HYSPLIT4 USER'S GUIDE INDEX

HYSPLIT_4 USER's GUIDE

TABLE OF CONTENTS

Roland R. Draxler

Version 4.7 - Last Revision May 2004

NOAA Technical Memorandum ERL ARL-230, June 1999, 35 p

ABSTRACT. The HYSPLIT_4 (Hybrid Single-Particle Lagrangian Integrated Trajectory) Model installation, configuration, and operating procedures are reviewed. Examples are given for setting up the model for trajectory and concentration simulations, graphical displays, and creating publication quality illustrations. The model requires specially preformatted meteorological data. Programs that can be used to create the model's meteorological input data are described. The User's Guide has been restructured so that the section titles match the GUI help menu tabs. Although this guide is designed to support the PC and Unix versions of the program, the executable of the on-line web version is identical. The only differences are the options available through the interface.

MODEL OVERVIEW

Features

The HYsplit_4 (HYbrid Single-Particle Lagrangian Integrated Trajectory) model is a complete system for computing trajectories to complex dispersion and deposition simulations using either puff or particle approaches. It consists of a modular library structure with main programs for each primary application: trajectories and air concentrations.

Gridded meteorological data, on one of three conformal (Polar, Lambert, Mercator) map projections, are required at regular time intervals. Versions 4.5 and higher also support computations on a regular latitude-longitude grid. The input data are interpolated to an internal sub-grid centered to reduce memory requirements and increase computational speed. Calculations may be performed sequentially or concurrently on multiple meteorological grids, usually specified from fine to coarse resolution.

Air concentration calculations require the definition of the pollutant's emissions and physical characteristics (if deposition is required). When multiple pollutant species are defined, an emission would consist of one particle or puff associated with each pollutant type. Alternately, the mass associated with a single puff may contain several species. The latter approach is used for calculation of chemical transformations when all the species follow the same transport pathway. Some simple chemical transformation routines are provided with the standard model distribution.

The dispersion of a pollutant is calculated by assuming either a Gaussian or Top-Hat horizontal distribution within a puff or from the dispersal of a fixed number of particles. A single released puff will expand until its size exceeds the meteorological grid cell spacing and then it will split into several puffs. An alternate approach combines both puff and particle methods by assuming a puff distribution in the horizontal and particle dispersion in the vertical direction. The resulting calculation may be started with a single particle. As its horizontal distribution expands beyond the meteorological grid size, it will split into multiple particle-puffs, each with their respective fraction of the pollutant mass. In this way, the greater accuracy of the vertical dispersion parameterization of the particle model is combined with the advantage of having an expanding number of particles represent the pollutant distribution as the spatial coverage of the pollutant increases and therefore a single particle can represent increasingly lower concentrations.

Air concentrations are calculated at a specific grid point for puffs and as cell-average concentrations for particles. A concentration grid is defined by latitude-longitude intersections. Simultaneous multiple grids with different horizontal resolutions and temporal averaging periods can be defined for each simulation. Each pollutant species is summed independently on each grid.

The routine meteorological data fields required for the calculations may be obtained from existing archives or from forecast model outputs already formatted for input to HYSPLIT. In addition, several different pre-processor programs are provided to convert NOAA, NCAR (National Center for Atmospheric Research) re-analysis, or ECMWF (European Centre for Medium-range Weather Forecasts) model output fields to a format compatible for direct input to the model. The model's meteorological data structure is compressed and in "direct-access" format. Each time period within the data file contains an index record that includes grid definitions to locate the spatial domain, check-sums for each record to ensure data integrity, variable identification, and level information. These data files require no conversion between computing platforms.

The modeling system includes a Graphical User Interface (GUI) to set up a trajectory, air concentration, or deposition simulation. The post-processing part of the model package incorporates graphical programs to generate multi-color or black and white publication quality Postscript printer graphics.

A complete description of all the equations and model calculation methods for trajectories and air concentrations has been published and it is also available on-line (). The on-line and the version included with the PC installation contains all the most recent corrections and updates.

