In order to use the RANKFILE option for a particular averaging period, the MAXTABLE option should first be defined in the **OU-Tabular Outputs** window.

You have access to the **OU-Rank Files** window by pressing the **Output** button located on the menu toolbar and then pressing **Rank Files** button located on the lower right side of any Output Pathway window. From the menu select **Data | Output | Rank Files**.



+

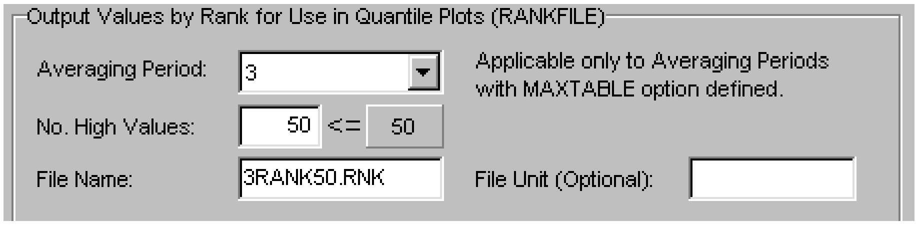

= Displays the **OU-Rank Files** window



OU-Rank Files window

Defining Rank Files (RANKFILES)

To define Rank files, you must specify the following parameters:



- **Averaging Period:** This is one of the short-term averaging periods that were selected in the Control Pathway (excluding **Period** or **Annual** averages). To specify the averaging period, click on the arrow and select the averaging period from the drop-down list box.

- **No. High Values:** This is the number of high values to be ranked. The number of high values must be less than or equal to the Maximum Values for the MAXTABLE option (**OU-Tabular Outputs** window).

- **File Name:** This is the name of the rank file to be created when you run the model. The file name can be up to 8 characters long. The default extension is *.RNK. By default, all files will be written to the following location:

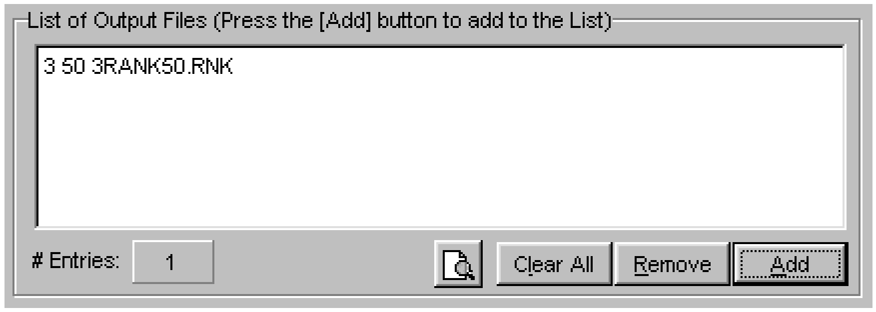
  - **AERMOD Runs:** *project directory\ProjectName.AE*

Press this button to define a different location (path) for your files.

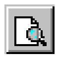
Press this button if you want the files to be written to the default directory (AER) located in the current project directory.

- **File Unit (Optional):** This optional parameter allows the user 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.

- **List of Output Files:** After specifying all parameters you must add them to the **List of Output Files** by clicking the **Add** button. The list is used to store all files specified. Only those combinations contained in the List of Output Files will be considered when running the model. The **# Entries** panel displays the total number of files added to the list. To store and manage the information in this list, you can use the buttons located on the bottom of the list. The purpose of these buttons is explained below:



List of Output Files


.................Select an entry from the list and press this button to view the contents of the file. Please note that the RANKFILE is generated only after running the model.

...........Clears all information from the List of Output Files.

...........Removes the selected line from the List of Output Files.

...........You must click this button to add the specified parameters to the List of Output Files.

The Rank File (RANKFILE) Format

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:

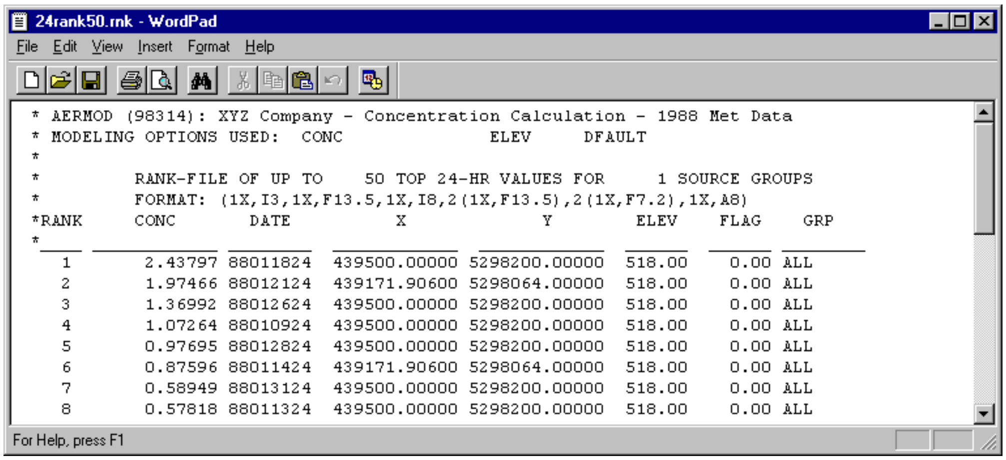
The variables provided on each data record include:

- **RANK:** Rank
- **CONC:** Concentration value
- **DATE:** Date (YYMMDDHH)
- **X:** X coordinate for the receptor location
- **Y:** Y coordinate for the receptor location

- **ELEV:** Receptor terrain elevation
- **FLAG:** Flagpole receptor height
- **GRD:** Source Group ID

Since only one occurrence per data period is included, the RANKFILE may not include the number of ranked values requested.

A sample from a RANKFILE output file is shown below:



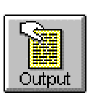
```
24rank50.rnk - WordPad
File  Edit  View  Insert  Format  Help

* AERMOD (98314): XYZ Company - Concentration Calculation - 1988 Met Data
* MODELING OPTIONS USED:  CONC                 ELEV      DFAULT
*
*          RANK-FILE OF UP TO    50 TOP 24-HR VALUES FOR     1 SOURCE GROUPS
*          FORMAT: (1X,I3,1X,F13.5,1X,I8,2(1X,F13.5),2(1X,F7.2),1X,A8)
*RANK      CONC        DATE           X              Y          ELEV     FLAG     GRP
*
   1      2.43797 88011824  439500.00000 5298200.00000  518.00    0.00 ALL
   2      1.97466 88012124  439171.90600 5298064.00000  518.00    0.00 ALL
   3      1.36992 88012624  439500.00000 5298200.00000  518.00    0.00 ALL
   4      1.07264 88010924  439500.00000 5298200.00000  518.00    0.00 ALL
   5      0.97695 88012824  439500.00000 5298200.00000  518.00    0.00 ALL
   6      0.87596 88011424  439171.90600 5298064.00000  518.00    0.00 ALL
   7      0.58949 88013124  439500.00000 5298200.00000  518.00    0.00 ALL
   8      0.57818 88011324  439500.00000 5298200.00000  518.00    0.00 ALL
For Help, press F1
```

OU – Evaluation Files Window (AERMOD Only)

(AERMOD Only)

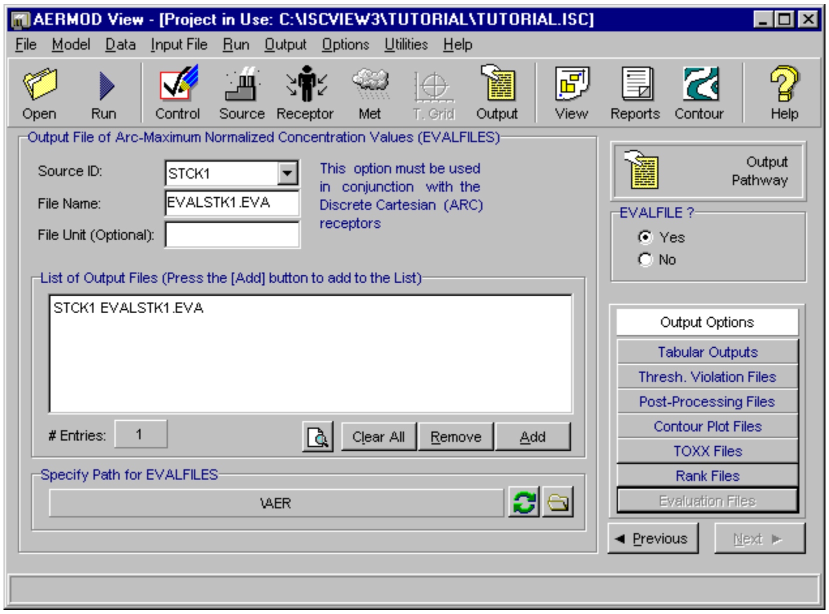
The Evaluation Files option (EVALFILE) is applicable only to the AERMOD model. The Evaluation Files option produces file of output values, including arc-maximum normalized concentration values for each hour of meteorology and for each source specified. The arc groupings of the receptors must be specified using the Discrete Cartesian (ARC) receptors.

You have access to the **OU-Evaluation Files** window by pressing the **Output** button located on the menu toolbar and then pressing **Evaluation Files** button located on the lower right side of any Output Pathway window. From the menu select **Data | Output | Evaluation Files**.



+

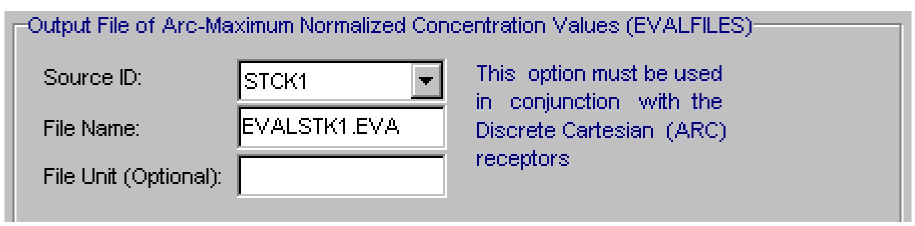

= Displays the **OU-Evaluation Files** window



OU-Evaluation Files window

Defining Evaluation Files (EVALFILE)

To define Evaluation files, you must specify the following parameters:

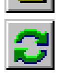


- **Source ID:** This is the source ID for which EVALFILE results are requested. The drop-down list box contains a list of all **Source IDs** created on the **SO-Source Inputs** window. To specify the **Source ID** you should select one of the source IDs from the drop-down list box.

- **File Name:** This is the name of the evaluation file to be created when you run the model. The file name can be up to 8 characters long. The default extension is *.EVA. By default, all files will be written to the following location:

  - **AERMOD Runs:** *project directory\ProjectName.AE*

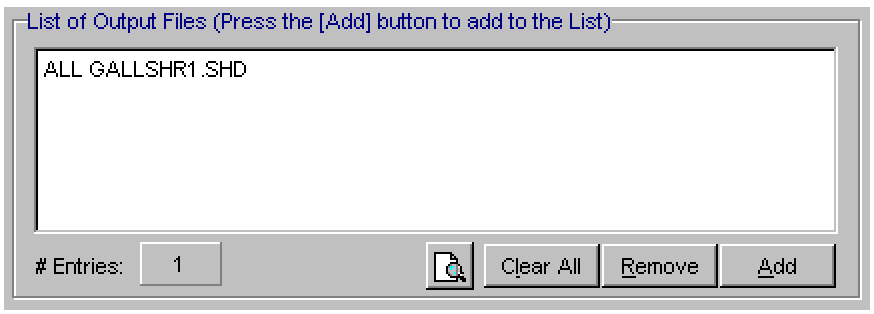
Press this button to define a different location (path) for your files.



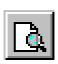
Press this button if you want the files to be written to the AER folder located in the current project directory.

- **File Unit (Optional):** This optional parameter allows the user 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 omitted, the model will dynamically allocate a unique file unit.

- **List of Output Files:** After specifying all parameters, you must add them to the **List of Output Files** by clicking the **Add** button. The list is used to store all files specified. Only those combinations contained in the List of Output Files will be considered when running the model. The **# Entries** panel displays the total number of files added to the list. To store and manage the information in this list, you can use the buttons located on the bottom of the list. The purpose of these buttons is explained below:



List of Output Files

 ....Select an entry from the list and press this button to view the contents of the file. Please note that the EVALFILE is generated only after running the model.

...........Clears all information from the List of Output Files.

...........Removes the selected line from the List of Output Files.

...........You must click this button to add the specified parameters to the List of Output Files.

The Evaluation File (EVALFILE) Format

For each hour of meteorological data processed and for each selected source, the EVALFILE option outputs five records containing the following:

Record 1:

- Source ID (eight characters)
- Date variable (YYMMDDHH)
- Arc ID (eight characters)
- 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 σ_v corresponding to arc maximum (m/s)
- Effective σ_w corresponding to arc maximum (m/s)
- σ_y 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 σ_z for current hour if stable (m)
- Buoyancy flux for current hour (m^4/s^3)
- Momentum flux for current hour (m^4/s^2)

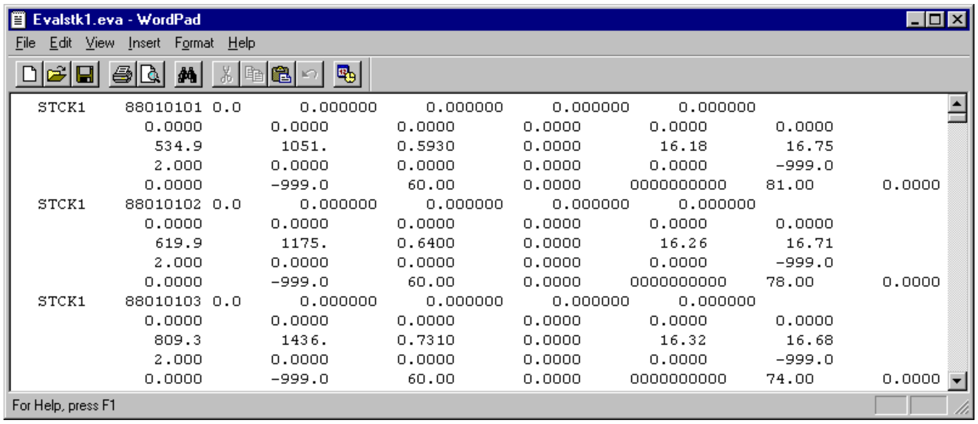
Record 4:

- Bowen ratio for current hour
- Plume penetration factor for current hour
- Centerline 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
- Developmental option settings place holder (string of 10 zeroes)
- Flow vector for current hour (degrees)
- Effective height for stable plume reflections (m)

A sample from an EVALFILE output file is shown below:


Evalstk1.eva - WordPad

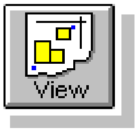
File Edit View Insert Format Help

```
  STCK1    88010101 0.0      0.000000     0.000000     0.000000     0.000000
                0.0000       0.0000       0.0000       0.0000       0.0000       0.0000
                 534.9        1051.       0.5930       0.0000        16.18        16.75
                 2.000       0.0000       0.0000       0.0000       0.0000       -999.0
                0.0000       -999.0        60.00       0.0000   0000000000     81.00         0.0000
  STCK1    88010102 0.0      0.000000     0.000000     0.000000     0.000000
                0.0000       0.0000       0.0000       0.0000       0.0000       0.0000
                 619.9        1175.       0.6400       0.0000        16.26        16.71
                 2.000       0.0000       0.0000       0.0000       0.0000       -999.0
                0.0000       -999.0        60.00       0.0000   0000000000     78.00         0.0000
  STCK1    88010103 0.0      0.000000     0.000000     0.000000     0.000000
                0.0000       0.0000       0.0000       0.0000       0.0000       0.0000
                 809.3        1436.       0.7310       0.0000        16.32        16.68
                 2.000       0.0000       0.0000       0.0000       0.0000       -999.0
                0.0000       -999.0        60.00       0.0000   0000000000     74.00         0.0000
```

For Help, press F1

CHAPTER 9

Graphical Input

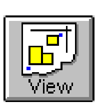


This chapter explains the options available under the Graphical Input window. The Graphical Input window allows you to view all the sources, receptors, grids, and buildings you have defined for the current run. In this window you will be able to import site maps in various file formats and graphically input sources, discrete receptors, receptor grids, and the plant boundary.

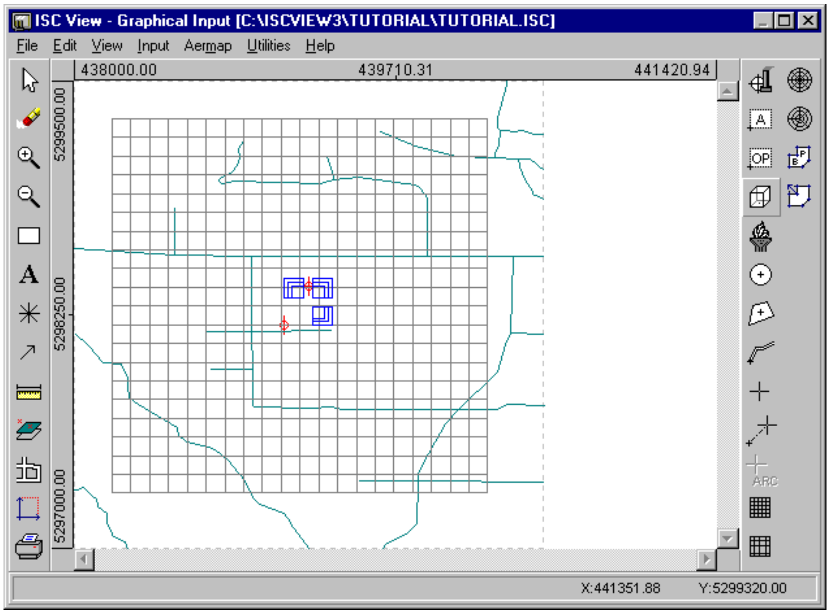
Contents

The Graphical Input Window

In the Graphical Input window you can view and graphically define your sources, receptors and plant boundary. You can also import site base maps in various file formats. The **Graphical Input** window can be displayed at any time during your project by pressing the **View** toolbar button located on the menu toolbar.



View toolbar button



Graphical Input window

The following components of the Graphical Input window will be discussed in the sections that follow:

- **Menu Bar**
- **Toolbars**
- **Axis Labels**
- **Scroll Bars**
- **Status Bar**
- **X and Y Coordinates**
- **Drawing Area**

Menu Bar

The menu in the Graphical Input window is different from the ISC-AERMOD View main window. The following is the description of each menu option in the Graphical Input window:

File (Alt, F)

Import | Base Map....... Displays the following submenu options:

- **DXF:** This option allows you to import a base map in AutoCAD DXF file format (*.dxf).
- **BMP:** This option allows you to import a bitmap base map image (*.bmp).
- **DLG:** This option allows you to import a USGS Digital Line Graph map (*.dlg).
- **LULC:** This option allows you to import a USGS Land Use and Land Cover map (*.luc).
- **Shapefile:** This option allows you to import an ArcView Shapefile (*.shp).

Export | Base Map....... Displays the following submenu options:

- **BMP:** Displays the **Export As Bitmap** dialog box, allowing you to save the contents of the drawing area as a Windows bitmap.

Print............................. Displays the **Print Preview** dialog box from where you can select printing options and print the contents of the drawing area.

Print Preferences........ Displays the **Preferences** dialog box, allowing you to define printing and labeling options.

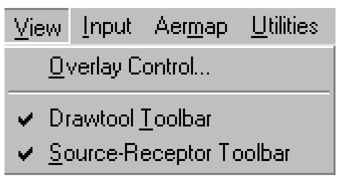
Exit............................... Closes the Graphical Input window.

Edit (Alt, E)

Move Site ....This option allows you to move the coordinates of all the sources and receptors by a specified value.

Copy (Ctrl+C) ....This option allows you to copy the contents of the drawing area to the Clipboard, so you can paste to any Windows application that supports pasting from the clipboard. As a Windows Metafile image, the clipboard image can be re-sized to accommodate your needs and preferences.

View (Alt, V)

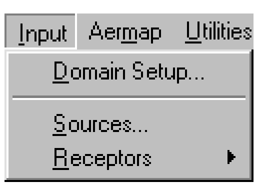


Overlay Control ....Displays the **Overlay Control** dialog box, where you control which objects are to be displayed on the drawing area.

Drawtool Toolbar ....Displays or hides the Drawtool toolbar.

Source-Receptor Toolbar ....Displays or hides the Source-Receptor toolbar.

Input (Alt, I)



Domain Setup ....Displays the **Domain Setup** dialog box, where you can define the domain extents of the drawing area.

Sources ....Displays the **Source Inputs** dialog box in which you can view or define one or more sources.

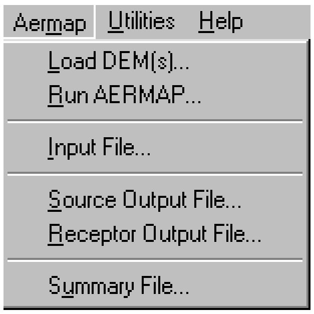
Receptors ....Displays the following submenu options:

- **Receptor Summary:** Displays the **Receptor Summary** dialog box, which contains terrain height options, flagpole receptors options, and a summary table listing all receptor types already defined in your project.

- **Network Types:** Displays submenus for the all the network types.

- **Discrete Locations:** Displays submenus for all the discrete receptor types.

Aermap (Alt, A)



Load DEM(s)................Displays the **DEM Import** dialog box, where you can define the DEM files to be used by AERMAP.

Run AERMAP ....Runs the U.S. EPA AERMAP model.

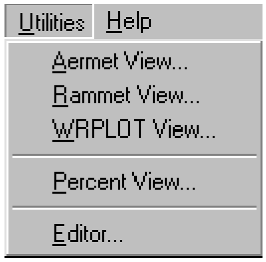
Input File ....Displays the input file that was generated to run AERMAP.

Source Output File......Displays the output file generated by AERMAP that contains the source data, including source elevations.

Receptor Output File ..Displays the output file 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.

Utilities (Alt, U)



Aermet View .... This utility preprocesses your met data for use with the AERMOD model.

Rammet View .... This utility preprocesses your met data for use with the ISCST3 and ISC-PRIME models.

WRPLOT View .... This utility creates wind roses of your met data.

Percent View .... This utility generates percentile plots of a given averaging period contained within a Post Processing File (POSTFILE).

Editor .... Opens Windows WordPad.

Help (Alt, H)

Contents .... Displays the Help Contents, in which you can select topics.

Search For Help on ..... Lets you search for Help on a particular topic.

Help on Help .... Displays information for How to Use Help.

Team .... Displays information on the development team for ISC-AERMOD View.

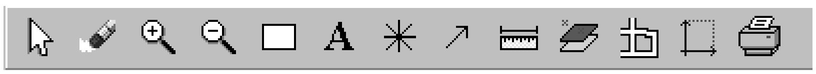
Technical Support ...... Displays a dialog box containing available technical support options for Lakes Environmental software.

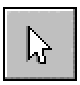
About ....Displays the copyright notice and version number for ISC-AERMOD View.

Toolbars

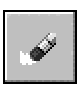
Two toolbars are available in the Graphical Input window: the **Drawtool** toolbar and the **Source-Receptor** toolbar. These toolbars can be docked or floating. A docked toolbar is a toolbar that is attached to one edge of the program window. You can dock the toolbars below the menu toolbar, above the status bar, or to the left or right edge of the Graphical Input window. A floating toolbar is a toolbar that floats and is not attached to the edge of the program window. To move a docked toolbar, click on the toolbar and drag to the new location. If you drag the toolbar to the edge of the program window, it becomes a docked toolbar.

The tools available on the **Drawtool** toolbar are:

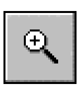




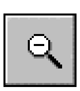
Select tool: With this tool, you can select annotation objects from the drawing area, move them around, and double-click on them to display their properties. You can also select the object and click the right mouse button to display its properties. With this tool, you will be able to select (click on) an object (source, receptor, grid, building, etc.) from the drawing area and get information on the type and name of the selected object, which is displayed on the **Status bar**.



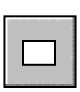
Delete tool: Use this tool to delete an object from the drawing area. Click the **Delete** tool and then click on the desired object to be deleted. A message box appears requesting confirmation that you want to delete the selected object.



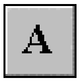
Zoom In tool: This tool allows you to magnify a portion of the drawing area. To zoom in a portion of the drawing area, click on the **Zoom In** tool. Next, move the cursor into the drawing area and, holding down the left mouse button, drag up or down on a diagonal until the dashed rectangle contains the area you want to magnify. Finally release the mouse left button.



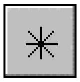
Zoom Out tool: Use this tool to return the drawing area back to the initial scale size.



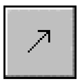
Rectangle Annotation tool: Use this tool to draw any rectangle on the drawing area. Click this tool and, holding down the left mouse button, draw a rectangle on the drawing area. Release the left mouse button.



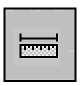
Text Annotation tool: Use this tool to write text on the drawing area. Click this tool and click on the drawing area where you want to write the text. The **Text Annotation** dialog box is displayed. Type the text and press the **Font** button to select the font type, size, style, color, etc.



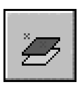
Marker Annotation tool: This tool enables you to place a marker on the drawing area. Click this tool and click the drawing area where you want to place a marker. The **Marker Annotation** dialog box is displayed. Select the marker style and color and click the OK button.



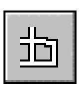
Arrow Annotation tool: Use this to draw an arrow or line on the drawing area. Click this tool and holding down the left mouse button, draw a line on the drawing area. Release the left mouse button.



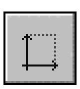
Measure Distance tool: This tool allows you to measure the length of objects. You simply click on the start point where you wish to measure from and drag the cursor to the end point. A line will be formed showing you what you are measuring and the distance readout will appear beside the cursor. The units of measurement are meters.



Overlay Control tool: Displays the **Overlay Control** dialog box, where you control which overlays are to be displayed on the drawing area. This tool is very useful when you want to hide certain objects but you don't want to delete them.



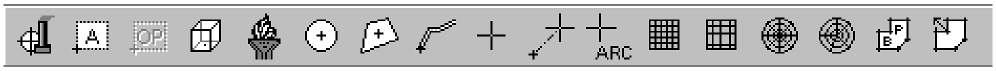
Import Base Map tool: Use this tool to import a site map to be displayed in the drawing as a backdrop. A floating menu with available file format import options is displayed when this tool is pressed.

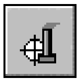


Domain Setup tool: Displays the **Domain Setup** dialog box, where you can define the domain extents of the drawing area.

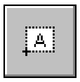
Print tool: Displays the **Print Preview** dialog box from where you can select printing options and print the contents of the drawing area.

The tools available on the **Source-Receptor** toolbar are:

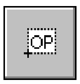




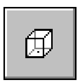
Point Source tool: Use this tool to graphically define the location of point sources.



Area Source tool: Use this tool to graphically define the location of area sources.

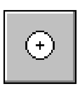


Open Pit Source tool: Use this tool to graphically define the location of open pit sources.

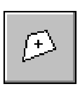


Volume Source tool: Use this tool to graphically define the location of volume sources.

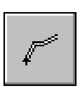
Flare Source tool: Use this tool to graphically define the location of flare sources.



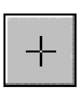
Circular Area Source tool: Use this tool to graphically define the location of circular area sources.



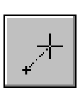
Polygon Area Source tool: Use this tool to graphically define the location of polygon area sources.



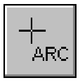
Line Source tool: Use this tool to graphically define the location of line sources.



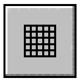
Discrete Cartesian Receptor tool: Use this tool to graphically define the location of discrete Cartesian receptors.



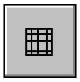
Discrete Polar Receptor tool: Use this tool to graphically define the location of discrete polar receptors.



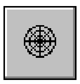
Discrete Cartesian Receptor (ARC) tool: Use this tool to graphically define the location of discrete Cartesian receptors (ARC). This option is only available for AERMOD.



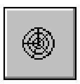
Uniform Cartesian Grid tool: Use this tool to graphically define the location and size of uniform Cartesian grids.



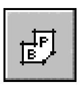
Non-Uniform Cartesian Grid tool: Use this tool to graphically define the location and size of non-uniform Cartesian grids.



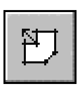
Uniform Polar Grid tool: Use this tool to graphically define the location and size of uniform polar grids.



Non-Uniform Polar Grid tool: Use this tool to graphically define the location and size of non-uniform polar grids.



Plant Boundary tool: Use this tool to graphically define the location of the plant boundary using discrete Cartesian receptors.



Node Editor tool: This tool allows you to change the shape of any polygon you have defined using any one of the following tools: Plant Boundary tool, Polygon Area Source tool, and Line Source tool.

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 domain area.

Scroll Bars

If you use the **Zoom In** button to magnify a section of the drawing area, then you can use the vertical and horizontal **Scroll Bars** to be able to view other sections of the contour plot area.

Status Bar

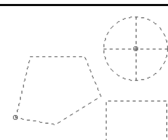
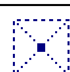
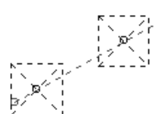
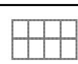
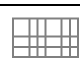
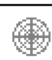
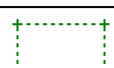
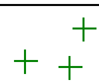
The **Status Bar**, located on the bottom of the Graphical Input window, is used to display information about the object that is currently selected with the Select tool.

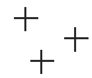
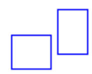
X and Y Coordinates

This section displays the X and Y coordinates of the cursor position in the drawing area. These coordinate values are the real domain coordinates for the site area being modeled.

Drawing Area

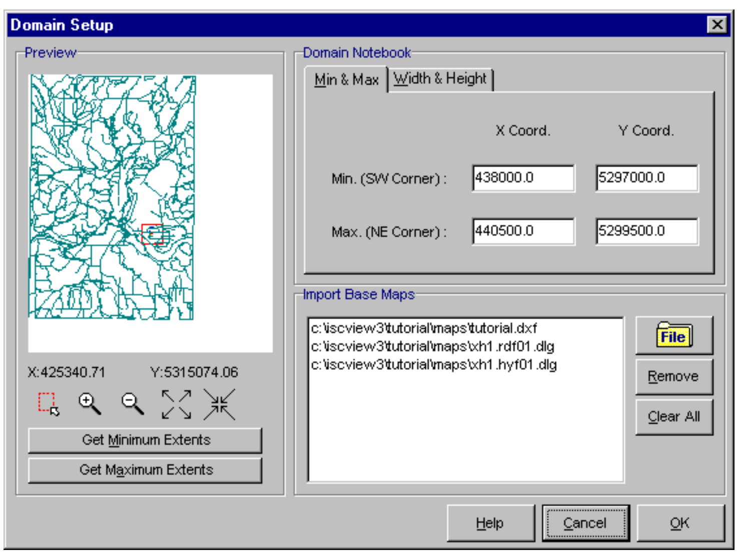
In the **Drawing Area**, a graphical representation of the sources, receptors, grids, buildings, etc. are drawn in the exact locations you defined. Each object (e.g., source, receptor, etc.) will have a different graphical representation. See below the graphical representation for each object:

Object Type	Representation	Line Style	Line Color
Point and Flare Sources		Solid	Red
Area Sources ***(Rectangular, Circular, Polygon)***		Dashed	Red
Volume Sources		Dashed	Navy Blue
Open Pit Sources		Dashed	Fuchsia
Line Sources (Represented by Volume Sources)		Dashed	Navy Blue and Red
Uniform Cartesian Grids		Solid	Dark Gray
Non-Uniform Cartesian Grids		Solid	Dark Gray
Uniform Polar Grids		Solid	Dark Gray
Non-Uniform Polar Grids		Solid	Dark Gray
Plant Boundary Receptors		Dashed	Green
Discrete Cartesian Receptors		Solid	Green

Object Type	Representation	Line Style	Line Color
Discrete Cartesian Receptors (ARC)		Solid	Maroon
Discrete Polar Receptors		Solid and Dashed Line attached to Source	Green
BPIP View Buildings		Solid	Blue
Site Maps	As Specified	As Specified	As Specified

Domain Setup

In the **Domain Setup** dialog box you define the domain extents of your modeling area. If you have not defined any sources or receptors before going to the **Graphical Input** window, then the **Domain Setup** dialog box is automatically displayed, so you can pre-define the extents of your modeling area. If you already specified sources or receptors in text mode then, when you go to the **Graphical Input** window, the domain extents are automatically setup based on the location of the sources and/or receptors you have already specified.



Domain Setup dialog box

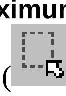
You can define the size of your domain in the following ways:

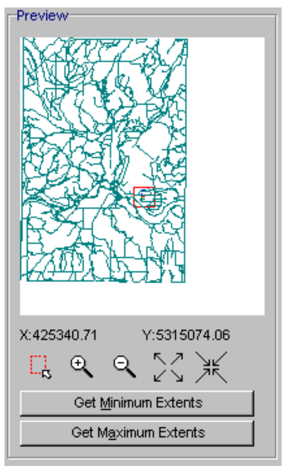
- **Min. & Max. tab :** Specify the X and Y coordinates for two points, the southwest (SW) point (**Min.**) and the northeast (NE) point (**Max.**) of your domain.

- **Width & Height tab:** Specify the X and Y coordinates for the southwest (SW) point (**SW Corner**) and the **X Length** and **Y Length** of your domain.

- **Import Site Base Maps:** You can import site base maps in various file formats (AutoCAD DXF files, USGS DLG files, and USGS LULC files). To import a site base map, click the **File** button and select one or more site base maps. The selected files are displayed in list box and the site base maps are displayed on the Preview area. Once these site maps are loaded into the Preview area, you can do one of the following:

  1. Use the extents of all the imported site base maps as your domain extents. To select this option, press the **Get Maximum Extents** button.

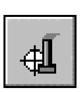
  2. Use the **Choose Extents** tool () located below the Preview area to graphically select your domain extents. Click the **Choose Extents** tool and holding down the left mouse button, draw a rectangle on the preview area to define your domain area. Release the left mouse button. Note that the selected domain area is represented by a red box. The coordinates for the selected domain area are displayed in the **Min. & Max.** and **Width & Height** tabs.



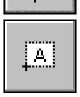
Preview area

Graphical Input Tools

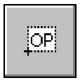
ISC-AERMOD View graphical tools allow you to graphically define the location of your sources, receptors, and plant boundary. The following graphical input tools are available in ISC-AERMOD View:



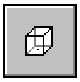
Point Source



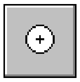
Area Source

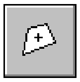


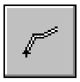
Open Pit Source

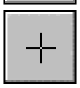

Volume Source

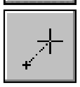

Flare Source

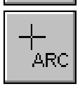

Circular Area Source


Polygon Area Source

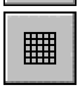

Line Source

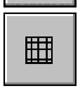

Discrete Cartesian Receptor

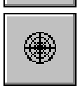

Discrete Polar Receptor

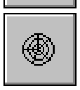


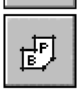
Discrete Cartesian Receptor (ARC)

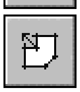

Uniform Cartesian Grid


Non-Uniform Cartesian Grid


Uniform Polar Grid

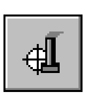

Non-Uniform Polar Grid


Plant Boundary


Node Editor

See the description of each one of these tools in the sections that follow.

Point Source Tool



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

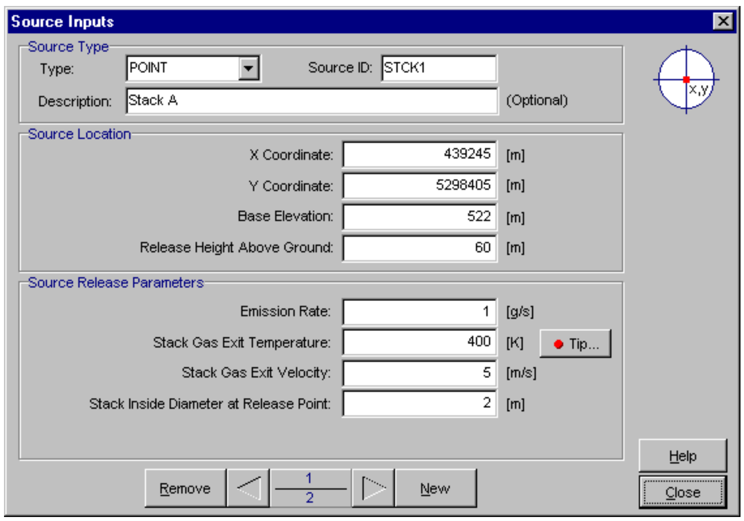
➤ **How to Graphically Define a Point Source:**

Step 1: Press the **Point Source** tool located in the toolbar.

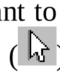
Step 2: Left-click, with the mouse pointer on the drawing area, the desired location. Note that as you move the mouse the current coordinates are displayed on the status bar.

Step 3: The **Source Inputs** dialog box is displayed to allow you to adjust the coordinates, if necessary, and define additional point source information. See *Chapter 4: Source Pathway / SO-Source Inputs window / POINT Source Parameters* for the additional parameters requested for a point source.

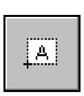
Step 4: When you finish entering all the required information, press the **OK** button. A marker will be placed at the selected location for the point source.



Source Inputs dialog box – Point Source Parameters

Note: If you want to review or modify any information for a particular source, press the **Select** tool (), select the desired source, and click the right mouse button. You can also select **Input | Sources** from the menu to display the **Source Inputs** dialog box.

Area Source Tool



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

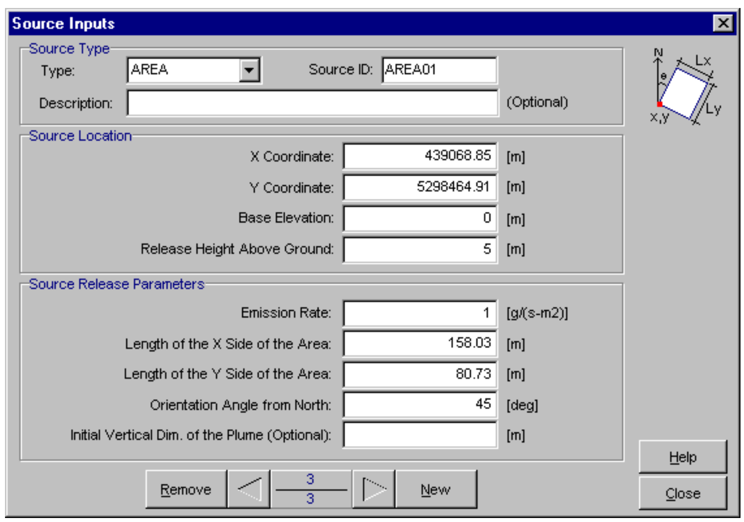
➤ **How to Graphically Define an Area Source:**

Step 1: Press the **Area Source** tool located in the toolbar.

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

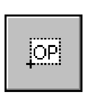
Step 3: The **Source Inputs** dialog box is displayed to allow you to adjust the coordinates, if necessary, and define additional area source information. See *Chapter 4: Source Pathway / SO-Source Inputs window / AREA Source Parameters* for the additional parameters requested for this type of source.

Step 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 and angle.



Source Inputs dialog box – Area Source Parameters

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.

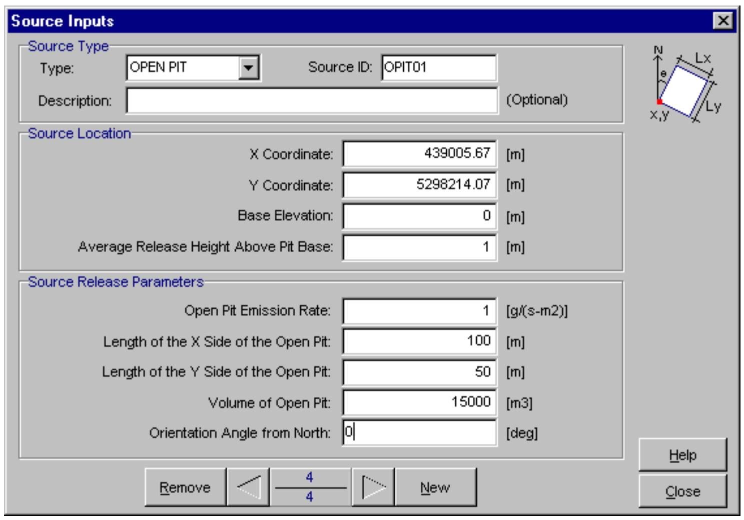
➤ **How to Graphically Define an Open Pit Source:**

Step 1: Press the **Open Pit Source** tool located in the toolbar.

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

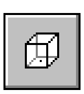
Step 3: The **Source Inputs** dialog box is displayed to allow you to adjust the coordinates, if necessary, and define additional open pit source information. See *Chapter 4: Source Pathway / SO-Source Inputs window / OPEN PIT Source Parameters* for the additional parameters requested for this type of source.

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



Source Inputs dialog box – Open Pit Source Parameters

Volume Source Tool



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

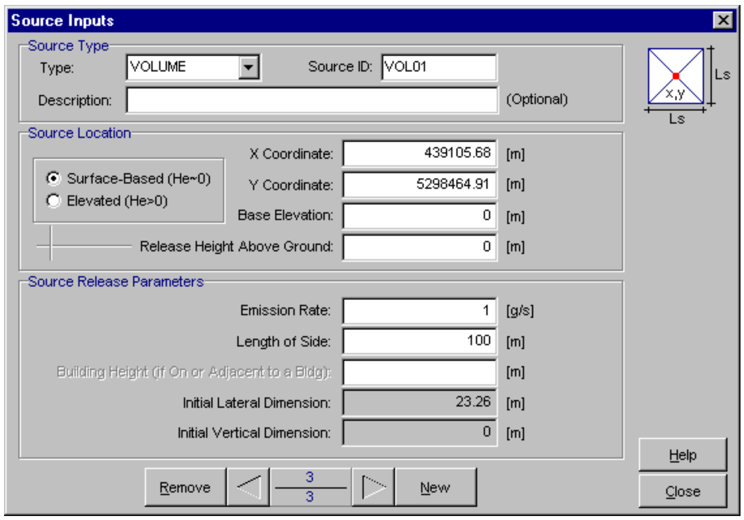
➤ **How to Graphically Define a Volume Source:**

Step 1: Press the **Volume Source** tool located in the toolbar.

Step 2: Left-click, with the mouse pointer on the drawing area, the desired location for one of the corners of the open pit source. Holding down the left mouse button, drag the mouse pointer diagonally until you reach the desired size. Release the left mouse button. Note that the volume source base is always square in shape. The release height is considered to be at the center of the volume source.

Step 3: The **Source Inputs** dialog box is displayed to allow you to adjust the coordinates, if necessary, and define additional volume source information. See *Chapter 4: Source Pathway / SO-Source Inputs window / VOLUME Source Parameters* for the additional parameters requested for this type of source.

Step 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 at the specified location.



Source Inputs dialog box – Volume Source Parameters

Flare Source Tool

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

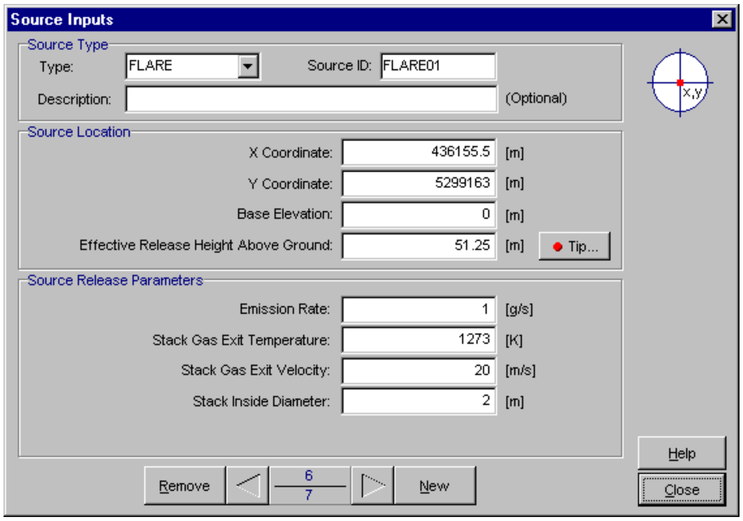
➤ **How to Graphically Define a Flare Source:**

Step 1: Press the **Flare Source** tool located in the toolbar.

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

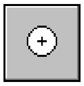
Step 3: The **Source Inputs** dialog box is displayed to allow you to adjust the coordinates, if necessary, and define additional source information. See *Chapter 4: Source Pathway / SO-Source Inputs window / FLARE Source Parameters* for the additional parameters requested for a flare source.

Step 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.



Source Inputs dialog box – Flare Source Parameters

Circular Area Source Tool



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

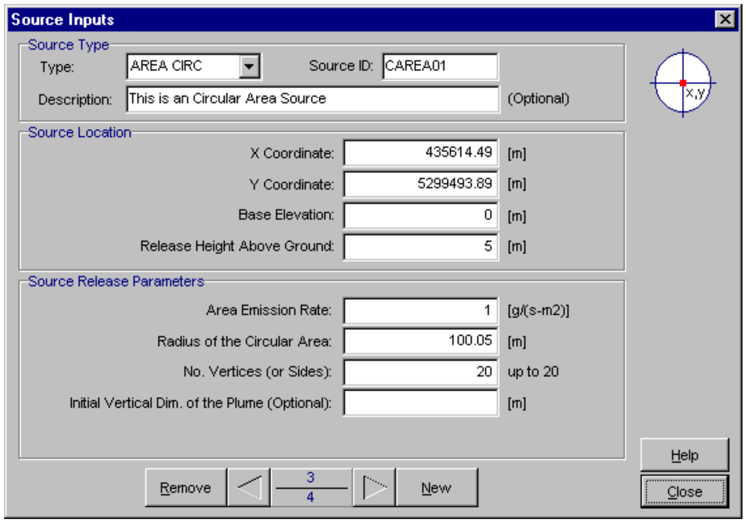
➤ **How to Graphically Define a Circular Area Source:**

Step 1: Press the **Circular Area Source** tool located in the toolbar.

Step 2: Left-click, with the mouse pointer on the drawing area, the desired location for the center of the source. Holding down the left mouse button, drag the mouse pointer in diagonal until you reach the desired size. Release the left mouse button.

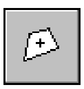
Step 3: The **Source Inputs** dialog box is displayed to allow you to adjust the coordinates, if necessary, and define additional source information. See *Chapter 4: Source Pathway / SO-Source Inputs window / CIRCULAR AREA (AREA CIRC) Source Parameters* for the additional parameters requested for this type of source.