Pre-Installation Preparation

There are two installation programs that can be downloaded. The trial version (*setup47U.exe - 15 Mb*) available to anyone and a fully functional version (*setup47R.exe - 25 Mb*) that requires a user registration through the web site. Both versions identical, except the trial version will not work with forecast meteorological data files. In addition, several supplemental programs (Ghostview and Tcl/Tk) are packaged with the trial version for convenience. The self-installing executable contains only HYSPLIT related programs. No additional software is required to run a simulation if the command line interface is sufficient. To enable the model's GUI, the computer should have Tcl/Tk script language installed. The most recent version can be obtained over the Internet from or an older version is packaged with the trial version. The installation of Tcl/Tk will result in the association of the *.tcl* suffix with the *Wish* executable and all Hysplit GUI scripts will then show the Tk icon. The HYSPLIT GUI has been tested with Tcl/Tk version 8.3.2.

The primary HYSPLIT graphical display programs convert the trajectory and concentration model output files to Postscript format. The Postscript files can also be viewed directly through the GUI if Ghostscript and Ghostview have been installed. See for more information on the Postscript file viewer. The HYSPLIT code and GUI have been tested with Ghostscript 6.5 and Ghostview 3.6. Installation of different versions might require editing the main GUI script's directory pointers (edit file: *\guicode\hysplit4.tcl*). Default installation to the \gs and \ghostgum directories is suggested.

The third optional GUI feature is the ability to convert the Postscript output file to a different graphical formats using ImageMagick. More information on this software can be found at The HYSPLIT code and GUI have been tested with ImageMagick 5.3. Installation should be to the *c:\ImageMagick* directory. Installation to a different directory requires editing the main GUI script's directory pointers (edit file : *\guicode\hysplit4.tcl*). Proper functioning of the conversion software, which works in conjunction with Ghostscript, requires that the Ghostscript binary (gswin32.exe) directory be defined in the command line execution path *(c:\gs\gs6.50\bin)* as defined in the *Autoexec.bat* file. Also insure that the correct *delegates.mgk* file for the computer's operating system is defined in the *ImageMagick* directory.

Windows Installation (all operating systems)

HYSPLIT installation to a computer running Windows is provided through a self-installing file. Executables are installed in various directories for trajectories, dispersion, display and manipulation of results, and the creation of input meteorological data files. The trajectory and dispersion model source code is not provided. However all the Fortran source code to create meteorological data files in a format that the model can read are provided in the *\source* directory. Each subdirectory contains a *@readme.txt* file with more complete information about the contents of that directory.

During the installation you will be prompted as to the directory location. It is suggested you select the default location (*C:\hysplit4*). The installation program is very simple, installs all code and executables to your selected directory, and creates a shortcut on the desktop to \guicode\hysplit4.tcl with the "Start In" directory as your selected default. You may have one of two versions of the installation program, setup46R, or setup46U. The suffixes R and U refer to the Registered or Unregistered versions. The two versions are almost identical, except that the unregistered (trial) version does not permit calculations with forecast meteorological data. Further the trial version comes packaged with Tcl/Tk and Ghostscript/Ghostview. If these are already installed on your system, then just "Cancel" that installation step.

The following subdirectories will be created after the installation has completed:

arcview - Contains information about the program ascii2shp that converts a text file in generate format and produces an ArcView Shapefile that can be displayed by ArcExplorer.

bdyfiles - This directory contains an ASCII version of gridded land use, roughness length, and terrain data. The current file resolution is 360 x 180 at 1 degree. The upper left corner starts at 180W 90N. The files are read by both HYSPLIT executables, *hymodelt* (trajectory model) and *hymodelc* (concentration model), from this directory. If not found, the model uses default constant values for land-use and roughness length. The data structure of these files is defined in the file *ASCDATA.CFG*, which should be located in either the model's startup directory or the *\bdyfiles* directory. This file defines the grid system and gives an optional directory location for the land-use and roughness length files. These files may be replaced by higher resolution customized user-created files. However, regardless of their resolution, the model will only apply the data from these files at the same resolution as the input meteorological data grid. More information on the structure of these files can be found in the local *@readme.txt* file.

browser - Contains the Tcl/Tk source code required to run the HTML help browser through the GUI.

csource - Contains the dll files required to run the HELP browser and the Particle Viewer / Editor.