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



Source Inputs dialog box – Circular Area Source Parameters

Polygon Area Source Tool



The **Polygon Area Source** tool allows you to graphically define the location of a polygon area source on the drawing area.

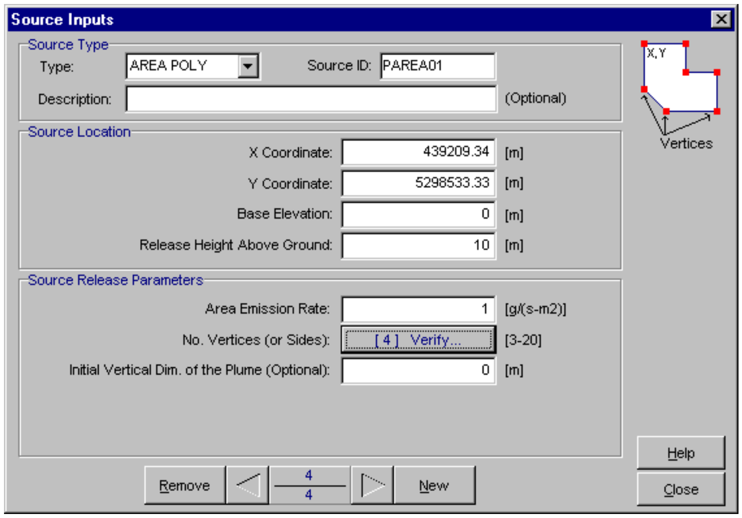
➤ **How to Graphically Define a Polygon Area Source:**

Step 1: Press the **Polygon Area Source** tool located in the toolbar.

Step 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.

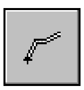
Step 3: The **Source Inputs** dialog box is displayed to allow you to adjust the coordinates, if necessary, and define additional source information. See *Chapter 4: Source Pathway / SO-Source Inputs window / POLYGON AREA (AREA POLY) Source Parameters* for the additional parameters requested for this type of source.

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



Source Inputs dialog box – Polygon Area Source Parameters

Line Source Tool



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

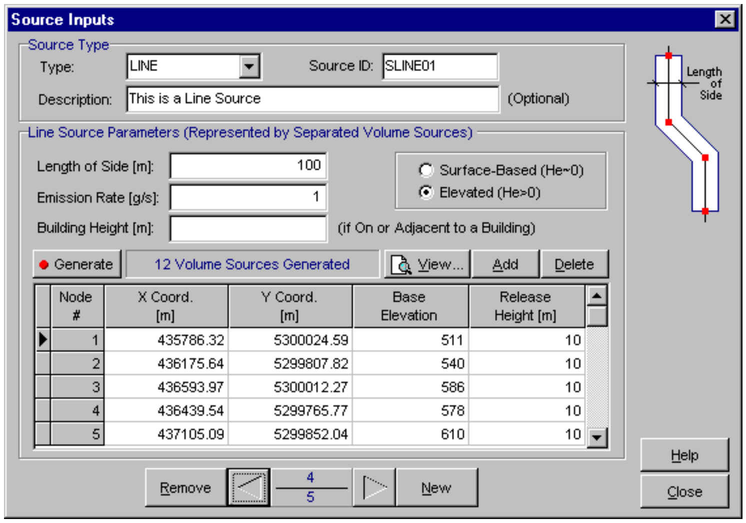
➤ **How to Graphically Define a Line Source:**

Step 1: Press the **Line Source** tool located in the toolbar.

Step 2: Left-click, with the mouse pointer on the drawing area, the location for one of the end nodes of the line source. Release the left mouse button. Drag the mouse pointer to the location of the next (adjacent) node and click the left mouse button. Follow this procedure until you digitize all the nodes. On the last node, click the right mouse button.

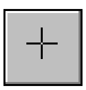
Step 3: The **Source Inputs** dialog box is displayed, allowing you to adjust the coordinates, if necessary, and define additional source information. See *Chapter 4: Source Pathway / SO-Source Inputs window / LINE Source Parameters* for the additional parameters requested for this type of source.

Step 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.



Source Inputs dialog box – Line Source Parameters

Discrete Cartesian Receptor Tool



The **Discrete Cartesian Receptor** tool allows you to graphically define the location of a discrete Cartesian receptor on the drawing area.

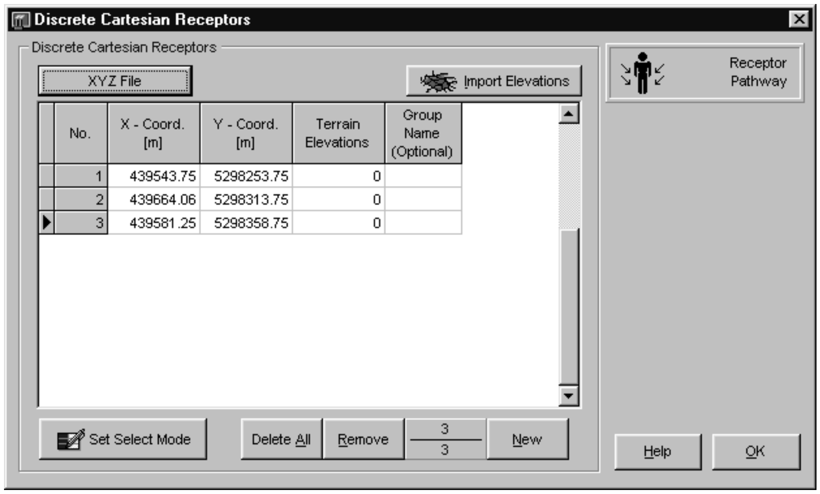
➤ **How to Graphically Define a Discrete Cartesian Receptor:**

Step 1: Select the **Discrete Cartesian Receptor** tool located in the toolbar.

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

Step 3: The **Discrete Cartesian Receptors** dialog box is displayed, allowing you to adjust the coordinates, if necessary, and to define additional receptor information. See *Chapter 5: Receptor Pathway / RE-Discrete Cartesian* for the additional parameters requested for this type of receptor.

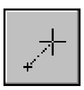
Step 4: When you finish entering all the required information, press the **OK** button. A cross marker will be placed at the selected location for the receptor.



Discrete Cartesian Receptor dialog box

Note: If you want to review or modify any information for a particular receptor, press the **Select** tool ( ), select the desired receptor, and click the right mouse button. You can also select **Input | Receptors** from the menu and select the appropriate submenu option for the desired receptor.

Discrete Polar Receptor Tool



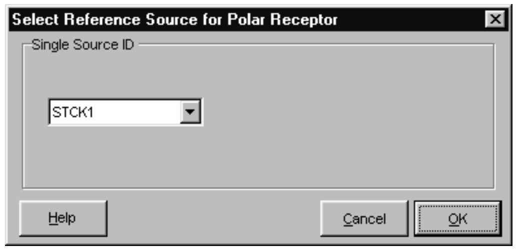
The **Discrete Polar Receptor** tool allows you to graphically define the location of a discrete polar receptor on the drawing area.

➤ **How to Graphically Define a Discrete Polar Receptor:**

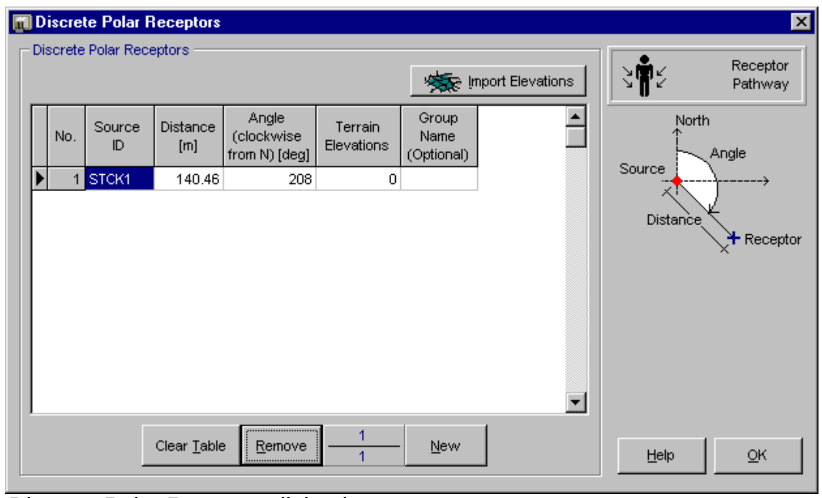
Step 1: Select the **Discrete Polar Receptor** tool located in the toolbar.

Step 2: Left-click, with the mouse pointer on the drawing area, the approximated location for the receptor.

Step 3: A dialog box is displayed requesting you to select the reference source for the polar receptor. By default, the first source from the list will be the one selected as the reference source. However, you can select any source from the drop-down list. Click the OK button.

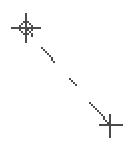


Step 4: The **Discrete Polar Receptors** dialog box is displayed to allow you to adjust the distance and the angle, if necessary, and define additional receptor information. See *Chapter 5: Receptor Pathway / RE-Discrete Polar* for the additional parameters requested for this type of receptor.

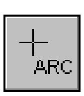


Discrete Polar Receptor dialog box

Step 5: When you finish entering all the required information, press the **OK** 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.



Discrete Cartesian Receptor (ARC) Tool



The **Discrete Cartesian Receptor (ARC)** tool allows you to graphically define the location of a discrete Cartesian ARC receptor on the drawing area.

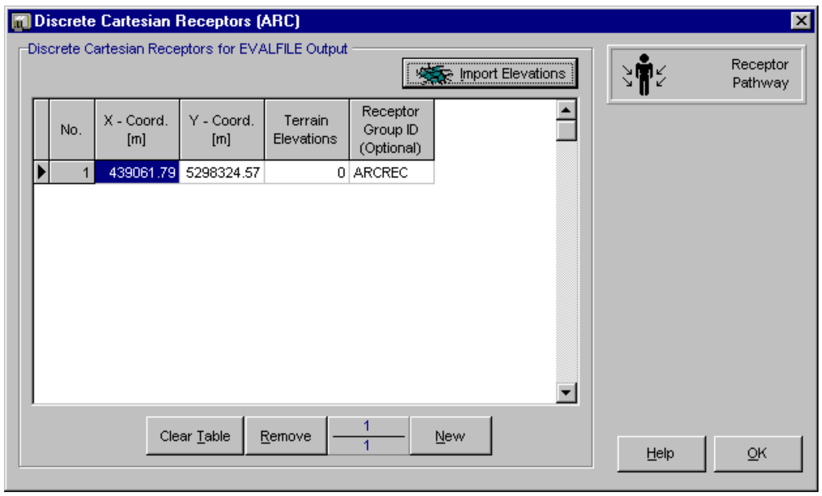
➤ **How to Graphically Define a Discrete Cartesian ARC Receptor:**

Step 1: Select the **Discrete Cartesian Receptor (ARC)** tool located in the toolbar.

Step 2: Left-click, with the mouse pointer on the drawing area, the approximated location for the receptor.

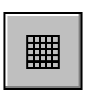
Step 3: The **Discrete Cartesian Receptors (ARC)** dialog box is displayed to allow you to adjust the coordinates, if necessary, and define additional receptor information. See *Chapter 5: Receptor Pathway / RE-Discrete Cartesian (ARC)* for the additional parameters requested for this type of receptor.

Step 4: When you finish entering all the required information, press the **OK** button. A cross marker will be placed at the selected location for the receptor.



Discrete Cartesian Receptors (ARC) dialog box

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.

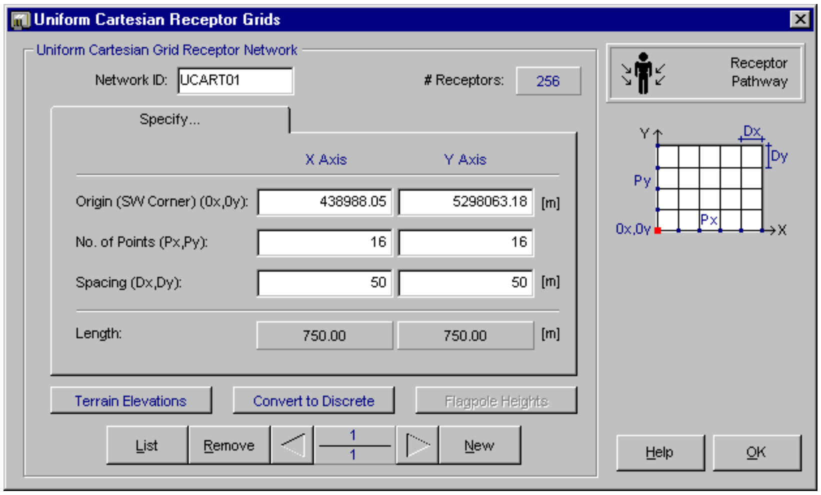
➤ **How to Graphically Define a Uniform Cartesian Receptor Grid:**

Step 1: Select the **Uniform Cartesian Grid** tool located in the toolbar.

Step 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.

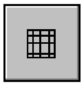
Step 3: The **Uniform Cartesian Receptor Grids** dialog box is displayed to allow you to adjust, if necessary, the origin coordinates, number of X and Y points, and spacing. See *Chapter 5: Receptor Pathway / RE-Uniform Cartesian Grid* *window* for the additional parameters requested for this type of network receptor grid.

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



Uniform Cartesian Receptor Grids dialog box

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.

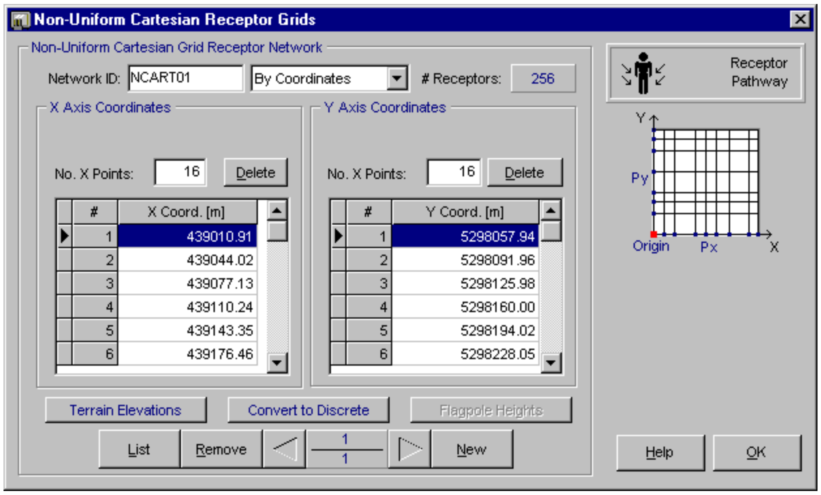
➤ How to Graphically Define a Non-Uniform Cartesian Receptor Grid:

Step 1: Select the **Non-Uniform Cartesian Grid** tool located in the toolbar.

Step 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.

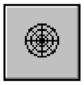
Step 3: The **Non-Uniform Cartesian Receptor Grids** dialog box is displayed to allow you to adjust, if necessary, the coordinates, number of points in the X- and Y-axis, and the spacing. *See Chapter 5: Receptor Pathway / RE-Non-Uniform Cartesian Grid window* for the additional parameters requested for this type of network receptor grid.

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



Non-Uniform Cartesian Receptor Grids dialog box

Uniform Polar Grid Tool



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

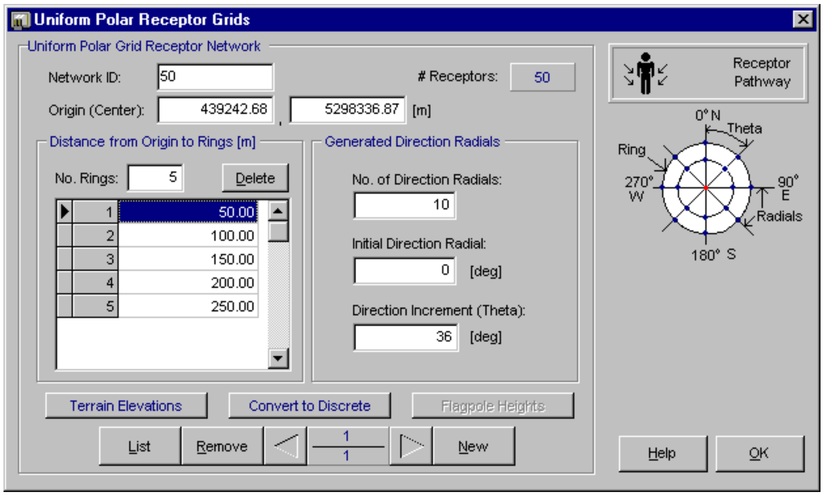
➤ How to Graphically Define a Uniform Polar Receptor Grid:

Step 1: Select the **Uniform Polar Grid** tool located in the toolbar.

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

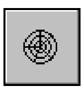
Step 3: The **Uniform Polar Receptor Grids** dialog box is displayed to allow you to adjust, if necessary, the distance from origin (center) to rings and direction radials. See *Chapter 5: Receptor Pathway / RE-Uniform Polar Grid window* for the additional parameters requested for this type of network receptor grid.

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



Uniform Polar Receptor Grids dialog box

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.

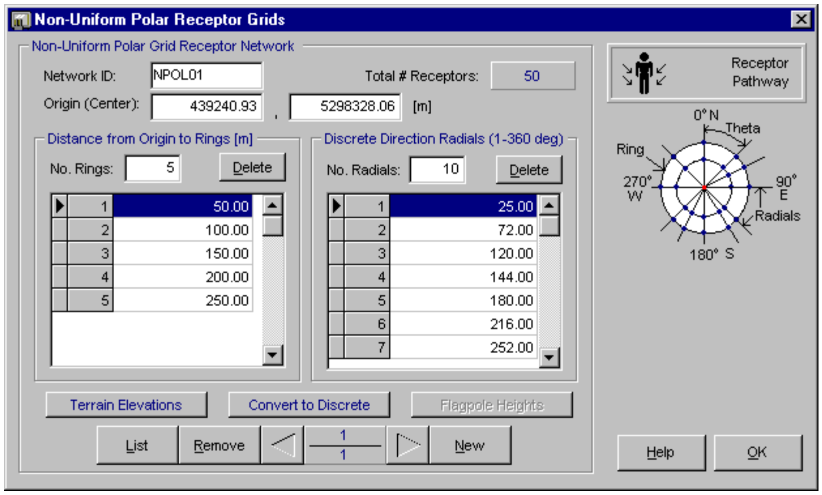
➤ **How to Graphically Define a Non-Uniform Polar Receptor Grid:**

Step 1: Select the **Non-Uniform Polar Grid** tool located in the toolbar.

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

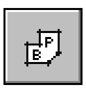
Step 3: The **Non-Uniform Polar Receptor Grids** dialog box is displayed to allow you to adjust, if necessary, the distance from origin (center) to rings and direction radials. See *Chapter 5: Receptor Pathway / RE-Non-Uniform Polar Grid window* for the additional parameters requested for this type of network receptor grid.

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



Non-Uniform Polar Receptor Grids dialog box

Plant Boundary Tool



The **Plant Boundary** tool allows you to graphically define the location of your plant boundary (fenceline) on the drawing area.

➤ **How to Graphically Define your Plant Boundary:**

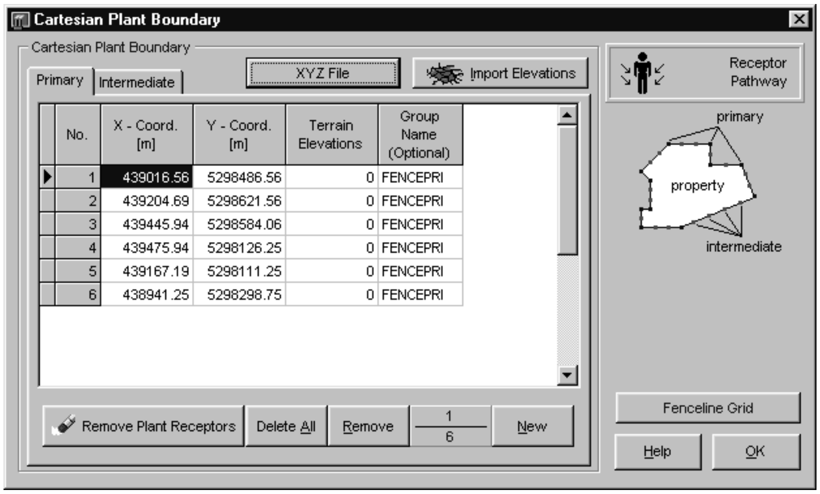
Step 1: Select the **Plant Boundary** tool located in the toolbar.

Step 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.

Step 3: The **Cartesian Plant Boundary** dialog box is displayed to allow you to adjust, if necessary, the coordinates, for the corners of the plant boundary polygon. See *Chapter 5: Receptor Pathway / RE-Cartesian Plant Boundary window* for the additional parameters requested for this type of receptor.

Primary Receptors

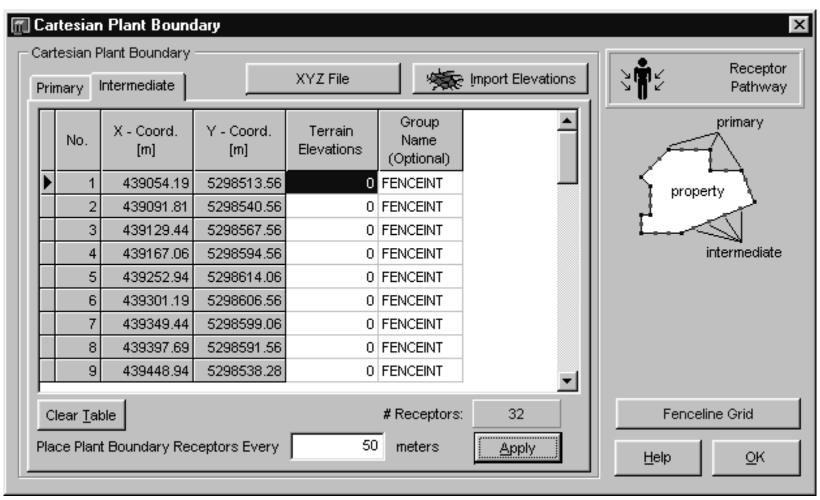
The **Primary** tab displays the X and Y coordinates for all the corners of the polygon that was defined as the plant boundary. At each corner, ISC-AERMOD View places a discrete receptor. You can adjust the coordinates of each polygon corner as desired. The terrain elevations are zero by default. If you have terrain elevations and flagpole heights you should define these values for each corner (discrete receptor).



Cartesian Plant Boundary dialog box - Primary Receptors

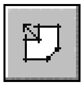
Intermediate Receptors

In the **Intermediate** tab, you can add additional receptor points along the plant boundary polygon. You need to specify the distance between receptors and press the **Apply** button. Intermediate 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. For each additional receptor, you need to specify terrain elevations and flagpole heights if these options apply to your case.



Cartesian Plant Boundary dialog box – Intermediate Receptors

Node Editor Tool



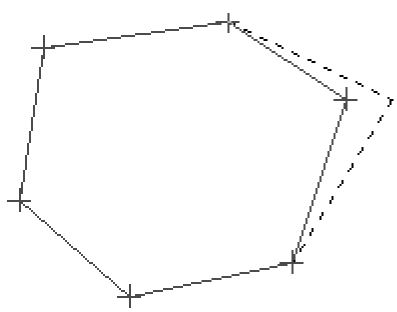
The **Node Editor** tool allows you to graphically re-define the location of one or more nodes (corners) of an object (source or receptor) that was defined using polylines. You can use the Node Editor tool with the following objects:

- Polygon Area Source
- Line Source
- Cartesian Plant Boundary

➤ **How to Use the Node Editor Tool:**

Step 1: Select the **Node Editor** tool located in the toolbar.

Step 2: With the mouse pointer, select the object to be edited. Note that the nodes are marked with a black bounding box. With the mouse pointer, click on the bounding box of the node you want to adjust and, pressing down the left mouse button, drag the box to the new location. Release the left mouse button. Repeat this procedure for each node you want to adjust.



Importing DXF Site Maps

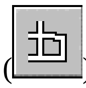
ISC-AERMOD View can import one or more digitized base maps in AutoCAD DXF file format. The DXF (Drawing Exchange Format) file is a standard format for exchanging data between CAD and GIS systems. For instance, you can use AutoCAD software to convert your site map into the DXF format.

➤ **How to Import DXF Site Base Maps:**

Step 1: You have three ways of importing DXF site base maps:

1. From the **Domain Setup** dialog box click on the **File** button.

2. Select **File | Import | Base Map | DXF** menu option from the Graphical Input window.

3. Click on the **Import Site Map** tool () located in the **Drawtool** toolbar.

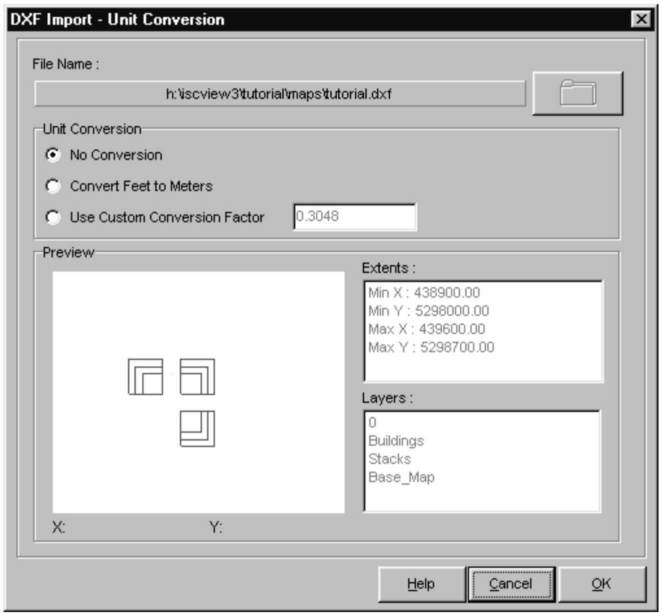
Step 2. The **Import DXF Base Map** dialog box is displayed. Enter the name and path for the site map file (.DXF) and click the **OK** button. The **DXF Import - Unit Conversion** dialog box is displayed.

Step 3. From the **DXF Import - Unit Conversion** dialog box, you can select a conversion unit, in case your DXF site map was created in a unit other than meters. See **DXF Import - Unit Conversion** section below for more information. Press the **OK** button to accept the DXF file and unit conversion. The **Domain Setup** dialog box is displayed and the imported DXF site base map can be seen in the **Preview** area in context with your other maps and/or modeling domain. Make the necessary adjustments to your domain area and click the **OK** button. The site map you selected will be placed as an overlay on the drawing area.

Note: It is important to note that the extents of the imported DXF site map should be within the site domain area defined on the **Domain Setup** dialog. ISC-AERMOD View will place your site map on the same coordinates defined in your DXF file. If the extents of the site map are outside the defined site domain, then this site map will not be displayed on the drawing area unless the domain extents are changed.

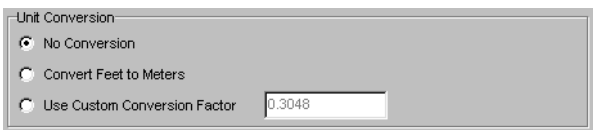
DXF Import - Unit Conversion

The **DXF Import - Unit Conversion** dialog box is displayed after you select a DXF file to be imported into your ISC-AERMOD View project. This dialog box provides information about the DXF file you have selected for importing such as the extents, the layers, and a preview of the selected DXF site map.



DXF Import – Unit Conversion dialog box

In the **DXF Import - Unit Conversion** dialog box you also have the option of converting the existing units of the DXF site map to meters. This is necessary, since ISC-AERMOD View only accepts dimensions in meters. For the unit conversion, three options are available:



- **No Conversion:** This is the default option and is used if the DXF file to be imported was drawn in meters.

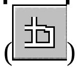
- **Convert Feet to Meters:** Use this option if the units in your DXF site map are in feet. In this case, ISC-AERMOD View converts your site map units automatically to meters.

- **Use Custom Conversion Factor:** Use this option if the units in your DXF site map are not in meters or feet. In this case, ISC-AERMOD View converts your site map units automatically to meters using the unit conversion factor you have specified in the text box.

Importing Bitmap Site Maps

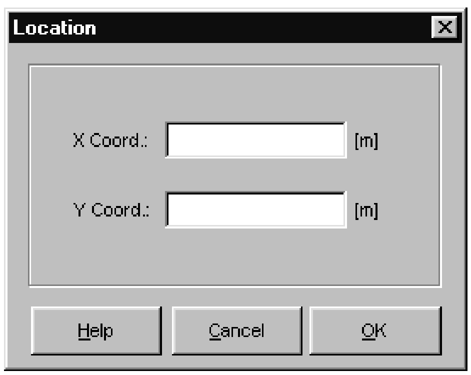
ISC-AERMOD View can import Bitmap images to be used as base maps. Bitmaps do not contain information on the site coordinates and extents of the image However, ISC-AERMOD View will prompt you for this information.

➤ How to Import Bitmap to be Used as Base Maps:

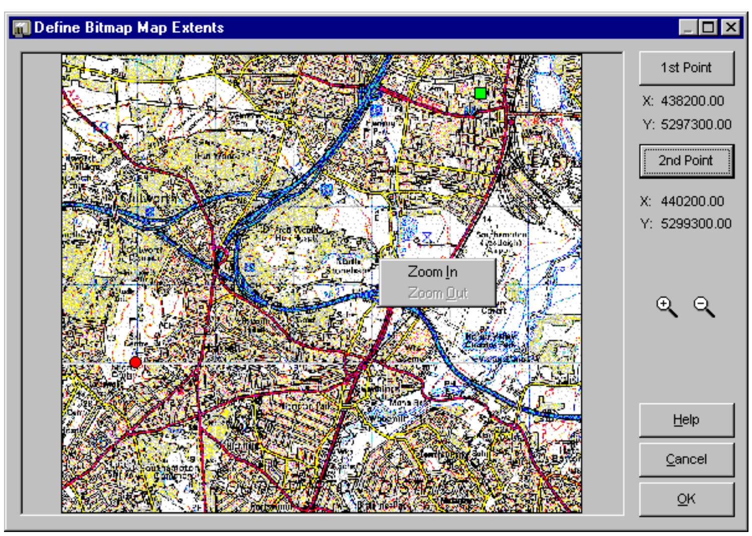
Step 1: Select **File | Import | Base Map | BMP** from the menu, or click on the **Import Site Map** tool () and choose **BMP Map**. The **Import BMP Base Map** dialog box is displayed.

Step 2: Enter the name and path for the Bitmap base map file (.BMP) and click the **OK** button. The **Define Bitmap Map Extents** dialog box is displayed containing the Bitmap image you have imported.

Step 3: Click on the **1st Point** button and then click on the image where the location of your first point is. The **Location** dialog box appears requesting you to input the **X** and **Y** coordinates for the first point.

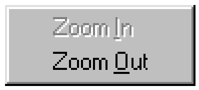


Step 4: Repeat **Step 3** for the second point, by clicking on the **2nd Point** button. Note that a red dot defines the location of your first point and a small green square defines the location of your second point. The specified coordinates for these two points are displayed on the right hand side of the dialog box below the **1st Point** button and the **2nd Point** button respectively.



Define Bitmap Map Extents dialog box

Step 5: Right click anywhere on the image to display the floating menu. 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 In** and **Zoom Out** tools ().



Step 6: After defining the two points, click on the **OK** button. ISC-AERMOD View will then display the imported Bitmap image as an overlay on the drawing area using the coordinates of the two points you have specified.

Importing DLG Site 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.

Digital Line Graphs (DLG) from selected quadrangles currently available from Earth Science Information Centers (ESIC) are:

- 1:24,000-scale
- 1:62,500-scale
- 1:63,360-scale
- 1:100,000-scale
- 1:2,000,000-scale (ISC-AERMOD View currently does not support this scale)

The DLG data files contain selected base categories of cartographic data in digital form. The following categories are included in current large-scale DLG files:

- **Boundaries:** This category consists of political boundaries (i.e. States, counties, cities, and other municipalities) and administrative boundaries (i.e. National and State forests).

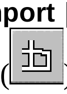
- **Hydrography:** This category of data is currently being collected as combined hydrography consisting of all flowing water, standing water, and wetlands.

- **Public Land Survey System (PLSS):** This category of data describes the rectangular system of land surveys that is administered by the U.S. Bureau of Land Management.

- **Transportation:** This category of data includes major transportation systems: (1) roads and trails, (2) railroads, (3) pipelines, transmission lines, and miscellaneous transportation features.

- **Other Significant Manmade Structures:** This category of data includes miscellaneous cultural features not included in the other major data categories.

➤ **How to Import USGS DLG Base Maps:**

Step 1: Select **File | Import | Base Map | DLG...** menu option, or click on the **Import Site Map** tool () and choose **DLG Map**. The **Import USGS DLG Base Map** dialog box is displayed.

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

Step 3: The **Domain Setup** dialog box is displayed and the imported site base map can be seen in the Preview area in context with your other maps and/or modeling domain. Make the necessary adjustments to your domain area and click the **OK** button. The site map you selected will be placed as an overlay on the drawing area.

Importing LULC Site Maps

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

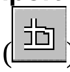
The 1:250,000-scale mapping format is usually a quadrangle unit of 1º of latitude x 2º of longitude. The 1:100,000-scale mapping format has been established as a 30’ x 60’ quadrangle, which is normally a quarter of a 1:250,000-scale quadrangle. Both series are based on the Universal Transverse Mercator (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.

Table 9-1. Color Codes and Land Use Grouping used by ISC-AERMOD View

Color	Level 1	Level 2
Pink	Urban or Built Up Land	12 Commercial and Services 14 Transportation, Communications and Utilities 15 Industrial and Commercial Complexes 16 Mixed Urban or Built-up Land 17 Other Urban or Built-up Land
Light Purple	Residential	11 Residential
Red	Industrial	13 Industrial
Yellow	Agricultural Land	21 Cropland and Pasture 22 Orchards, Groves, Vineyards, Nurseries, and Ornamental Horticultural Areas 23 Confined Feeding Operations 24 Other Agricultural Land
Light Green	Rangeland	31 Herbaceous Rangeland 32 Shrub and Brush Rangeland 33 Mixed Rangeland
Forest Green	Forest Land	41 Deciduous Forest Land 42 Evergreen Forest Land 43 Mixed Forest Land
Light Blue	Water	51 Streams and Canals 52 Lakes 53 Reservoirs 54 Bays and Estuaries
Olive	Wetland	61 Forested Wetland 62 Non-forested Wetland
Brown	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

➤ **How to Import LULC Base Maps:**

Step 1: Select **File | Import | Base Map | LULC...** menu option, or click on the **Import Site Map** tool () and choose **LULC Map**. The **Import USGS LULC Base Map** dialog box is displayed.

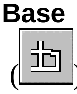
Step 2: Enter the name and path for the LULC base map file (.LUC) and click the **OK** button.

Step 3: The Domain Setup dialog box is displayed and the imported site base map can be seen in the **Preview** area in context with your other maps and/or modeling domain. Make the necessary adjustments to your domain area and click the **OK** button. The site map you selected will be placed as an overlay on the drawing area.

Importing ArcView Shapefiles

ArcView shapefiles store the geometric location and attribute information of geographic features, along with attribute information describing what these features represent.

➤ **How to Import ArcView Shapefiles**:

Step 1: Select **File | Import | Base Map | Shapefile** from the menu, or click on the **Import Site Map** tool () and choose **Shapefiles**. The **Import ArcView Shapefile** dialog box is displayed.

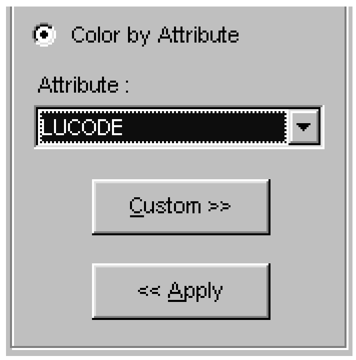
Step 2: In the **Import ArcView Shapefile** dialog, the following information is requested:

- **File Name:** Click the **File** button and specify the shapefile (.SHP) you want to import.

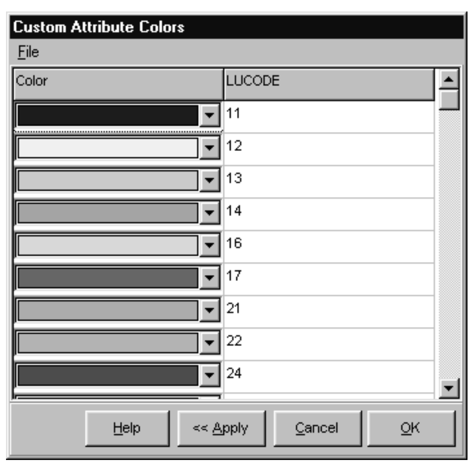
- **Color Scheme:** You have two options, **Fixed Color** and **Color by Attribute**. See description of each one of these options below.

  1. **Fixed Color:** This option is the default. If this option is selected, then your shapefile will be imported in the fixed color you have selected in the drop down list box.

2. **Color by Attribute:** This option allows you to select one of the shapefile attributes as the color scheme. For example, suppose you have setup an attribute called LUCODE for your shapefile, which contains land use features such as residential areas, industrial areas, water, agricultural land, etc. following the Land Use and Land Cover classification system. If you select this attribute from the drop down list box, then ISC-AERMOD View will assign random colors 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 this button to display the **Custom Attribute Colors** dialog box from where you can define colors for each attribute class.

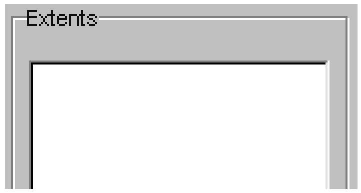


Custom Attribute Colors dialog box

Press this button to apply the selected color scheme to the imported shapefile.

- **Preview:** The **Preview** area displays the shapefile to be imported in the selected color scheme.

- **Extents:** This are shows the extents of the shapefile being displayed on the **Preview** area.



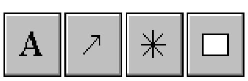
Step 3: Click the OK button when you finish specifying the coordinate unit and color scheme for the shapefile. The shapefile will be imported into the drawing area only if the extents of the shapefile are within the extents of your ISC-AERMOD View project.

Export Options

ISC-AERMOD View offers the following exporting option. You have access to this exporting option by selecting from the menu:

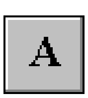
- **File | Export | Base Map | BMP… :** This option exports the contents of the drawing area (sources, receptors, buildings, grids, plant boundary, annotations, and maps) to a Windows Bitmap file format (*.bmp).

Annotation Tools



From the toolbar you have access to some annotation tools. These tools will be described in the subsections that follow.

Text Annotation Tool



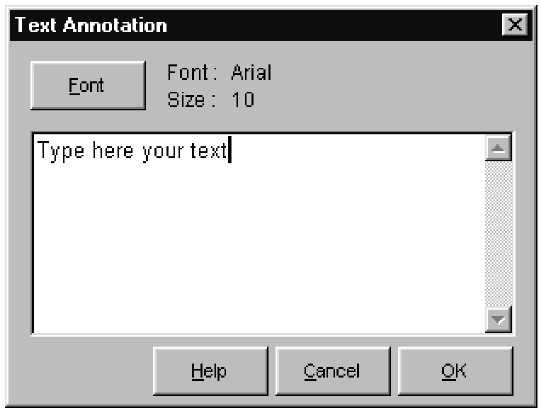
The **Text Annotation** tool is used to write text on the drawing area. This allows you to add more information to your printout outputs.

➤ **How to Write Text on the Drawing Area:**

Step 1: Press the **Text Annotation** tool located in the toolbar.

Step 2: Click the left mouse button on the drawing area, where you want the beginning of the text to be placed.

Step 3: The **Text Annotation** dialog box is displayed. Type the text in the list box and press the **Font** button if you want to select a different font, style, size, color, etc.

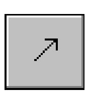


Text Annotation dialog box

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

Step 5: To move the text to a new location, use the **Select** tool (). To delete the text from the drawing area, use the **Delete** tool ().

Arrow Annotation Tool



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

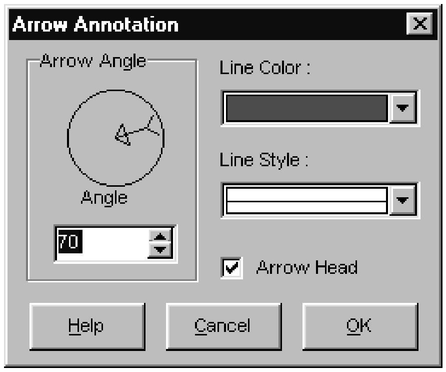
➤ **How to Draw Arrows and Lines on the Drawing Area:**

Step 1: Press the **Arrow Annotation** tool located in the toolbar.

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

Step 3: Change the angle, the line color, or the line style, and select if you want the arrow head to be displayed. Press the **OK** button.

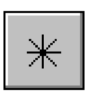
Step 4: The arrow or line is placed on the drawing area in the location and direction you have previously selected.



Arrow Annotation dialog box

Step 5: To move the arrow to a new location, use the **Select** tool (). To delete the arrow from the drawing area, use the **Delete** tool ().

Marker Annotation Tool

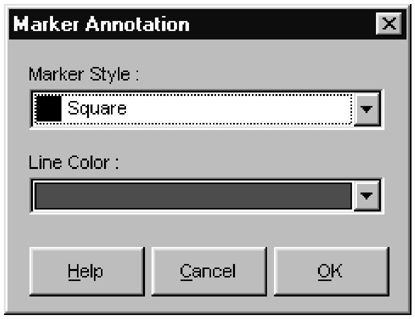
 The **Marker Annotation** tool is used to draw markers on the drawing area.

➤ **How to Draw Markers on the Drawing Area:**

Step 1: Press the **Marker Annotation** tool located in the toolbar.

Step 2: Click the left mouse button on the drawing area, where you want the marker to be placed.

Step 3: The **Marker Annotation** dialog box is displayed. Select the marker style and color and press the **OK** button.

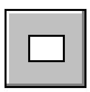


Marker Annotation dialog box

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

Step 5: To move the marker to a new location, use the **Select** tool (). To delete the marker from the drawing area, use the **Delete** tool ().

Rectangle Annotation Tool

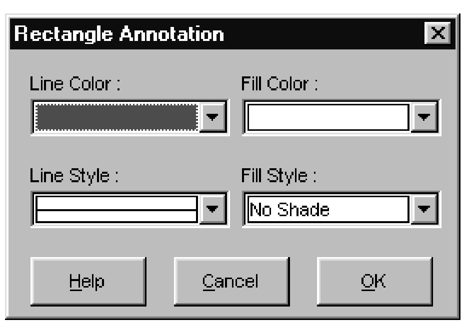


The **Rectangle Annotation** tool is used to draw rectangles on the drawing area.

➤ **How to Draw a Rectangle on the Drawing Area:**

Step 1: Press the **Rectangle Annotation** tool located in the toolbar.

Step 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. Release the left mouse button. The **Rectangle Annotation** dialog box is displayed.



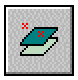
Rectangle Annotation dialog box

Step 3: Change the color and style for the line and fill of the rectangle and press the **OK** button.

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

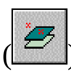
Step 5: To move the rectangle to a new location, use the **Select** tool (). To delete the rectangle from the drawing area, use the **Delete** tool ().

Overlay Control



From time to time, during your project, you may wish to hide some objects being displayed on the drawing area, such as hiding a receptor grid, or a DXF site map, so you can view or print only the desired objects. From the **Overlay Control** dialog box, you can specify which objects should be displayed in the drawing area.

➤ **How to Turn On or Off the Display of Objects:**

Step 1: Click on the **Overlay Control** tool () or select **View | Overlay Control** from the menu. The **Overlay Control** dialog box is displayed.

Step 2: All the objects defined in your ISC-AERMOD View project are listed in this dialog box. The list located on the left hand side, **Show These Overlays**, contains all the objects being displayed in the drawing area. The list on the right hand side, **Don't Show These**, contains the objects that are not being displayed. You can easily move these objects from one list to the other using the buttons located between the two lists.

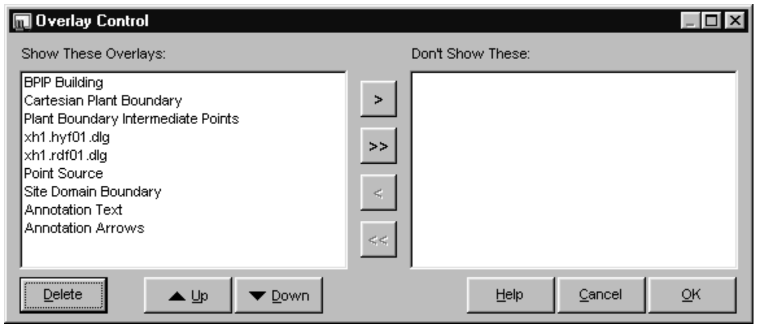
> and < .... Moves the selected overlays to the appropriate list.

>> and << .... Moves all overlays from one list to the other.

Delete .... Deletes the selected map from the project. This button can only delete map objects.

▲ Up .... Moves the currently selected overlay up in the list. This list controls the order in which the layers are drawn on screen. You can also click and drag the overlay to the desired list position.

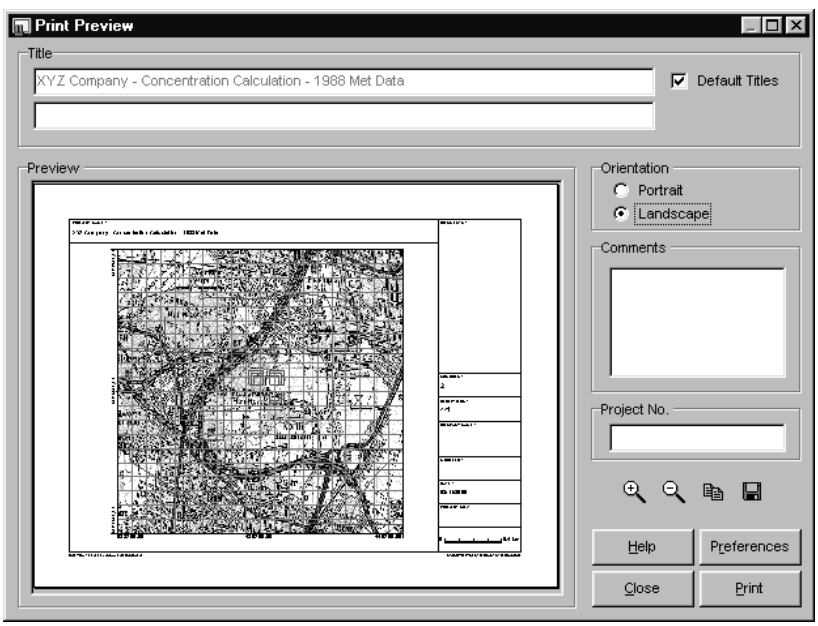
▼ Down .... Moves the currently selected overlay down in the list. This list controls the order in which the layers are drawn on screen. You can also click and drag the overlay to the desired list position.