data2arl - Current forecast or archive meteorological data can be obtained from the ARL ftp server*: ftp://gus.arlhq.noaa.gov /pub/archives (or /forecasts)*. Older archive data can be ordered from the NCDC (National Climatic Data Center). However if you have access to your own meteorological data or data formatted as GRIB (Gridded Binary), this directory contains various example decoder programs to convert meteorological data in various formats to the format (ARL packed) that HYSPLIT can read. Sample programs include GRIB decoders for ECMWF model fields, NCAR/NCEP (National Centers for Environmental Prediction) re-analysis data, and NOAA Aviation, ETA, and Regional Spectral Model files. All the required packing and unpacking subroutines can be found in the *\source* subdirectories. Sample compilation scripts for Compaq Visual Fortran 6.6 are in some of the decoder directories. More information on how to run these programs can be found in the "Meteorology" section.

concmdl - The directory contains the HYSPLIT air concentration prediction model scripts and sample control and configuration files. Although the model can be run through the GUI, at times it may be desirable to run the model from the command line (e.g. using automated scripts). The example *Control* file should produce some results for viewing. The sequence of commands would be ..\exec\ *hymodelc* to execute the model, and ..\exec\ *concplot* to create a Postscript file called *concplot.ps*. Command line arguments are required for *concplot*. See *run_conc.bat* for more detailed information. Normally the GUI is used to create the *Control* file for the simulation. If the file is missing, the model will prompt you for input from the keyboard. Your inputs are copied to a file called *Startup*. That file may then be edited and renamed to *Control* for subsequent simulations. *.Conread* and *con2asc* are provided as examples of how to read concentration files for users to develop other applications.

html - Contains all the HELP files in HTML format. These files can be displayed with any browser or interactively through the GUI. The files that are opened in the GUI depend upon the context from which HELP is invoked.

document - This directory contains PDF (Adobe Portable Document Format) versions of the User's Guide (all the HTML help files put together in one document) and other documentation such as ARL-224, the principal ARL Technical Memorandum describing the model and equations. This User's Guide (this document) provides detailed instructions in setting up the model, modifications to the *Control* file to perform various simulations and output interpretation. The *@readme.txt* file contains additional information about compilation, typical CPU times, and a summary of recent model updates.

exec - Is the directory that contains all the executable programs. The GUI looks for all programs in this directory. When running examples from the command prompt in certain directories, the relative path should be included prior to the executable: "..\exec\program.exe"

grads - Contains source code and instructions for programs to convert meteorological data, concentration files, and trajectory files to a grads compatible format. Note that all executables reside in ..\exec.

graphics - There are two types of graphical plotting programs provided in the ..\exec directory. Publication quality graphics can be created using the postscript conversion programs, *concplot* and *trajplot*, which use a Fortran Postscript library created by Kevin Kohler. All graphical routines use the map background file *arlmap* in this directory. The map background file uses a simple ASCII format and contains the world's coastal and political boundaries at relatively coarse resolution. Other higher resolution map background files are available in the *\graphics\mapfiles* directory or from the HYSPLIT download web page. All graphical programs search the startup directory first for *arlmap* before going to *\graphics*, therefore customized maps can be created without changing the HYSPLIT installation structure.

guicode - This directory contains a Tcl/Tk GUI interface source code script for HYSPLIT. The interface is used to set up the input *Control* file as well as run the graphical output display programs. To use the interface you must first install Tcl/Tk. The upper-level Tcl script is called *hysplit4.tcl*, which calls all other Tcl scripts. Executing this script starts the HYSPLIT GUI. The Desktop shortcut as well as the Start Menu options should point to this script. If the installation program did not properly setup the Desktop, you can manually create a shortcut to the script and edit its properties such that the "Start In" directory is *\hysplit4*. You should also select the HYSPLIT icon from the *\icons* directory.

metdata **-** This directory contains the sample meteorological data file: *oct1618.BIN*. It is an extract of the NGM (NOAA's Nested Grid Model) over the US from 0000 UTC 10/16/95 through 0000 UTC of 10/18/95. The file is used for all calculations shown in the User's Guide. Source code and compilation instructions are provided for the simple diagnostics program: chk_file. In addition, several additional executable programs are provided in \exec. These can be used to examine and display the meteorological data files. More information on these programs can be found in the "Meteorology" section.