Overlay Control dialog box

Printing Options

Before you can print the contents of the drawing area, ISC-AERMOD View displays the **Print Preview** dialog box. Here you can select printing options and preview how your printouts will look. ISC-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.

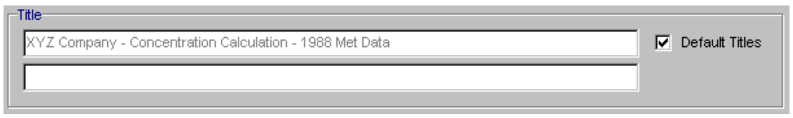


Print Preview dialog box

Print Preview

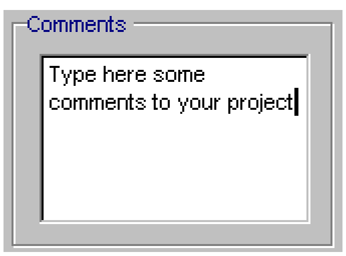
The **Print Preview** dialog box is displayed every time you press the **Print** tool located on the toolbar or if you select **File | Print…** from the menu. The **Print Preview** dialog box contains the following options:

- **Title:** ISC-AERMOD View places the first and second title you have specified in the Control Pathway as the default titles. However, you have the option of specifying different titles. To specify different titles, uncheck the **Default Titles** box and type in the titles.

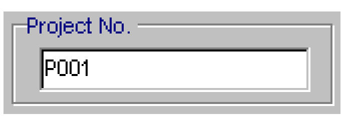


- **Orientation:** This is the orientation for your printout. Note that the preview area shows a preview of your results in both orientations: portrait and landscape.

- **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 the number or code for your project.



- **Buttons:** At the lower right corner of the dialog box a series of buttons are available. See the function of each one 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 Bitmap: Select this tool if you want to copy the image to the clipboard as a Bitmap. You can then paste into any Windows application that supports pasting of Bitmaps from the clipboard.

Save to File: Select this option if you want to save the printout to a file. The printout can be saved as an Enhanced Windows Metafile (*emf) or as a Bitmap (*.bmp).

.... Displays the help contents for the options contained in the **Print Preview** dialog box.

.... Displays the **Preferences** dialog box where you can specify printing and labeling options for your printout outputs.

.... Displays the **Print** dialog box where you can specify printing options such as printer, number of copies, paper size, orientation, etc.

.... Closes the **Print Preview** dialog box.

Print Preferences

In the **Print Preferences** dialog box you define default options for printing the contents of the drawing area. Note that the options you have selected in this dialog box will be used as the default for your current project or any other project. You have access to the **Print Preferences** dialog box by selecting **File | Print Preferences** from the menu or clicking on the **Preferences** button located in the **Print Preview** dialog box.

The **Print Preferences** dialog box contains two tabs, the **Printing Options** tab and the **Labeling Options** tab. See below the contents of each one of these tabs.

Printing Options Tab

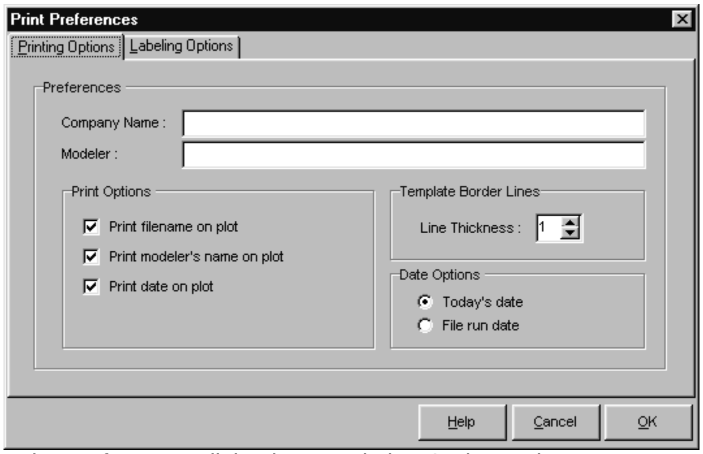
The **Printing Options** tab contains the following options:

- **Company Name:** Here you specify the name of your company to be printed on the space available for the company name on the printout template. If this field is left blank then no company name will be printed.

- **Modeler:** Here you type the name of the modeler to be printed in the space available on the printout template. If this field is left blank then no modeler name will be printed. You can also eliminate the modeler's name by leaving the **Print modeler's name on plot** check box unchecked.

- **Print Options:** You can select one or more of the following print options:

  1. **Print filename on plot:** Check this box if you want the ISC-AERMOD View project file name to be printed on the lower right side of the page.
  2. **Print modeler's name on plot:** Check this box if you want the name of the modeler to be printed on the printout template.
  3. **Print date on plot:** Check this box to print the date on the printout template.



Print Preferences dialog box – Printing Options tab

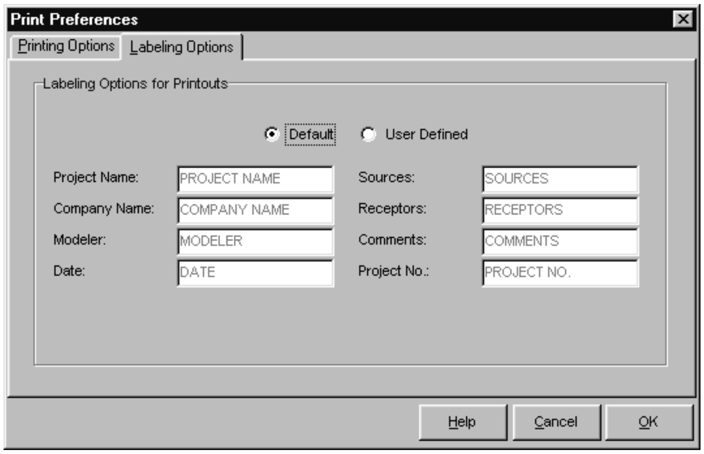
- **Line Thickness:** This is the thickness of all the border lines for the printout templates.

- **Date Options:** The following options are available:

  1. **Today's Date:** This is the current date. Make sure your computer has the correct date setup.
  2. **File Run Date:** This is the date of the ISC-AERMOD View project file.

Labeling Options Tab

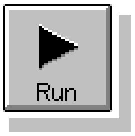
In the **Labeling Options** tab you can specify the labeling for the printout templates. This option is very helpful for users that want the templates labeling in a language other then English. By default, ISC-AERMOD View uses the labeling as shown in the fields when the **Default** option is selected. To be able to specify your own label, select the **User Defined** option and specify the appropriate label in each field.



Print Preferences dialog box – Labeling Options tab

CHAPTER 10

Running the U.S. EPA Model



This chapter explains how to verify the status of your project, select the U.S. EPA model to run, run the model, and view your results. For every ISC-AERMOD View project, you can preview and print reports containing all the options you specified in your project. These reporting options are also described in this chapter.

Contents

Selecting the U.S. EPA Model

ISC-AERMOD View allows you to specify which U.S. EPA executable model should be used to run your project. The latest releases of the U.S. EPA models (ISCST3, AERMOD, and ISC-PRIME) are supplied with the ISC-AERMOD View interface. You can also download the latest models from the U.S. EPA web site:

www.epa.gov/scram001

or from our web site:

www.lakes-environmental.com

With the ISC-AERMOD View interface, you enter data into your project once and you can run any one or all of the following U.S. EPA models: ISCST3, AERMOD, and ISC-

PRIME. The original U.S. EPA model for AERMOD and ISC-PRIME have some limitations on the number of sources, number of receptors, number of source groups, etc. Version 99155 of the U.S. EPA ISCST3 model, however 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.

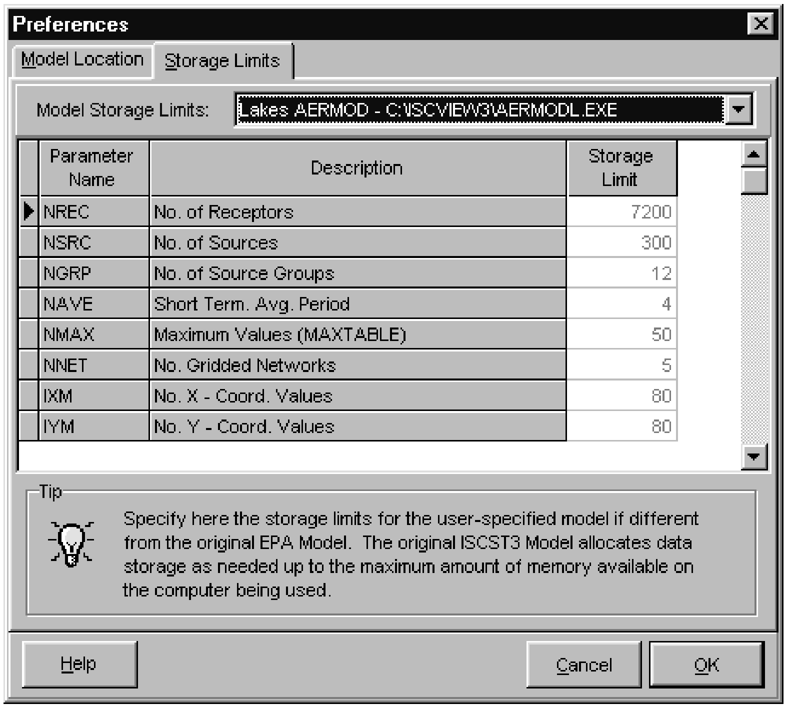
For example, the following limits apply for the total number of receptors allowed in each model:

U.S. EPA Model	**Storage Limit for Total No. of Receptors (NREC)**
ISCST3 (dated 99155)	Allocates data storage as needed up to the maximum amount of memory available on a particular computer.
AERMOD (dated 99211)	1500
ISC-PRIME (dated 98069)	1200

Due to the restrictions of the original AERMOD and ISC-PRIME models, a project containing more than 1200 receptors cannot be run using the original U.S. EPA ISC-PRIME executable model (ISCP3.exe). A project with more than 1500 receptors can only be run using the ISCST3 executable model (ISCST3.exe). For the above reasons, ISC-AERMOD View allows you to specify an alternative model executable for ISCST3, AERMOD and ISC-PRIME.

The U.S. EPA AERMOD and ISC-PRIME models make use of a static storage allocation design. The FORTRAN code for these models can be modified and recompiled to accept higher storage limits for many parameters.

Lakes Environmental Software recompiled AERMOD.EXE and ISC3P.EXE to accept higher storage limits. The new executables were renamed to AERMODL.EXE and PRIMEL.EXE, respectively. The original U.S. EPA models can be found in your installation directory. The storage limit for the recompiled codes (AERMODL.EXE and PRIMEL.EXE) are defined in the **Storage Limits** tab of the **Preferences** dialog box.



Preferences dialog box

If you want to use an U.S. EPA model other than the original than you need to specify the location of the executable and the storage limits.

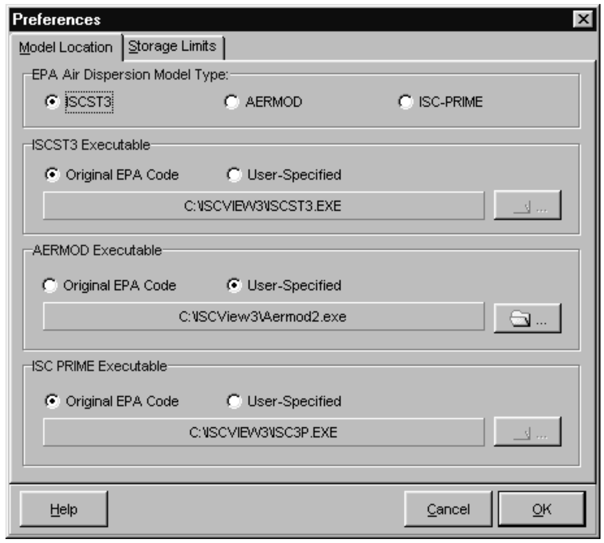
If you want to use an U.S. EPA model other than the original than you need to specify the location of the executable and the storage limits.

➤ **How to Specify the Model Executable:**

Step 1: Select **File | Preferences** from the menu. The **Preferences** dialog box is displayed. By default, the original EPA executable codes are specified for each model type.

Step 2: If you want to select an alternative model executable, then you need to select the **User-Specified** option for that particular model type.

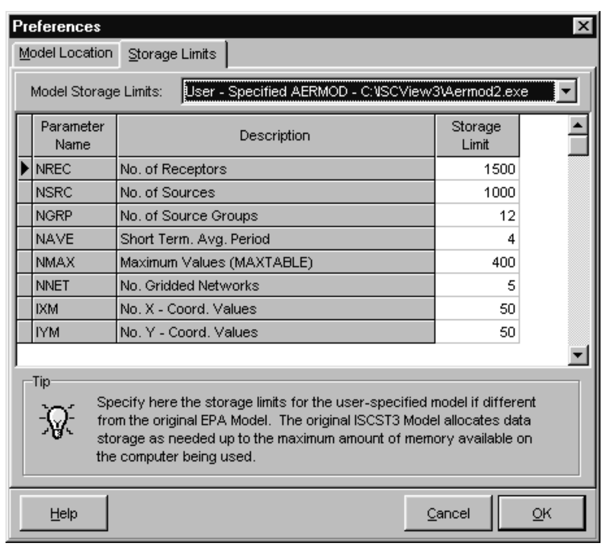
Step 3: Press the **Specify File** button () and specify the location and file name of the executable model.



Preference dialog box – Model Location tab

Step 4: Click on the **Storage Limits** tab. Select from the drop-down list box the reference to your alternative model executable.

Step 5: Specify the new storage limit 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 the user changes these limits.



Preference dialog box – Storage Limits tab

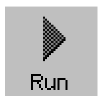
Note: All ISC-AERMOD View data validation is based on the storage limit values. Therefore, it is important that the correct limits be specified for the user-specified model executables.

Checking the Project Status

The **Project Status** dialog box provides you with a concise way of viewing all the options selected in your project.

➤ **How to Display the Project Status Dialog Box:**

Step 1: Click the **Run** toolbar button or



Step 2: Select **Run | Status** from the menu.

The **Project Status** dialog box is composed of six (6) tabs, each one containing the options you have selected for each pathway, including: **Control**, **Source**, **Receptor**, **Meteorology**, **Terrain Grid**, and **Output**. The icon located on the upper right corner of each tab also can easily identify each pathway tab; the same icon that is also used to identify each pathway window.

A summary of the options selected for the current project is displayed on the appropriate tab. Options not defined or not being used will appear as a dashed line (----).

You can identify which model you are using, by checking the dialog box title. On the top section of the dialog box, the name of the input file and output file for the selected model is displayed.

On the bottom section of the dialog box, a message identifies if your project is complete or not.

The following buttons are available in the **Project Status** dialog box:

.... Click this button to display the **Details** dialog box and get information on any missing data.

.... 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.

.... Click this button to run the U.S. EPA model (ISCST3, AERMOD, or ISC-PRIME).

.... Click this button to close the **Project Status** dialog box.

Checking for Missing Data in Details

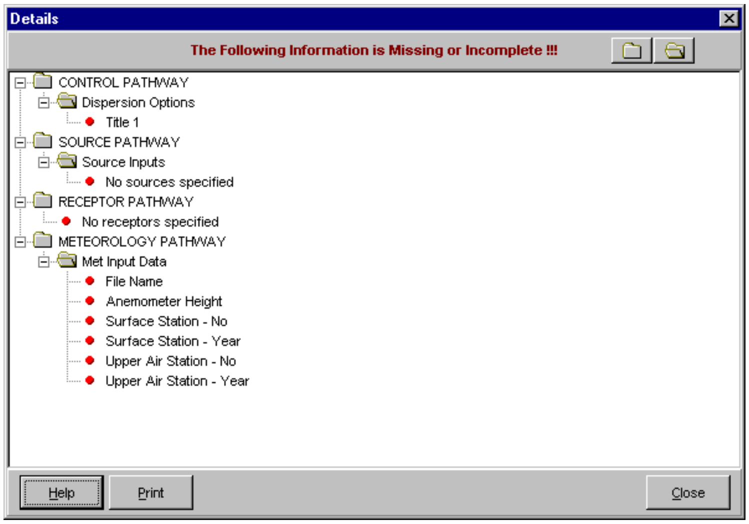
The **Details** dialog box informs you of any missing data or errors in your project. ISC-AERMOD View will detect most of the data that is missing or incorrect, however, it is advisable that you always verify your project input data by selecting **Run | Verify Run (Do Not Run)** from the menu or pressing the **Verify Run** button from the **Project Status** dialog box.

The **Details** dialog box contains a list of all missing information for the current project. The list is subdivided into pathways, with all the missing inputs for each pathway listed under each pathway heading.

➤ **How to Display the Details Dialog Box:**

Step 1: Select **Run | Details** from the menu, or

Step 2: Press the **Details** button from the **Project Status** dialog box.



Details dialog box

The following buttons are available in the **Details** dialog box:

 Collapses all folders

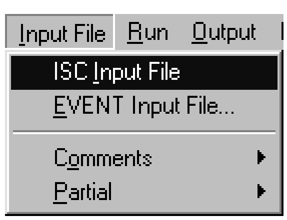
 Expands all folders

Checking the Input File

The input file, also known as the input runstream file, contains the user-specified options for running the U.S. EPA model (ISCST3, AERMOD, or ISC-PRIME). The basic structure of the input runstream file is the same for the ISCST3, AERMOD, and ISC-PRIME 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 ISC-AERMOD View will automatically create the appropriate input file for you.

➤ **How to View the Input File:**

Step 1: If you are in the ISCST3 mode then select **Input File | ISC Input File** from the menu.

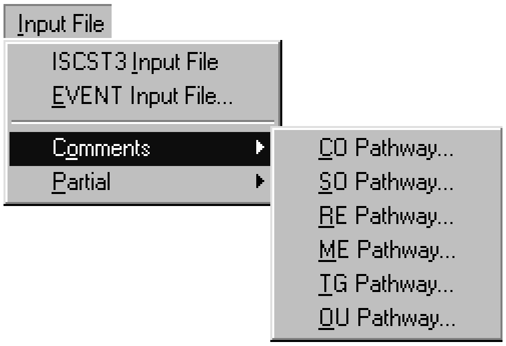


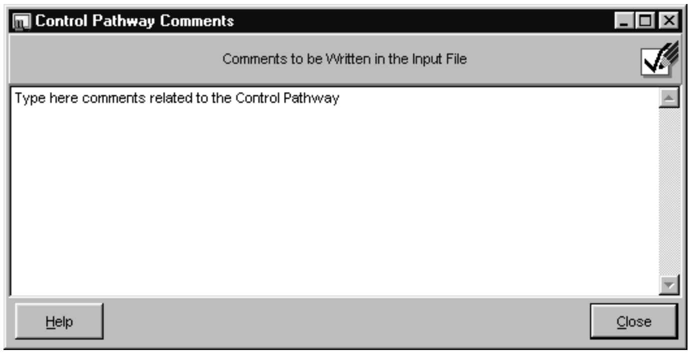
Step 2: If you are in the AERMOD mode then select **Input File | AERMOD Input File** from the menu.

Step 3: If you are in the ISC-PRIME mode then select **Input File | ISC-PRIME Input File** from the menu.

➤ **How to Add Comments to the Input File:**

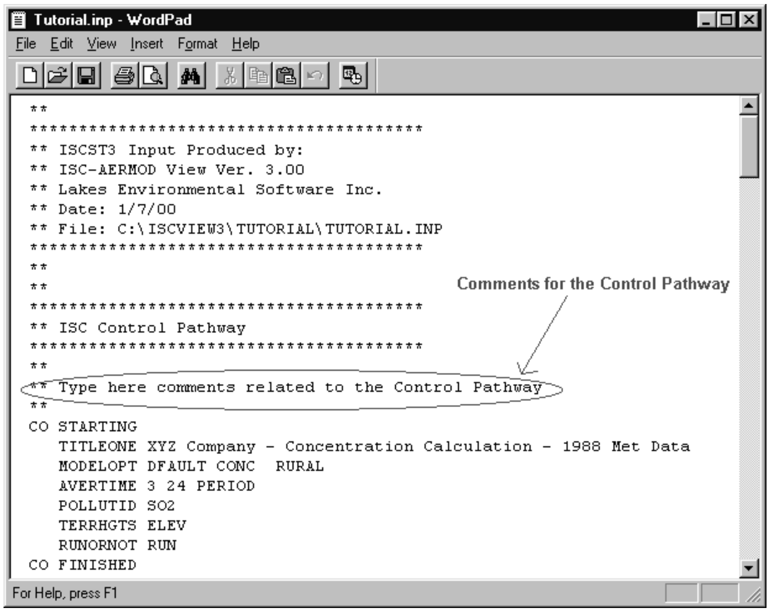
Step 1: Select **Input File | Comments** from the menu and then select the pathway to which you want your comments to be written (CO, SO, RE, ME, TG, or OU). The **Comments** dialog box is displayed.





Comments dialog box

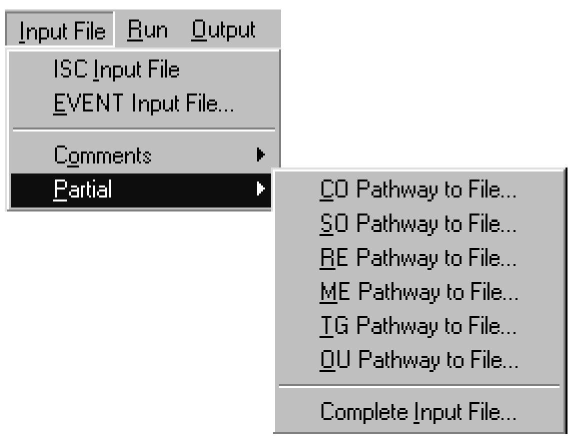
Step 2: Type in the comments and press the **Close** button. Repeat Steps 1 and 2 if you want to write comments to other pathways. The comments that you typed are added to the input file at the start of each pathway section.



ISCST3 Input File

➤ **How to Partial the Input File:**

Step 1: You can save a specific pathway section of your input file into another file. Do this by selecting **Input File | Partial** from the menu and then selecting the pathway you want to save as a partial file. The **Pathway File** dialog box is displayed.



Step 2: Specify the name and location of the file you want to save as the partial input file and click the **OK** button.

Running the U.S. EPA Model

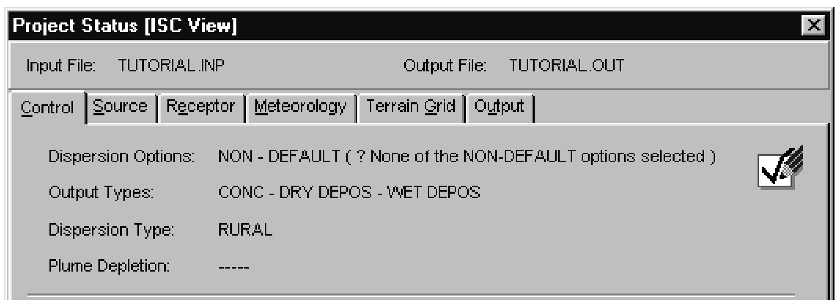
When the data input for your project is complete, you can run the U.S. EPA models: ISCST3, AERMOD, and ISC-PRIME.

The **Run** option indicates that the model will read through all of the inputs in the runstream file regardless of any errors or warnings that may be encountered. If a fatal error occurs in the processing of the run stream information, then further model calculations will be aborted.

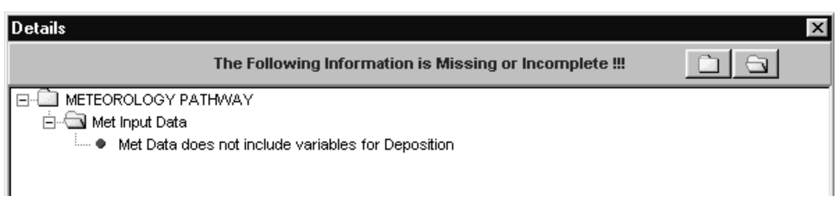
It is advisable that before you run any one of the models, you verify the completeness of your project.

➤ How to Run the U.S. EPA Models:

Step 1. For each model mode (ISCST3, AERMOD, and ISC-PRIME), **select Run | Status**. The **Project Status** dialog box is displayed. Go through all the tabs to verify that all the options you selected are correct.

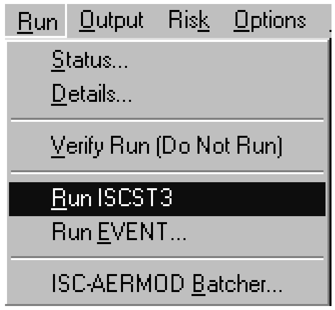


Step 2: Press the **Details** button. The **Details** dialog box is displayed. Check if there is any errors noted in the list. Press the **Close** button. If errors were listed, go and correct them.



Step 3: If no errors were listed, then press the **Verify Run** button. With the **Verify Run** option (Do Not Run option) the U.S. EPA model will only process the input runstream (input file) data for errors and summarize the setup information. The errors and warning messages are then written into the Output File.

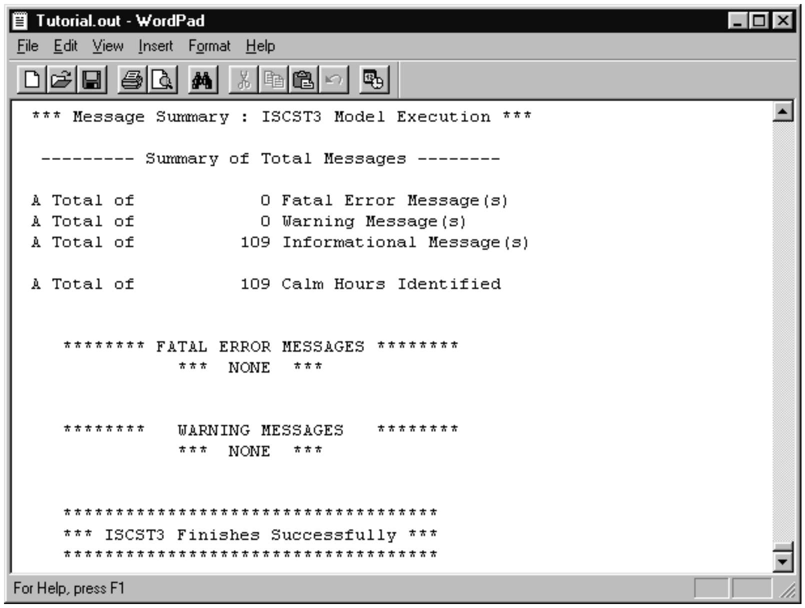
Step 4: If any errors are detected, you should correct them. After correcting all the problems, click on the **Run** button from the **Project Status** dialog box, or select one of the following, depending on the model you are currently working on: **Run | Run ISCST3**, or **Run | Run AERMOD**, or **Run | Run ISC-PRIME**.



Step 5: A window appears on your screen, to provide status information about the simulation. In this window, the model first indicates that setup information is being processed and then gives the Julian day currently being processed. If no status is seen on your window screen, the model did not load into memory properly. If the model stops after completing the setup processing, then either the **Do Not Run** option was selected for the current run or a fatal error was encountered during the setup processing.

Step 6: When the execution of the model is completed, the window is closed and then a dialog box is displayed informing you whether or not your run was successful. From this dialog box, you have the option of viewing the Output File, contours for your Plotfiles, or just going back to your ISC-AERMOD View project.

Step 7: Press the **Output file** button, if you want to check the output file. The output file contains an echo of the input runstream file at the beginning of the file. This is followed by a summary of the inputs (modeling options, source data, receptor data, and meteorological data). Model results are summarized next. The final part of the output file is the summary of messages for the complete model run.



Output File – Message Summary

Step 8: Press the **Contours** button, if you want to view contours of your plotfiles. POST View is displayed and the contour plot for the first Plotfile defined on the **OU-Contour Plot Files** window will be automatically generated. You can find out more about POST View in Chapter 11.

Step 9: Press the **None** button if you just want to go back to your project.

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.

The EVENT model was designed to work in conjunction with the ISCST3 and ISC-PRIME models. The ISCST3 and ISC-PRIME models actually generate the input file needed to run the EVENT model. The input file for the EVENT model is based on the options selected for the ISCST3 or ISC-PRIME model.

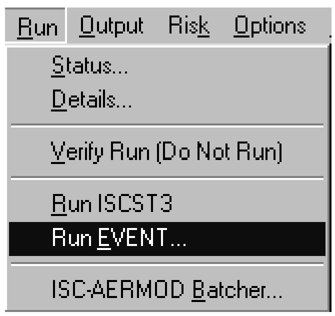
The results contained in the EVENT output file will be affected by the selections made in the **OU-Tabular Outputs** window, such as the highest and second highest 24 hour averages etc… The user can also specify files, called MAXIFILES, that contain all the information on occurrences where concentrations of a substance violated the threshold limit for the particular substance. These files are optional and are specified in the **OU-Threshold Violation Files** window.

The EVENT model is an optional process. In order to use this model the user must go to the **CO-Optional Files** window and select the **EVENT Input File** option.

➤ **How to Run the U.S. EPA EVENT Model:**

Step 1: Go to the **CO-Optional Files** window and select the **EVENT Input File** option.

Step 2: Select **Run | Run ISCST3** from the menu, if you are in the ISCST3 mode or select **Run | Run ISC-PRIME** from the menu if you are in the ISC-PRIME mode. After the ISCST3 or ISC-PRIME run has finished successfully, go to the menu and select **Run | Run EVENT**.



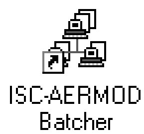
Step 3: A dialog box will appear giving you status information. If all the necessary options to run the Event model were satisfied, then you can press the **Run** button to run the EVENT model. Please note that you need to run the ISCST3 (or ISC-PRIME) model before running the EVENT model.

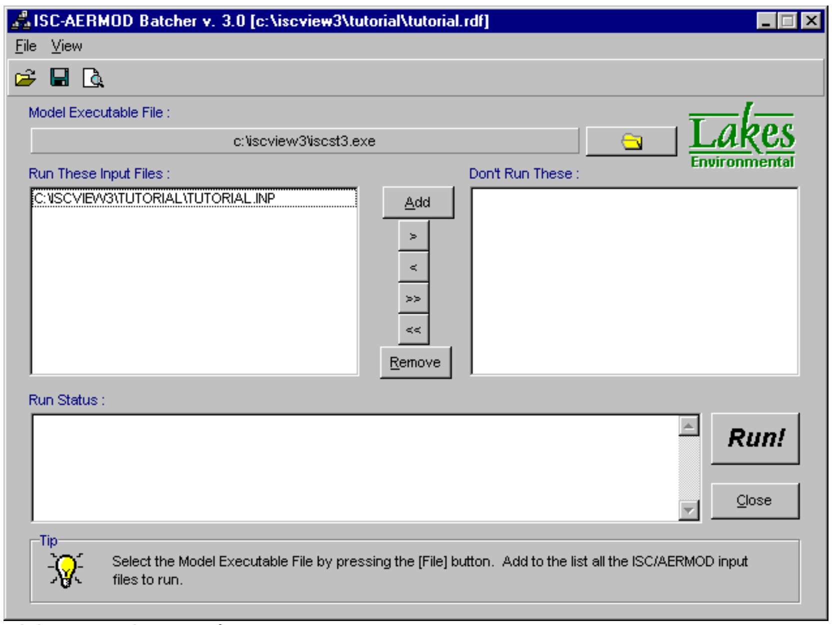
Step 4: Once the model run is complete, another dialog box will appear asking you if you want to see the EVENT output file.]

CAUTION! 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. Thus, please be sure you have adequate space on your hard drive.

Running Your Project Using ISC-AERMOD Batcher

The ISC-AERMOD View Batcher is a utility that allows you to run multiple input files outside the ISC-AERMOD View interface. This utility can be accessed from the ISC-AERMOD View interface or by double clicking on the ISC-AERMOD Batcher icon located in the Lakes Environmental Program Manager group.





ISC-AERMOD Batcher

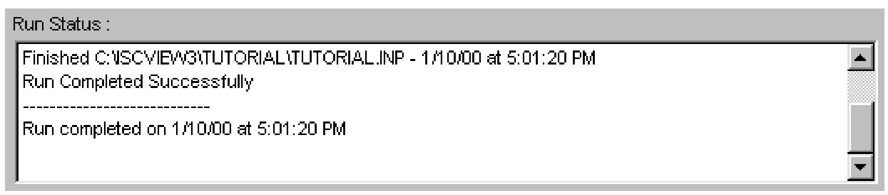
➤ **How to Run Your Project Using the ISC-AERMOD Batcher:**

Step 1: Select **Run | ISC-AERMOD Batcher** from the menu. The ISC-AERMOD Batcher is displayed.

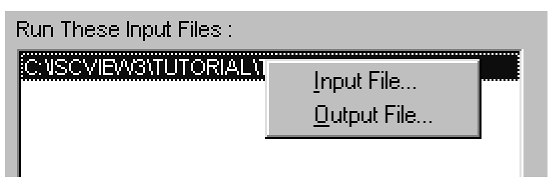
Step 2: Press the **Specify File** ( ) to specify the Model Executable File. If you access the ISC-AERMOD Batcher from the ISC-AERMOD View interface, then the model executable will be automatically specified and will be the same as the one you have specified in the **Preferences** dialog box.

Step 3: Press the **Add** button to add Input Files to the **Run These Input Files** list. If you access the ISC-AERMOD Batcher from the ISC-AERMOD View interface, then your current project input file is automatically specified under the **Run These Input Files** list.

Step 4: Press the **Run** button to run the selected Input Files. After the run is complete, the **Run Status** window will display the date and time the run finished and if the run was successfully.



Step 5: To preview the input file or output file, select the file from the list and right click on it. A floating menu is displayed. Select the option you want to preview.

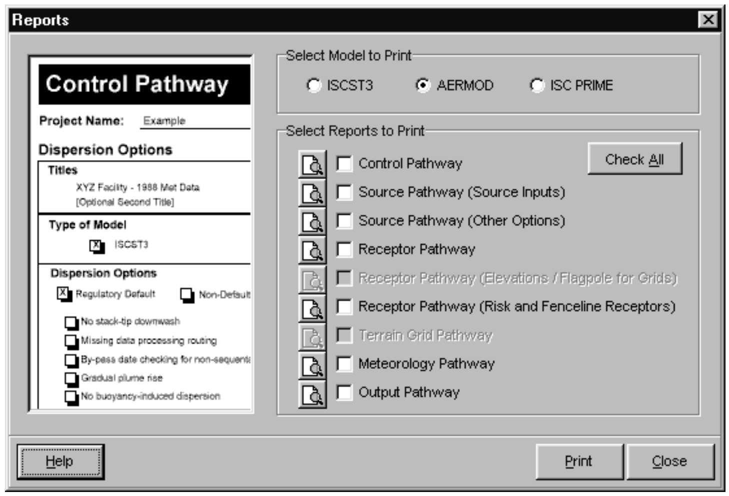


Printing Reports

ISC-AERMOD View can produce reports on the input options for your project. These reports are divided into pathways. Reports are available for all the three modes: ISCST3, AERMOD, and ISC-PRIME.

➤ **How to Preview and Print Reports:**

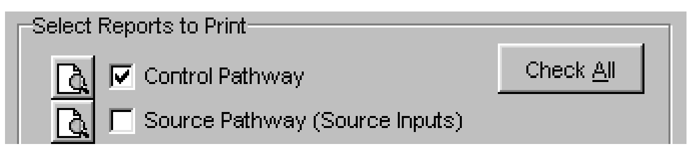
Step 1: Select **File | Reports** from the menu or press the **Reports** menu toolbar button. The **Reports** dialog box is displayed.

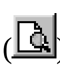


Reports dialog box

Step 2: Select the model for which you want the reports to be generated (ISCST3, AERMOD, or ISC-PRIME).

Step 3: Check the boxes for all the reports you want to print. Press the **Check All** button if you want to select all the available reports.



Step 4: To preview a report, press the **Preview** button () for the desired report. From the preview window, you can also print the report.

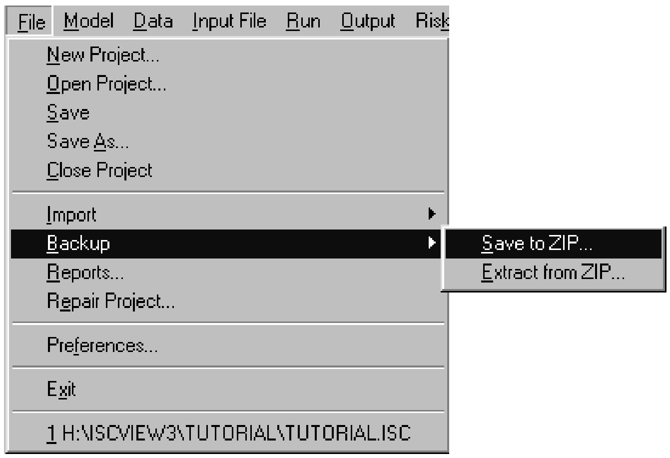
Step 5: To print all the selected reports, press the **Print** button located on the bottom of the **Reports** dialog box.

Backing Up Your Project

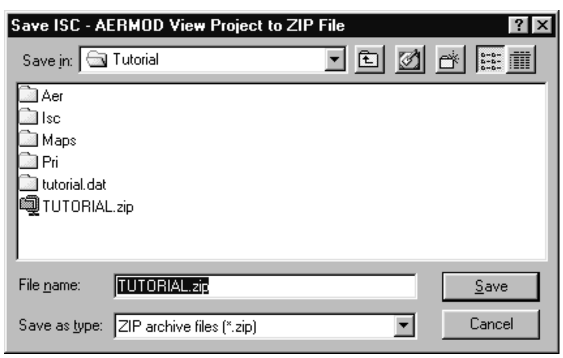
To be safe, it is a wise idea to backup your ISC-AERMOD View project from time to time. This way, you can maintain a "good" copy of the project in the event of a power loss that corrupts the database files (see also Repairing your Project). The backup file is a compressed version of your project file and includes all the project files and folders created by the application. Your maps, meteorological data files, terrain data, or any other file that do not have the same name as your project will not be included in the backup. The backup file has the extension .ZIP.

➤ **How to Backup your Project:**

Step 1: Select **File | Backup | Save to ZIP** from the menu.

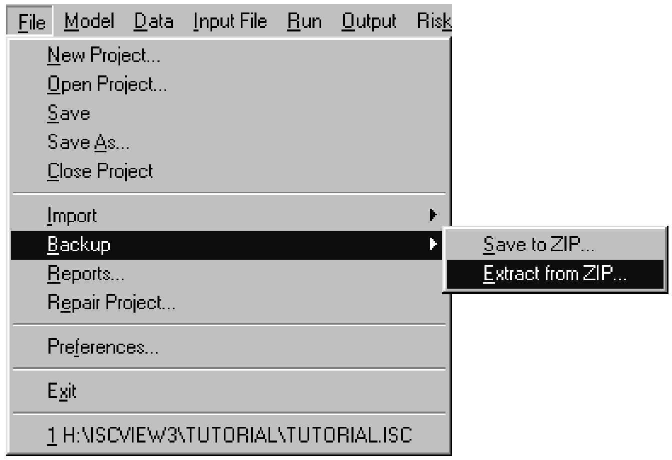


Step 2: By default, the project backup will be saved in the project directory with the same name as your project but extension .ZIP. Change these defaults if you want and press the **Save** button. A backup copy of your project is created.

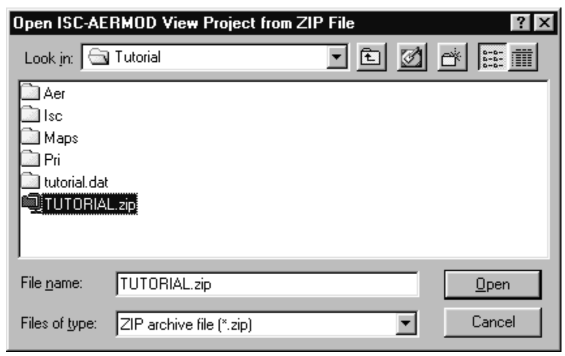


➤ **How to Extract your Project from the Backup File:**

Step 1: Select **File | Backup | Extract from ZIP** from the menu.



Step 2: Locate the ISC-AERMOD View backup file and press the **Open** button. The project is extracted from the backup copy and automatically loaded into ISC-AERMOD View.

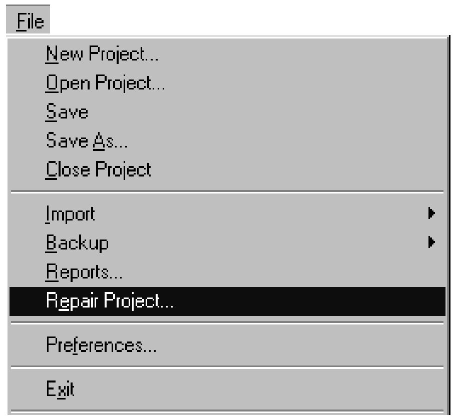


Repairing Your Project

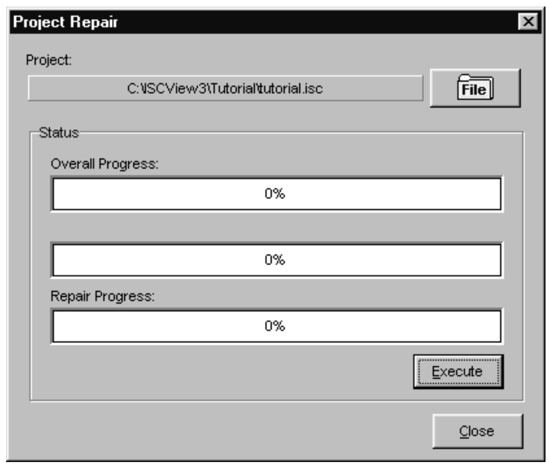
If your system crashes or the power goes off while you are working in your ISC-AERMOD View project, the database files may become corrupted. If you try to open your project and ISC-AERMOD View gives you a message saying that it can not open the project because it may be corrupted, then you should use the **Repair** tool.

➤ **How to Repair Your Project:**

Step 1: Select **File | Repair Project** from the menu.

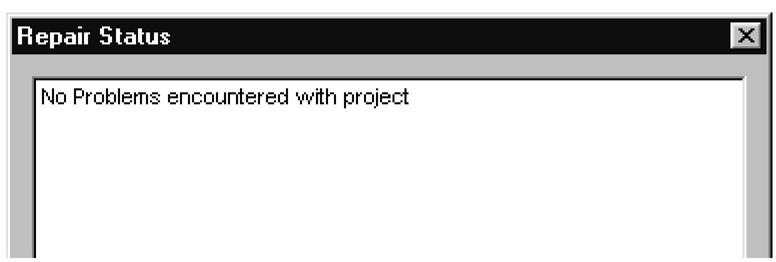


Step 2: The **Project Repair** dialog box is displayed. Press the **File** button to specify the ISC-AERMOD View project to be repaired. Click on the **Execute** button to start the repair process.



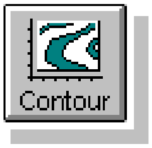
Project Repair dialog box

Step 3: When the repair process is finished, the **Repair Status** dialog box is displayed informing you about the status of the repair.



CHAPTER 11

POST View



POST View is a Windows post-processor specially designed to handle ISC/AERMOD plotfiles. Plotfiles are generated for every ISC-AERMOD View run. In ISC-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 does all the time-consuming tasks of getting all the plotfiles generated by the models (ISCST3, AERMOD, and ISC-PRIME), gridding them and displaying them for you. For easy reference and visualization of your results, you can import site 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 to the clipboard and paste them into your favorite Windows application.

This chapter explains all the options available in POST View to post-process ISC/AERMOD Plotfile results.

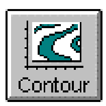
Contents

Starting POST View

POST View can be used from within ISC-AERMOD View or as a stand-alone application. See below the ways to start POST View.

1. When the model run finishes successfully, you have the option of going straight to POST View by pressing the **Contours** button. This button is located on the dialog box that is displayed when you finish running the model.

2. Press the **Contour** toolbar button located on the ISC-AERMOD View menu toolbar.



3. Select **Output | Contour** from the ISC-AERMOD View menu.

4. Double click the **POST View** icon located on the Lakes Environmental program group.

The POST View Window

POST View is a true Windows MDI (multiple-document interface) program which allows you to have multiple contour plots open, in multiple windows, at one time. POST View offers you all the basic options necessary to view and enhance the results of your ISC/AERMOD Plotfiles. See below the components of the POST View window.

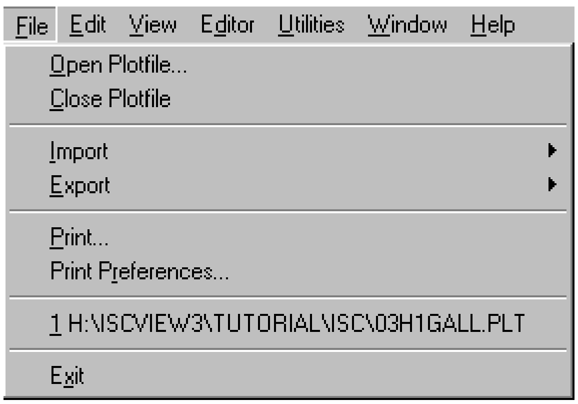
❒ **Menu Bar**
❒ **Menu Toolbar Buttons**
❒ **Title Bar**
❒ **Plotfile List**
❒ **Output Type List**
❒ **Max Panel**
❒ **Toolbar**
❒ **Color Ramp**
❒ **Axis Labels**
❒ **Scroll Bars**

❒ **Status Bar**

Menu Bar

The following is the description of each menu option:

File (Alt, F)



Open Plotfile .... Displays the **Open Plotfile** dialog box, allowing you to open an ISC/AERMOD Plotfile.

Close Plotfile .... Closes the contour plot window that has the focus.

Import .... Displays the following submenu options:

- ➤ **Base Map:** Displays the following submenu options:
  - ◆ **DXF:** This option allows you to import a base map in AutoCAD DXF file format (*.dxf).
  - ◆ **BMP:** This option allows you to import a bitmap base map image (*.bmp).
  - ◆ **DLG:** This option allows you to import a USGS Digital Line Graph map (*.dlg).
  - ◆ **LULC:** This option allows you to import a USGS Land Use and Land Cover map (*.luc).
  - ◆ **Shapefile:** This option allows you to import an ArcView Shapefile (*.shp).
- ➤ **Blanking File:** This option allows you to import the ISC-AERMOD View plant boundary file (*.rpb) to function as a blanking file.