source - Contains the Fortran source code, and makefiles, to compile and create the subroutine library need for the example source code given in the various other directories.

trajmdl **-** This directory contains scripts, sample control and configuration files, for the HYSPLIT trajectory model (*hymodelt*). Although the model can be run through the GUI, at times it may be desirable to run the model from the command line (e.g. automated scripts). The example *Control* file should produce some results for viewing. The sequence of commands would be ..\exec\ *hymodelt* to execute the model, and ..\exec\ *trajplot* to create a Postscript file called *trajplot.ps*. See *run_traj.bat* for more detailed information. Normally the GUI is used to create the *Control* file for the simulation. If the file is missing, the model will prompt you for input from the keyboard. Your inputs are copied to a file called *Startup*. That file may then be edited and renamed to *Control* for subsequent simulations. A sample tcl script, *Auto_traj.tcl*, is provided as an example of how one might automate multiple trajectory calculations. The script creates the *Control* file and executes the model in a loop, varying specific parameters with each simulation.

utilities - Installation programs for Tcl/Tk, Ghostscript, and Ghostview may be found in this directory for the unregistered distribution. Otherwise this directory will be empty.

vis5d - Source code is supplied to convert Hysplit concentration files output for input to VIS5D.

working - This should the default directory for satellite data and other files manipulated through the particle editor/viewer.

Hysplit4 root directory **-** Contains sample CONTROL files that can be used for initial guidance to set up more complex simulations. These should be loaded into the GUI from the approprate "Retrieve" menu tab. Examples include:

sample_conc - concentration simulation example from users guide

sample_traj - trajectory simulation example from users guide

back_conc - backward dispersion simulation for concentration

back_traj - backward trajectory simulation

volc_conc - particle settling simulation for a volcanic eruption

The "plants.txt" file contains a sample listing of starting locations that can be opened in the GUI to select from a list of previously determined starting locations. This file can easily be customized.

Problems

If Tcl/Tk does not exist on your system or there are other problems with the GUI interface, it is very easy to run the sample cases directly in either the *\trajmdl* or *\concmdl* sub-directories by running the batch file "run_{model}.bat" If the sample simulation works well, then it is only necessary to manually edit the *Control* file to try out different simulation variations. The Control file options are explained in more detail in the individual Trajectory and Concentration Setup sections.

In general, premature termination during the model initialization phase will result in messages to standard output. However after the model has started, fatal, diagnostic, and progress notification messages are written to a file called *Message*. If the model output is not what you expected, first check the *Control* file to determine if the input setup is what is desired, then check the *Message* file for indication of abnormal performance. These files are always written to the model's startup directory - *\Hysplit4* if the model is run from the GUI. At times error messages may be lost in the display buffer after premature termination. In this case the model should be rerun from the command line window for proper display of all standard output messages. The "Ad*vanced*" menu contains a "Special File Display *"* tab that can be used to select the Message file for viewing. Other features of the advanced menu are used to modify the model's configuration file and are explained in more detail in that section. Modifications to these parameters require a complete understanding of the model's design and operation.

GUI OVERVIEW

Although the model can be configured and run manually from the command line, it is usually much quicker to use the GUI. A summary of the main features in each of the primary GUI menus is reviewed below.

Meteorology - Forecast Data

ftp ARL Files - Links to only the most recent forecast data for a variety of different meteorological models available on the ARL FTP server. The data have already processed to a HYSPLIT compatible format.

ftp NCEP AVN - Accesses the global AVN model forecast fields on the NCEP FTP server in their native GRIB format. Files are decoded and interpolated to the grid specified on the user's PC. If the GRIB files have been saved then selecting "skip" as a password permits another grid to be extracted from the data previously downloaded. A running archive can be maintained by only processing the 00 and +03 forecasts every 6 hours.

local ETA Grib - Processes forecast GRIB fields from the NCEP ETA model on the AWIPS 212 grid into ARL format. The GRIB files must already been downloaded to the user's computer. FTP to the server is not supported through this menu tab.

local AVN Grib - Forecast GRIB fields from the NCEP global AVN model may be processed locally on the PC into ARL format. These files must already be present. These are the same files as the FTP menu tab, but the GRIB decoder provides more options. This procedure is designed only to work with forecast data. The last two digits of the file name represent the forecast hour.