- **ISC-AERMOD View Project:** This option allows you to import the ISC-AERMOD View project file (*.ISC) for the current plotfile so that all the sources and buildings that were specified for your ISC-AERMOD View project are displayed together with your contour plots. Receptor locations are automatically displayed for each plotfile (represented by a green cross).

Export...................................... Displays the following submenu options:

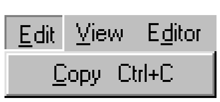
- **Base Map:** Displays the following submenu options:
    - **DXF:** Displays the **Export As DXF Base Map** dialog box, allowing you to save the current contour plot in DXF file format.
    - **BMP:** Displays the **Export As Bitmap** dialog box, allowing you to save the current contour plot as a Windows bitmap.
- **Surfer:** Displays the **Export to Surfer** dialog box, allowing you to export the current contour plot to Surfer in grid file format (*.grd).

Print... Displays the **Print Preview** dialog box from where you can select printing options and print your current contour plot.

Print Preferences.................... Displays the **Preferences** dialog box, allowing you to define printing and labeling options.

Exit.. Closes POST View.

Edit (Alt, E)



Copy (Ctrl+C).......................... This option allows you to copy the contents of the drawing area of the current window to the Clipboard, so you can paste to any Windows application that supports pasting from the clipboard. This feature is very useful when you are writing your final report in a Windows word processor and want to paste a copy of your contour plots, within a specific section of your report. As a

Windows Metafile image, the clipboard image can be re-sized to accommodate your needs and preferences.

View (Alt, V)

Contour Options.................... Displays the **Contour Options** dialog box, where you can select contouring options for the current ISC/AERMOD Plotfile.

Overlay Control...................... Displays the **Overlay Control** dialog box, where you control which objects are to be displayed on the drawing area.

Toolbar.................................. Turns on and off the display of the toolbar on the screen.

Color Ramp .... Turns on and off the display of the color ramp on the screen.

Editor (Alt, D)

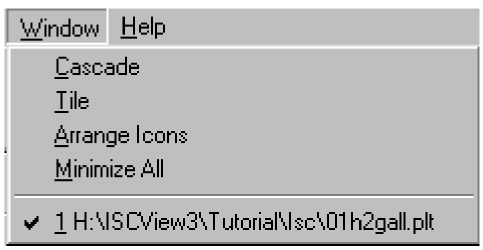
Plotfile .... Automatically displays the ISC/AERMOD Plotfile for the current contour plot in Microsoft WordPad.

Other File................................ Opens the Windows WordPad word processor. This allows you quick access to a word processor, where you can manipulate any text file.

Utilities (Alt, U)

Concentration Converter........ Displays the Concentration Converter utility, which allows you to generate new ISC Plotfiles from existing ones without having to re-run the ISCST3 model. You can apply a multiplier or an additive to the concentration values (can be used, for example, when you want to take into account background concentrations), you can change the averaging period, or change the concentration units to PPM.

Window (Alt, W)



Cascade.................................. Displays the contour plot windows in cascading position.

Tile... Arranges the contour plot windows vertically on the screen (no overlapping).

Arrange Icons.......................... Rearranges the contour plot windows on the screen so that each window has a portion of the screen.

Minimize All............................ Minimizes all the contour plot windows that are open.

List of Files .... Lists of ISC/AERMOD Plotfiles that are currently open. The window that is currently selected appears in the list with a check mark. By selecting a plotfile from this list, you can switch from one plotfile to another.

Help (Alt, H)

Contents.................................Displays POST View Help Contents, from which you can select topics.

Search For Help on.................Lets you search for Help on a particular topic.

Help on Help...........................Displays information for How to Use Help.

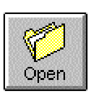
Team.......................................Displays information on the POST View development team.

Technical Support..................Displays a dialog box containing available technical support options for Lakes Environmental software.

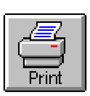
About......................................Displays the copyright notice and version number of POST View.

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:

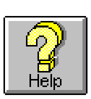


.......Displays the **Open Plotfile** dialog box, allowing you to open an ISC/AERMOD Plotfile.



.......Displays the **Print Preview** dialog box from where you can select printing options and print your current contour plot.

.......Automatically displays the ISC/AERMOD Plotfile for the current contour plot in Microsoft WordPad.



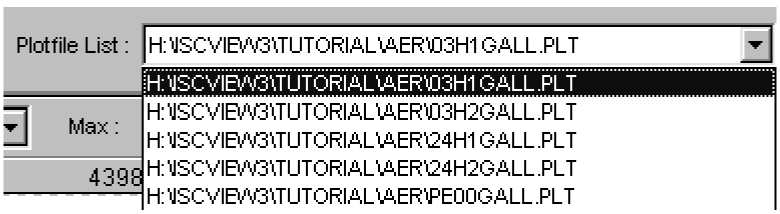
.......Displays POST View Help Contents, from which you can select topics.

Title Bar

The **Title bar** displays the name of the application program, POST View. If any contour plot window is maximized and has the focus, then the name of the ISC/AERMOD Plotfile for that particular window is also displayed on the Title bar.

Plotfile List

The **Plotfile List** drop-down list box will display the following:



Plotfile List drop-down list box

1. If you open POST View from within ISC-AERMOD View then all the plotfiles that you have defined on the **OU-Contour Plot Files** window, for the current run and current model will be displayed in the **Plotfile List** drop-down list box. When POST View is open, the first plotfile displayed in the list will be automatically loaded and displayed on the screen. Every time you open a file using the **File | Open Plotfile...** menu command or the **Open** menu toolbar button, then the name of the file will be displayed on the **Plotfile List** drop-down list box in addition to the existing ones.

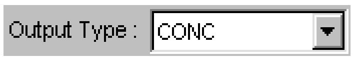
2. If you open POST View as a stand-alone application, then the **Plotfile List** drop-down list box will be empty. However, all the plotfiles you open will be automatically stored in this list. If you close a contour plot window, the name of the plot file for that window will still be in the **Plotfile List**. At any time, you can select the plotfile from the list to re-open it.

Output Type List

The **Output Type** drop-down list box displays output type options for the current ISC/AERMOD Plotfile. The following output types may be available for each plotfile:

- **CONC:** Select this option to show concentration results for the current plotfile.
- **DEPOS:** Select this option to show deposition results for the current plotfile.
- **DDEP:** Select this option to show dry deposition results for the current plotfile.
- **WDEP:** Select this option to show wet deposition results for the current plotfile.

When more than one **Output Type** option was selected in ISC-AERMOD View (**CO-Dispersion Options** window), then the ISC/AERMOD Plotfile will include all of the output types selected.



Contour plots for each Output Type option can be viewed by selecting one of these options from the **Output Type** drop-down list box. Every time you select a different Output Type option for the same ISC/AERMOD Plotfile, the contour plot results for the new option will replace the contour plot results for the previously selected Output Type option.

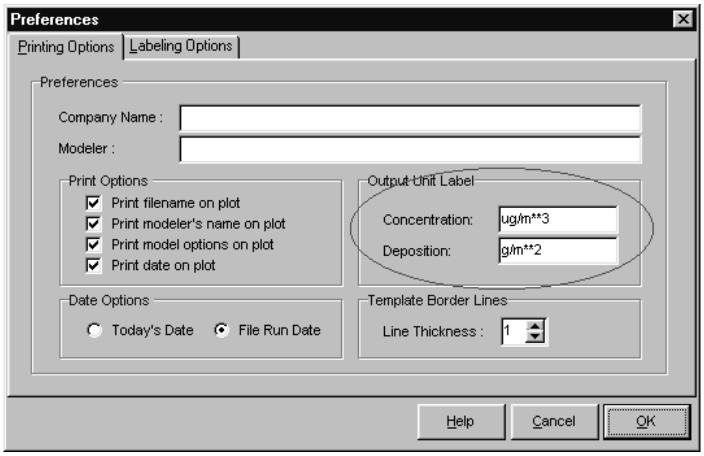
Max Panel

The **Max** panel displays the maximum **Concentration** or **Deposition** value for the current ISC/AERMOD Plotfile. The maximum concentration and/or deposition value is read from the plotfile, which gives the calculated concentration and/or deposition values at each receptor location.

The unit label displayed for the concentration or deposition is the unit label specified on the **Printing Options** tab on the **Preferences** dialog box. The default unit value for concentration calculations is ug/m**3 (micrograms per cubic meter) and for deposition calculations is g/m**2 (grams per square meter).

Also displayed on the **Max** panel is the receptor location where the maximum concentration or deposition value occurs.

Note: There is no information in the ISC/AERMOD Plotfiles regarding units for concentration and deposition values. The user is responsible for specifying the appropriate units in the **Preferences** dialog box.

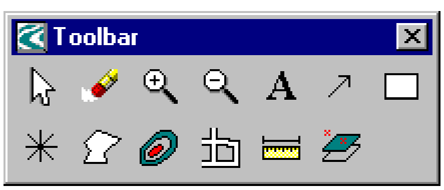


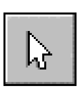
Output Unit Label on Preferences dialog box

Toolbar

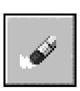
The POST View toolbar can be docked or floating. A docked toolbar is a toolbar that is attached to one edge of the program window. You can dock the toolbars below the menu toolbar, above the status bar, or to the left or right edge of the POST View window. A floating toolbar is a toolbar that floats and is not attached to the edge of the program window. To move a docked toolbar, click on the toolbar and drag it to the new location. If you drag the toolbar to the edge of the program window, it becomes a docked toolbar.

The tools available on the toolbar are:

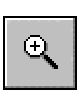




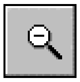
Select Tool: With this tool, you can select annotation objects from the drawing area, move them around, and double-click on them to display their properties. You can also select an object and click the right mouse button to display its properties.



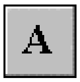
Delete Tool: Use this tool to delete an object from the drawing area. Click the **Delete** tool and then click on the desired object to be deleted. A message box appears requesting confirmation that you want to delete the selected object.



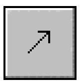
Zoom In Tool: This tool allows you to magnify a portion of the drawing area. To zoom in on portion of the drawing area, click on the **Zoom In** tool. Next, move the cursor into the drawing area and, holding down the left mouse button, drag up or down on a diagonal until the dashed rectangle contains the area you want to magnify. Finally release the mouse left button.



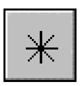
Zoom Out Tool: Use this tool to return the drawing area back to the initial scale size.



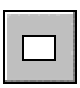
Text Annotation Tool: Use this tool to write text on the drawing area. Click this tool and click on the drawing area where you want to write the text. The **Text Annotation** dialog box is displayed. Type the text and press the **Font** button to select the font type, size, style, color, etc.



Arrow Annotation Tool: Use this tool to draw arrows or lines on the drawing area. Click this tool and, holding down the left mouse button, draw an arrow on the drawing area. Release the left mouse button. The **Arrow Annotation** dialog box is displayed. Change the angle, the line color or the line style, and select if you want an arrowhead to be displayed.



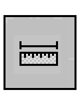
Marker Annotation Tool: This tool enables you to place a marker on the drawing area. Click this tool and click the drawing area where you want to place a marker. The **Marker Annotation** dialog box is displayed. Select the marker style and color and click the OK button.



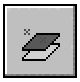
Rectangle Annotation Tool: Use this tool to draw any rectangle on the drawing area. Click this tool and, holding down the left mouse button, draw a rectangle on the drawing area. Release the left mouse button.

Importing Blanking File Tool: This tool allows you to import a blanking file (*.rpb) which contains the plant boundary coordinates. Blanking is used to mask the contour plot lines inside the plant boundary area. The first line of a blanking file must contain the number of X, Y coordinate pairs that follow (one pair per line). ISC-AERMOD View automatically generates this file in the **RE-Cartesian Plant Boundary** window.

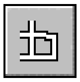
Contour Options Tool: Displays the **Contour Options** dialog box, where you can select contouring options for the current ISC/AERMOD Plotfile.



Measure Distance Tool: This tool allows you to measure the length of objects or distance between two locations. You 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 you what you are measuring and the length readout will appear beside the cursor. The units of measurement are meters.



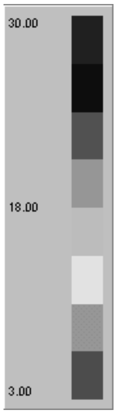
Overlay Control Tool: Displays the **Overlay Control** dialog box, where you control which overlays are to be displayed on the drawing area. This tool is very useful when you want to hide certain objects but you do not want to delete them.



Import Base Maps Tool: Use this tool to import a site map to be displayed in the drawing as a backdrop. A floating menu with available file format import options is displayed when this tool is pressed.

Color Ramp

The **Color Ramp** is displayed only for shaded contour plots. You can dock the **Color Ramp** to the left or right edge of the POST View window. To move from one location to the other, click on the **Color Ramp** panel and drag it to the new location.

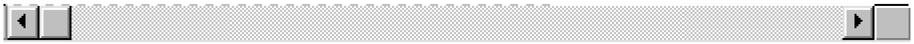


Axis Labels

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 domain area.

Scroll Bars

If you use the **Zoom In** button to magnify a section of the drawing area, then you can use the vertical and horizontal **Scroll Bars** to view other sections of the contour plot area.



Status Bar

The **Status Bar**, located on the bottom of the POST View window, is used to display information about the current contour plot (contour plot window that has the focus). The **Status Bar** is divided into two sections:

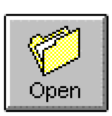
- **Modeling Options:** This section displays the modeling option keywords applicable to the ISC/AERMOD Plotfile results being displayed on the current window (window that has the focus). The displayed modeling options are read from the fourth line of the header of the ISC/AERMOD Plotfile.

- **X and Y Coordinates:** This section displays the X and Y coordinates of the cursor position on the drawing area. These coordinate values are the real domain coordinates for the site area being modeled.

Opening ISC/AERMOD Plotfiles

➤ **How to Open One or More Plotfiles:**

Step 1: Click the **Open** menu toolbar button or from the menu select **File | Open Plotfile...**. The **Open Plotfile** dialog box appears.



Step 2: Select the plotfile to be opened and click the **OK** button. A progress meter dialog box appears on the screen. Wait a moment for the contour plot to be generated.

Note: If you have plotfiles listed in the **Plotfiles List** drop-down list box, then you can open any one of these files by clicking on its file name.

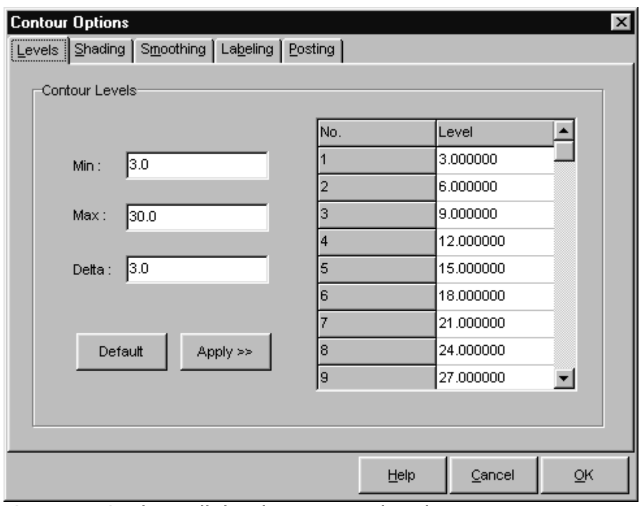
Contour Options

You can have access to the **Contour Options** dialog by clicking on the **Contour Options** tool or selecting **View | Contour Options** from the menu. The **Contour Options** dialog contains options to enhance and change the contour settings for the current contour plot.

Five tabs compose the **Contour Options** dialog box. The options contained in each one of these tabs are explained below:

Levels Tab

In the **Levels** tab, you specify the contour levels for the current contour plot. When selecting contour levels, you have two choices, automatic or user defined contour levels.



Contour Options dialog box – Levels tab

1. **Automatic Option:** You should specify the minimum and maximum concentration value to be displayed and at which interval, and press the **Apply >>** button. POST View automatically sets up the contour levels according to the selected interval and displays them in the table.

2. **User Defined Option:** You can specify values for the **Min.**, **Max.**, and **Delta** fields, apply these values to the table, and modify them as desired. Also, you can type new values directly in the table.

   - **Min.:** This is the minimum concentration or deposition level to be displayed for the plotfile. If you press the **Default** button, the minimum concentration/deposition level found in the ISC/AERMOD Plotfile is displayed in this field.

   - **Max.:** This is the maximum concentration/deposition level to be displayed for the plotfile. If you press the **Default** button, the maximum concentration/deposition level found in the ISC/AERMOD Plotfile is displayed in this field.

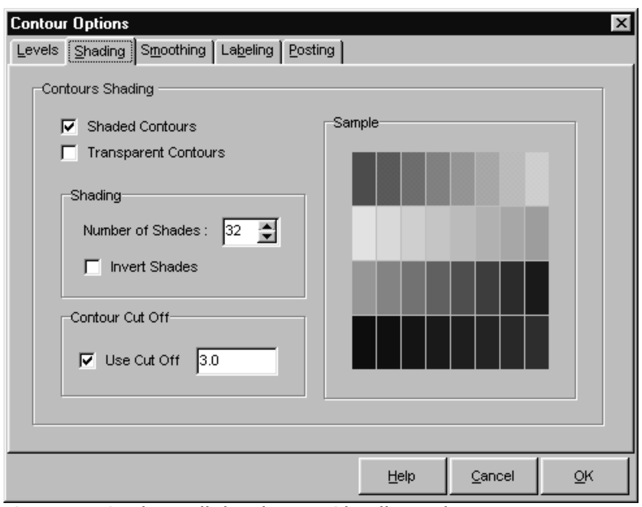
- **Delta:** This is the interval to be used between contour levels. The default delta is set equal to the **Min.** value.

Use this button to get the minimum and maximum concentration values available for the current ISC/AERMOD Plotfile.

Use this button to apply the values specified in the Min, Max, and Delta fields to the table. Once these values are applied, they can be customized in the Level table.

Shading Tab

In the **Shading** tab you define the shading style for the contour plots. The following are the options available:



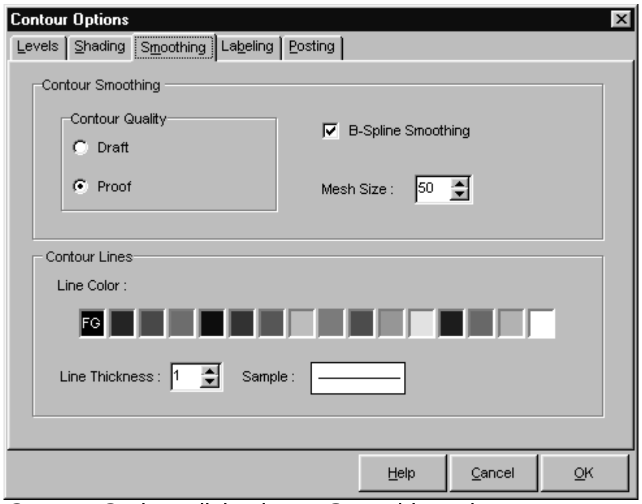
Contour Options dialog box – Shading tab

- **Shaded Contours:** Check this box if you want your contours to be shaded.
- **Transparent Contours:** Check this box if you want your contours to have transparent shading. This is very useful when you have a site map on the background and still want to show information from the site map.
- **Use Cut Off:** The use of the cut off allows the contouring package to not show shades for contours below a certain value. This is useful when you want mask regions of a plume where the concentration is zero.
- **Number of Shades:** Here you specify the number of colors for the shading. As you change the number of shades, the sample area is updated.

- **Invert Shades:** Check this box to invert the colors for the shading. Note that as you check or uncheck this box, the shading colors shown on the sample area also change.

Smoothing Tab

In the **Smoothing** tab, you control the size of the contour mesh and the smoothing, color, and thickness of the contour lines. The following are the options available:



Contour Options dialog box – Smoothing tab

Contour Smoothing

- **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.

- **B-Spline Smoothing:** Select this option to get smoother contour lines. This option should be used to get high quality printed outputs.

- **Mesh Size:** This option determines the size of the mesh (grid) used to generate the contour plots. A mesh of 50x50 is used as the default. However, this can be changed at any time.

Contour Lines

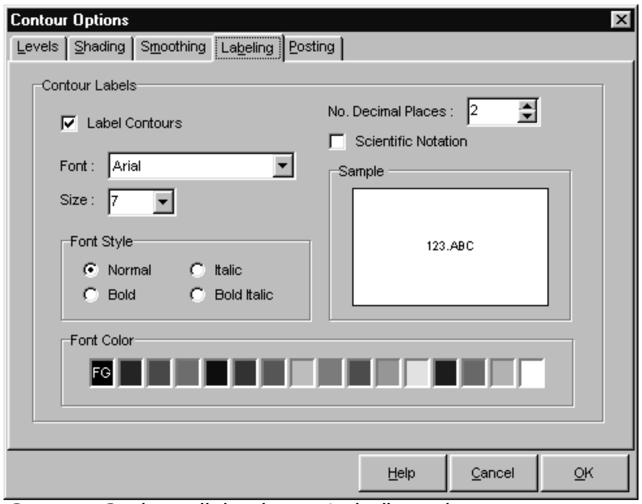
- **Line Thickness:** Specify the line thickness to be used for the contour lines.

- **Sample:** this area shows a sample of the line thickness currently specified for the contours.

- **Line Color:** Click with the mouse pointer to select the color to be used for the contour lines.

Labeling Tab

In the **Labeling** tab you control the font type, font size, style, and color of the contour labels. See below the description of each one of these options:

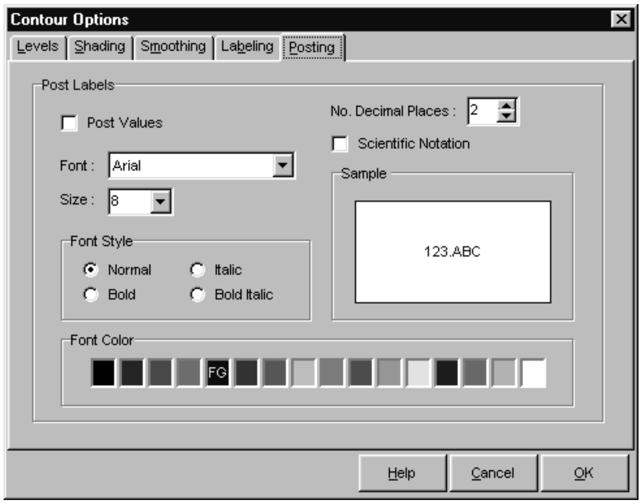


Contour Options dialog box – Labeling tab

- **Label Contours:** Check this box if you want labels to be displayed at your contour lines.
- **Font:** Select a font for the contour label 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 contour labels.
- **Font Color:** Click with the mouse pointer to select the color for the contour labels.
- **No. Decimal Places:** Specify the number of decimal places for the contour labels.
- **Scientific Notation:** Check this box if you want the contour labels to be written in scientific notation.
- **Sample:** As you make selections for color, font, size, and style, the sample is updated to reflect the changes.

Posting tab

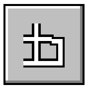
The following are the options available on the **Posting** tab:



Contour Options dialog box – Posting tab

- **Post Values:** Check this box if you want post values to be shown at each receptor node.
- **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.
- **Font Style:** Select a font style for the post value labels.
- **Font Color:** Click with the mouse pointer to select the color for the post value labels.
- **No. Decimal Places:** Specify the number of decimal places for the post values.
- **Scientific Notation:** Check this box if you want post values to be written in scientific notation.
- **Sample:** As you make selections for color, font, size, and style, the sample is updated to reflect the changes.

Importing Site Maps



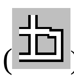
POST View can import one or more base maps in the following file formats:

- AutoCAD DXF File format (DXF)
- USGS Digital Line Graph (DLG)
- USGS Land Use and Land Cover (LULC)
- Bitmap images (BMP)
- ArcView Shapefiles

See more information about each one of these file formats in *Chapter 9 – Graphical Input.*

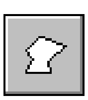
➤ **How to Import Site Base Maps:**

Step 1: Select **File | Import | Base Map** from the menu and then select the appropriate map option.

Step 2: You can also press the **Import Base Map** tool () located in the toolbar and then select the appropriate map option from the floating menu.

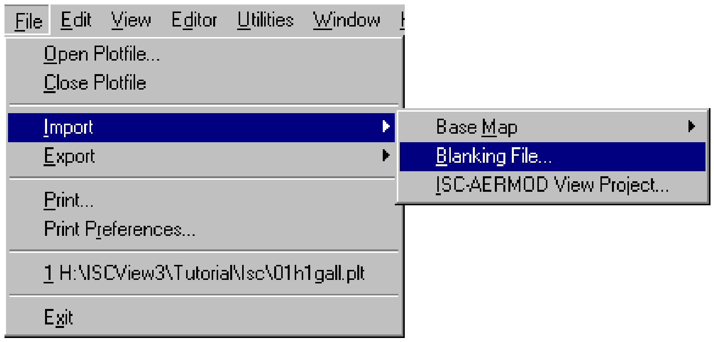
Note: It is important to note that the extents of the imported site map should be within the modeling domain area. If the extents of the site map are outside the modeling domain, then the site map will not be displayed on the drawing area.

Importing a Blanking File

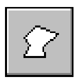


A blanking file must contain the coordinates of all the corners of the polygon that will be used to mask the contour plot lines. The first line in a blanking file must specify the number of X, Y coordinate pairs that follow (one pair per line). You can import the ISC-AERMOD View plant boundary file (*.rpb) to function as a blanking file.

➤ **How to Import a Blanking File:**



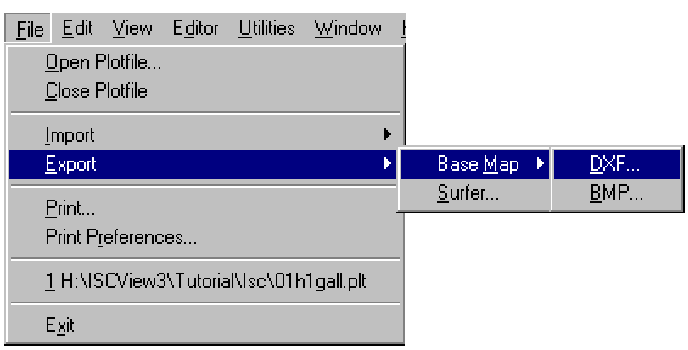
Step 1: Select **File | Import | Blanking File…** from the menu or press the **Import Blanking File tool** from the toolbar. The **Import Blanking File** dialog box is displayed.


Import Blanking File tool

Step 2: Specify the name and path of the blanking file and click the OK button. For every project, ISC-AERMOD View generates the plant boundary file (*.rpb) that contains information on the coordinates for each corner of your plant boundary or fenceline. You can use this file as the blanking file if you want to mask the contour plot lines inside your plant boundary.

Export Options

POST View offers three exporting options. You have access to these exporting options by selecting from the menu:



◆ **File | Export | Base Map | DXF:** This option exports the current contour plot to a file in DXF format (*.dxf). This option will only export the contour plot and any DXF base map. Annotations or any other type of base map will be ignored.

- **File | Export | Base Map | BMP:** This option exports the contents of the drawing area (contour plot, annotations, and maps) to a file in Windows Bitmap format (*.bmp).

- **File | Export | Surfer:** This option generates a grid file (*.grd) of the ISC/AERMOD Plotfile data that can be read by Surfer. POST View generates the grid file using the mesh size specified on the **Contour Options** dialog box – **Smoothing** tab. The default mesh size used by POST View is 50x50.

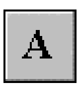
➤ **How to Open the Exported Grid File in Surfer for Windows:**

Step 1: From the menu choose **Contour | Contour Map** to display the **Open Grid** dialog box. Specify the name of the grid file (*.grd) that POST View has setup for you and click the **OK** button. The **Contour Map** dialog box is displayed.

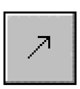
Step 2: Specify any parameters you want in the **Contour Map** dialog box and click the **OK** button.

Annotation Tools

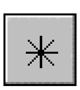
From the toolbar you have access to some annotation tools. These tools are the following:



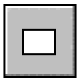
Text Annotation Tool: Use this tool to write text on the drawing area. Click this tool and click on the drawing area where you want to write the text. The **Text Annotation** dialog box is displayed. Type the text and press the **Font** button to select the font type, size, style, color, etc.



Arrow Annotation Tool: Use this tool to draw arrows or lines on the drawing area. Click this tool and, holding down the left mouse button, draw an arrow on the drawing area. Release the left mouse button. The **Arrow Annotation** dialog box is displayed. Change the angle, the line color or the line style, and select if you want an arrowhead to be displayed.



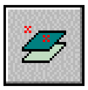
Marker Annotation Tool: This tool enables you to place a marker on the drawing area. Click this tool and click the drawing area where you want to place a marker. The **Marker Annotation** dialog box is displayed. Select the marker style and color and click the OK button.



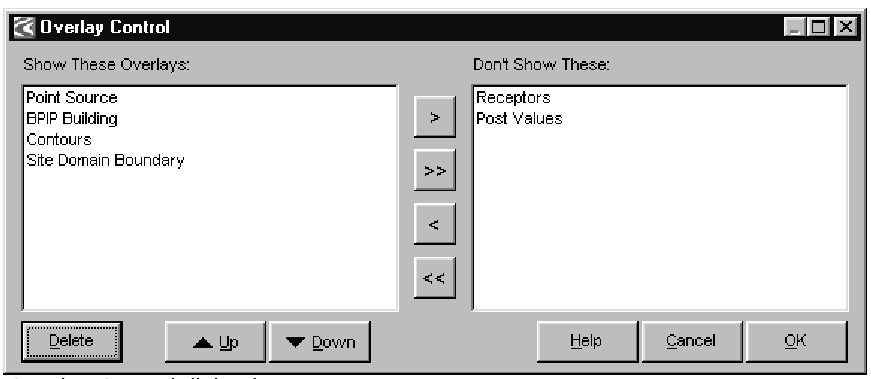
Rectangle Annotation Tool: Use this tool to draw any rectangle on the drawing area. Click this tool and, holding down the left mouse button, draw a rectangle on the drawing area. Release the left mouse button.

See a detailed description on how to use these tools in *Chapter 9 – Graphical Input.*

Overlay Control

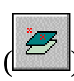


From time to time, during your project, you may wish to remove some objects/overlays from the drawing area, so that you can view or print only the desired objects. From the **Overlay Control** dialog box, you can specify which objects should be displayed on the drawing area.

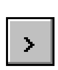
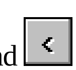


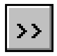
Overlay Control dialog box

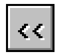
➤ **How to Turn On or Off the Display of Overlays:**

Step 1: From the toolbar, click on the **Overlay Control** tool () or select **View | Overlay Control…** from the menu. The **Overlay Control** dialog box is displayed.

Step 2: The list located on the left-hand side, **Show These Overlays**, contains all the overlays that are currently being displayed. The list on the right hand side, **Don't Show These**, contains the overlays that are not being displayed at the moment. All the base maps you have loaded into POST View (e.g., *.DXF, *.BMP, *.DLG, etc.) are listed in this dialog box. You can easily move overlays from one list to the other using the buttons located between the two lists.

 and  **.......** Moves the selected overlays to the appropriate list.



and 

.... Moves all overlays from one list to the other.

.... Deletes the selected map from the project. This button can only delete map objects.

.... Moves the currently selected overlay up in the list. This list controls the order in which the layers are drawn on screen. You can also click and drag the overlay to the desired list position.

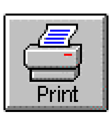
.... Moves the currently selected overlay down in the list. This list controls the order in which the layers are drawn on screen. You can also click and drag the overlay to the desired list position.

Saving Contour Plot Options

When you close a contour plot window, POST View will automatically save all the options you have selected for that particular contour plot, including all the imported base maps. These options are saved in a separate file with the same name of the ISC/AERMOD Plotfile but with the extension .INI.

For example, if you open the plotfile **PLTTUT3.FIL**, then when you close this file, POST View will automatically save all the contour options and base maps you have selected for the generated contour plots to the **PLTTUT3.INI** file. The next time you open **PLTTUT3.FIL**, POST View will read the **PLTTUT3.INI** file and display all the options you have previously selected for that contour plot.

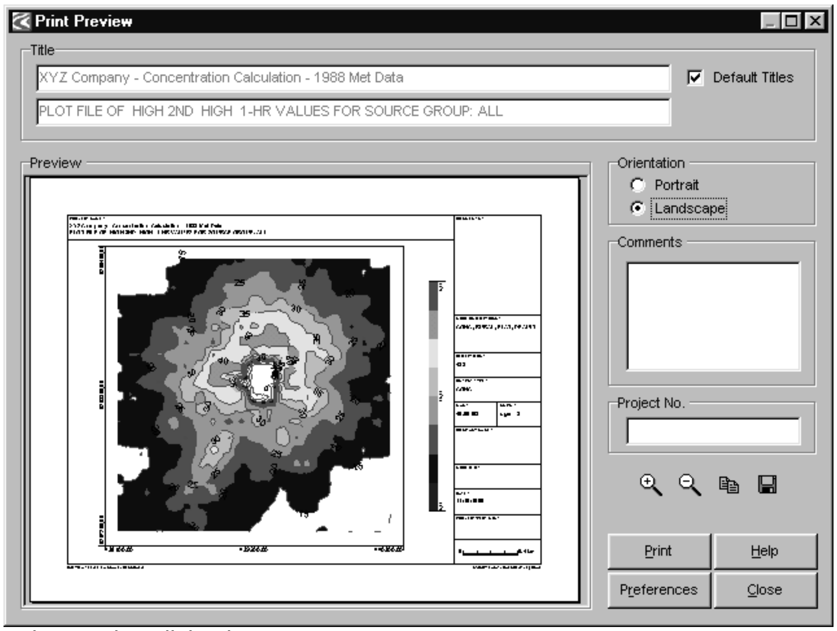
Printing Options



Before you can print your contour plots, POST View displays the **Print Preview** dialog box from where you can select printing options and preview how your printouts will look like. POST View prints your results in templates. These templates were designed so you can have important information on the generated contour plot automatically printed along with your results.

Print Preview

The **Print Preview** dialog box is displayed every time you press the **Print** menu toolbar button or select **File | Print…** from the menu. The **Print Preview** dialog box contains the following options:



Print Preview dialog box

- **Title:** POST View places the first line of title you have specified in your ISC-AERMOD View project (**CO-Dispersion Options** window) as the default title. However, you have the option of specifying a different title by unchecking the **Default Titles** box. The second title is by default the description for the plotfile (read from the header of the plotfile).

- **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 contour plot.

- **Project No.:** In this field you can specify the number or code for your project or plot.

At the lower right corner of the dialog box a series of tools and buttons are available. See the function of each one 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 Windows Metafile. You can then paste into any Windows application that supports pasting of a Windows Metafile from the clipboard.

 **Save to Metafile** ....Select this option if you want to save the printout to a file. The printout will be saved as an Enhanced Windows Metafile (*emf).

.........................Displays the help contents for the options contained in the **Print Preview** dialog box.

.........................Displays the **Preferences** dialog box where you can specify printing and labeling options for your printout outputs.

.........................Displays the **Print** dialog box where you can printing options such as printer, number of copies, paper size, orientation, etc.

.........................Closes the print preview window.

Preferences

In the **Preferences** dialog box you define default options for printing your contour plots. Note that the options you have selected in this dialog box will be used as the default every time you print any contour plot, for your current project or any other project. You have access to the **Preferences** dialog box by selecting **File | Print Preferences** from the menu or clicking on the **Preferences** button located on the **Print Preview** dialog box.

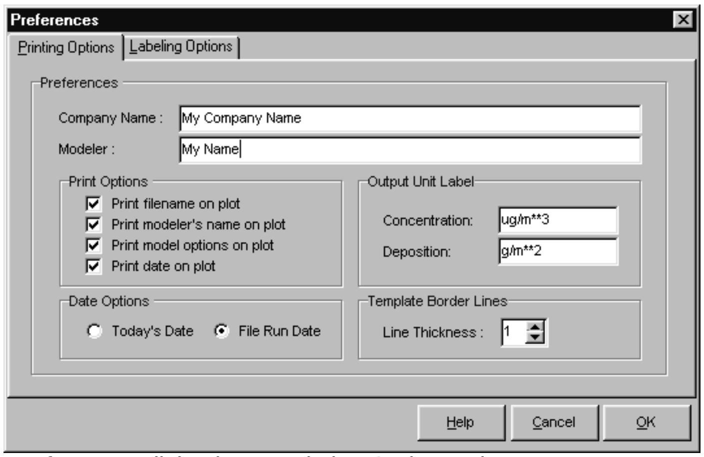
The **Preferences** dialog box contains two tabs, the **Printing Options** tab and the **Labeling Options** tab. See below the contents of each one of these tabs.

Printing Options Tab

The **Printing Options** tab contains the following options:

- **Company Name:** Here you specify the name of your company, which will be printed on the space available for the company name on the printout template. If this field is left blank then no company name will be printed.

- **Modeler:** Here you input the name of the modeler, which will be printed in the space available on the POST View printout template. If this field is left blank then no modeler name will be printed. You can also eliminate the modeler's name by leaving the **Print modeler's name on plot** check box unchecked.



Preferences dialog box – Printing Options tab

<u>Print Options</u>

- **Print filename on plot:** Check this box if you want the ISC/AERMOD Plotfile name to be printed on the page. If you select this option then the full path of the plotfile will be printed on the lower right side of the page.

- **Print modeler's name on plot:** Check this box if you want the name you have typed on the **Modeler** field to be printed in the space available on the printout template.

- **Print model options on plot:** Check this box if you want the modeling options to be printed on the printout template.

- **Print date on plot:** Check this box to print the date on the printout template.

Date Options

- **Today's Date:** This is the current date. Make sure your computer has the correct date setup.
- **File Run Date:** This is the date that the ISC/AERMOD Plotfile was created.

Output Unit Label

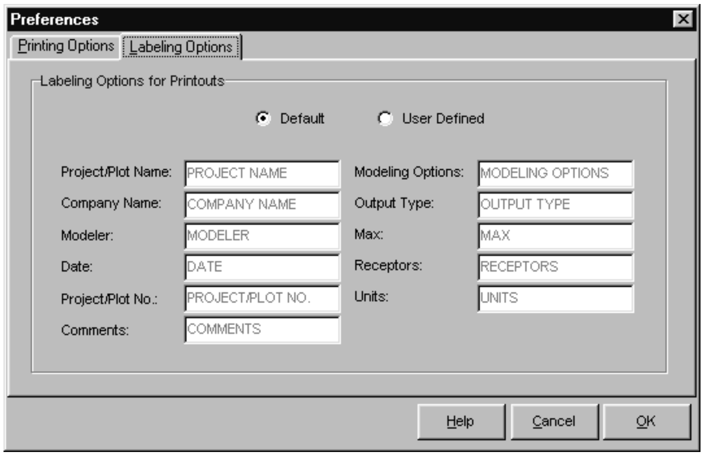
- **Concentration:** You should specify here the unit that was used in ISC-AERMOD View for the calculated concentration values. The ISC/AERMOD Plotfiles do not contain the output unit information and, as a result, POST View uses the ISC-AERMOD View default output unit of $\mu g/m^3$ (micrograms per cubic meter) for concentration values.
- **Deposition:** You should specify here the unit that was used in ISC-AERMOD View for the calculated deposition values. The ISC/AERMOD Plotfiles do not contain the output unit information and, as a result, POST View uses the ISC-AERMOD View default output unit g/m^2 (grams per square meter) for deposition values.

Template Border Lines

- **Line Thickness:** This is the thickness of all the border lines for the printout templates.

Labeling Options Tab

In the **Labeling Options** tab you can specify the labeling for the printout templates. This option is very useful for users that need to have the labeling for the contour plot printouts in a language other than English. By default, POST View uses the labeling as shown in the fields when the **Default** option is selected. To be able to specify your own label, select the **User Defined** option and specify the appropriate label in each field.

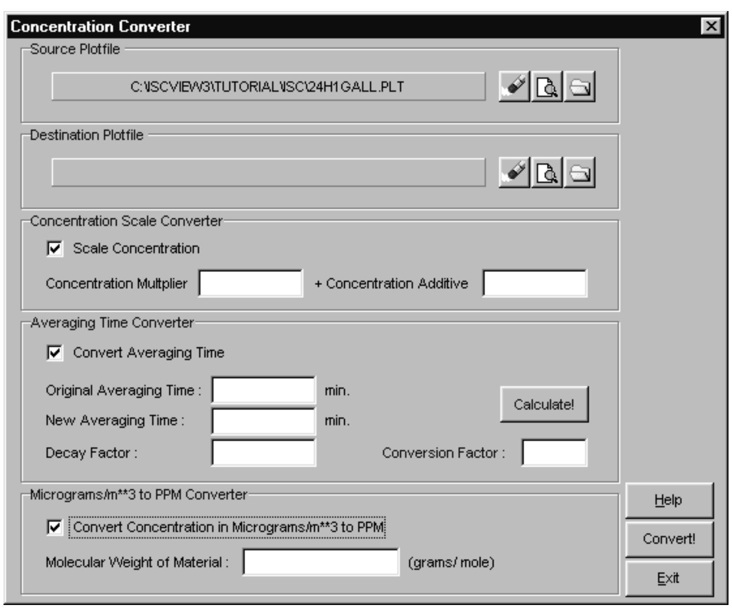


Preferences dialog box – Labeling Options tab

Concentration Converter

Concentration Converter is a utility that allows you to generate new ISC/AERMOD Plotfiles from existing ones without having to re-run the ISCST3, AERMOD, or ISC-PRIME models. You can apply a multiplier or an additive to the concentration values (can be used, for example, when you want to take into account background concentrations), you can change the averaging period, or change the concentration units to PPM.

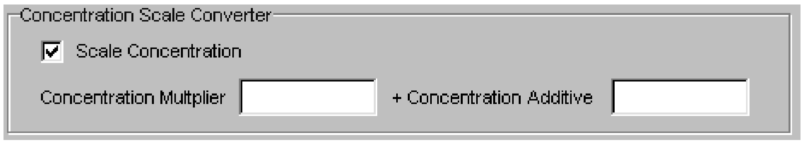
In order to generate a new plotfile using the conversion options present in this utility, you must specify the existing plotfile (**Source Plotfile**) and the filename for the new plotfile (**Destination Plotfile**). You specify these two files by clicking on the **File** button. You can view the contents of the specified plotfile by clicking on the view tool ().



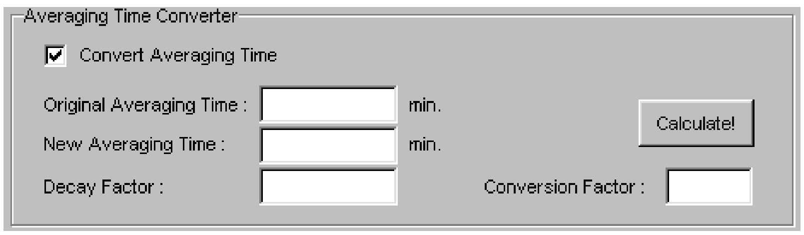
Concentration Converter dialog box

With the **Concentration Converter** utility you can do the following:

1. **Concentration Scale Converter:** This option allows you to specify a multiplier and an additive for the concentration values to generate a new ISC/AERMOD Plotfile. To select this option, you must check the **Scale Concentration** check box and specify a value for the **Multiplier** and **Additive**.



2. **Averaging Time Converter:** This option allows you to specify a different averaging period for your existing ISC/AERMOD Plotfile. To select this option, you must check the **Convert Averaging Time** check box and specify the following:



- **Original Averaging Time (T_{old}):** This is the averaging period for your original ISC/AERMOD Plotfile. The averaging time has to be given in minutes. If your ISC/AERMOD Plotfile was generated using ISC-AERMOD View you will have 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	this will depend on the total number of years of met data

- **New Averaging Time (T_{new}):** This is the averaging period you want your ISC/AERMOD Plotfile to be converted to. The new averaging time must be given in minutes.

- **Decay Factor (q):** The Decay Factor has a value between 0.17 and 0.20 and is used as follows.

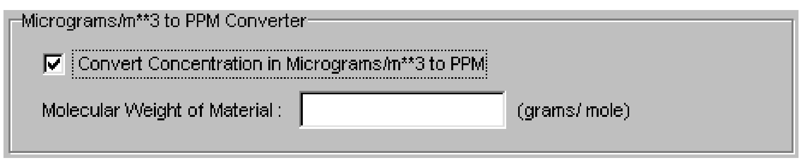
  $C_{new} = C_{old} * (T_{new}/T_{old})^{q}$

  Where:

  C_{new} is the desired concentration
  C_{old} is the original concentration
  T_{new} is the new averaging time
  T_{old} is the original averaging time
  q is the decay factor

After you have specified the above information, you must press the **Calculate!** button in order to get the conversion factor. You can also adjust the Conversion Factor by typing in a new value. The averaging time conversion will then be based on this conversion factor.

3. **Micrograms/m**3 to PPM Converter:** This option allows you to convert your concentration values from micrograms per cubic meter to PPM. To select this option, you must check the **Convert Concentration in Micrograms/m**3 to PPM** check box and specify the molecular weight of the pollutant (chemical).



After you have specified the converting options you must press the **Convert!** button in order to generate the new ISC/AERMOD Plotfile. You can view the contents of the new ISC/AERMOD Plotfile by pressing the view tool ( ) located beside the **Destination Plotfile** panel.

Note: Please note that the **Concentration Converter** utility will work only for **Concentration** values, not for **Deposition** values.

References

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.

U. S. Environmental Protection Agency, 1992. 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, NC.

U. S. Environmental Protection Agency, 1995a. User's Guide for the Industrial Source Complex (ISC3) Dispersion Models (Revised), Volume 1. EPA-454/B-95-003a. Office of Air Quality Planning and Standards, Research Triangle Park, NC.

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.

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.

U. S. Environmental Protection Agency, 1998a. Revised Draft - User's Guide for the AMS/EPA Regulatory Model – AERMOD. Office of Air Quality Planning and Standards, Research Triangle Park, NC.

U. S. Environmental Protection Agency, 1998b. Revised Draft - User's Guide for the AERMOD Terrain Preprocessor (AERMAP). Office of Air Quality Planning and Standards, Research Triangle Park, NC.

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.

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.