Meteorology - Archive Data

ftp ARL Files - Access to a running archive of the last 48 hours of pseudo-analysis for a variety of different meteorological models with data already processed in a HYSPLIT compatible format.

ftp Reanalysis - The NCAR/NCEP reanalysis data archived at NOAA's CDC has been reformatted for HYSPLIT and placed on the ARL ftp server. These data are available from 1948 through 2003.

Local EDAS Grib - Archived GRIB fields from the NCEP ETA or EDAS model on the AWIPS 212 grid may be processed locally into ARL format. These files must already be present on the local computer. FTP is not supported through this menu tab. The procedure is designed only to work with archive data, where the last four digits of the file name represent the day and hour of the analysis.

Local FNL Grib - Archived GRIB fields from the NCEP global AVN or FNL models may be processed locally into ARL format. These files must already be present. FTP is not supported. The procedure is designed only to work with archive data, where the last four digits of the file name represent the day and hour of the analysis.

local ECMWF Grib - Archive GRIB fields from the global model of ECMWF may be processed locally into ARL format. These files must already be present. FTP is not supported. The procedure is designed only to work with archive data, where the last four digits of the file name represent the day

Meteorology Display

Contour Map - A simple postscript contouring program that can be used to view the data fields in any ARL packed meteorological data set. The output is always written to a file called display.ps. The program requires keyboard data entry.

Text Profile - This program is used to extract the meteorological data profile interpolated to a user selected latitude-longitude position. The raw data values at the surface and each level are output on the left while temperature converted to potential temperature and winds rotated from grid orientation to true are shown on the right. Output is written to a file called profile.txt. Requires keyboard data entry.

Grid Domain - Program to generate a map showing the domain of the meteorological data grid with marks at each selected data grid point. Output is written to the file showgrid.ps. Requires keyboard data entry.

Utility Programs

Convert Postscript - Converts any Postscript file to another format using ImageMagick. Enter the name of the Postscsript file and the desired name of the output file, with the file suffix indicating the desired conversion.

GIS to Shapefile - When the GIS output option is selected in selected display programs an ascii text file (in generate format) is created that contains the latitude-longitude values of each contour. This GIS text file can be converted to an ERSI Arc View shapefile. This menu tab is available from three different points. From the meteorology tab the conversion is "line", from the trajectory tab the conversion is "points", and from the concentration tab the conversion is "polygon".

Trajectory

Trajectory Setup - This menu is used to write the CONTROL file that defines all the model simulation parameters. Note that once a simulation has been defined it can be "Saved As" and later "Retrieved." Note that the setup must be "Saved" before proceeding to the "Run Model" step.

Run Standard Model - The selection opens the window for standard output and starts the execution of "hymodelt.exe". Incremental progress is reported on WinNT and later systems, however on Win95/98 the display remains frozen until the calculation has completed (a beep will sound).

Trajectory Display - Display options are available to label the time increment along the trajectory and the amount of white space surrounding the trajectory on the map. The HIRES option limits the domain to just fit the trajectory. Output is written to the file trajplot.ps.

Special Simulations - The Ensemble option invokes a special executable that will run multiple trajectories from the same location, with each trajectory using meteorological data slightly offset from the origin point. The Matrix option simultaneously runs a regular grid of trajectory starting locations.

Concentration

Concentration Setup - The menu is almost identical to the Trajectory Setup menu except for the added tab for "Pollutant, Deposition, and Grids", which is used to customize the concentration computation.

Unlike trajectories, which are terminated at the top of the model domain, pollutant particles reflect at the model top.

Run Standard Model - Similar in function to Trajectory, executes "hymodelc.exe".

Concentration Display - Output is written to concplot.ps. Options on the menu permit concentration contours to be dynamically scaled to the maximum for each map (default), or fixed for all maps based upon the highest value on any one map, or the maximum concentration contour can be entered as the exponent of the power of 10. Further, a multiplier may be applied to all concentrations before contouring. This permits units to be changed for display purposes. Note that the units label as well as the plot title can be defined in a file called labels.cfg is found in the \working directory. Special display programs are available for Matrix and Ensemble simulation outputs.

Utility Programs

Convert to ASCII - Writes an ASCII file for each time period of concentration output with one record for every grid point that has a non-zero concentration value. Each record is identified by the ending date and time of the sample and the latitude and longitude of each grid point. The output file name is constructed from the name of the [input file]_[julian day]_[output hour].

Grid to Station - This program is used to extract the time-series of concentration values for a specific location. That location may be specified by a single latitude-longitude through the GUI or multiple stations may be defined by entering the name of a file that contains an integer station number, latitude, and longitude on each record. The output is written to con2stn.txt which can be read by the time series plotting program to produce the file timeplot.ps. Note that the time series plot is always in integer units and therefore may require the specification of a units multiplication factor.

Advanced Topics

Contains several menus for custom model configurations that can be used to change the nature of the simulation and the way the model interprets various input parameters. These menus would be used to configure matrix, ensemble, simple pollutant transformations, and even special model initialization options.

METEOROLOGY - Help

Meteorological data fields to run the model are usually already available or can be created from routine analysis archives or forecast model outputs. Archive meteorological data to run the model may be ordered from NOAA's National Climatic Data Center for the regional EDAS or global FNL archives. More recent data files can be copied from ARL's server ftp://gus.arlhq.noaa.gov. There are more complete descriptions of the different data sets available on-line. The meteorological data available through the menu system are summarized in the following three sections.

The meteorology menu is divided into four sections: forecast data, analysis data, data conversion, and data display. The forecast section provides access to various forecast data files that may be FTP'd from the ARL server compatible for immediate input for HYSPLIT, or global AVN model GRIB files that can be FTP'd from the NCEP server. These files must first be converted from GRIB, which is accomplished automatically through the GUI. In addition, the forecast menu tab also permits the processing of either ETA or AVN GRIB files that are local to the computer. The GRIB decoding of local files has more menu conversion options than the FTP menu tab.

The analysis menu tab is similarly divided between FTP of files and files local to the computer. The only FTP option for the analysis archives is from the ARL server for data files that are compatible with the ARL forecasts and the NCAR/NCEP reanalysis data, which have already been converted for HYSPLIT applications.

The convert menu gives GRIB data processing options for FNL (the analysis equivalent of AVN), EDAS (the analysis equivalent of ETA), and ECMWF global data files. The local archive processing is fundamentally different that the forecast data, in that each file contains one time period, and the GUI script loops through all the time periods within a fixed number of days to generate one output file.

The display data menu tab provides some simple tools that can be used to examine the data already in ARL packed format. The contour map program will display a contour plot of any variable at any one time in the file. Multiple time periods may be processed. The profile program returns a text file of the profile of meteorological parameters at a pre-selected point. The grid domain menu shows the extent of the meteorological grid with a mark at each point.

More detailed information about the format of the ARL packed data is available. In addition, various library routines and utility programs are provided to manipulate both the GRIB and ARL packed data files.

METEOROLOGY - Forecast Data FTP - ARL server

The ARL web server contains several meteorological model data sets already converted into a HYSPLIT compatible format on the public directories. Although these directories do not have read permission, file access is permitted via FTP if the file name is known. The data file names are "hardwired" into the GUI. The data files are automatically updated on the server when each new forecast cycle becomes available. Only an email address is required for the password to access the server. The "FTP menu" is locked until the FTP process is completed, at which point a message box will appear indicating the file name that was just transferred.

Pre-processed forecast data are available for NOAA's ETA model at 40-km resolution covering North America out to +60 h or +84 h depending upon the forecast cycle. Output from the GFS is available at several different resolutions. The AVN run of the global aviation model, on hemispheric projections (northern and southern) at 191-km, is available out to +72 h. The GFS is also output on its usual global 1-degree latitude-longitude grid out to +96 h, an extended (GFSX) duration out to +180 h, and to a much longer range (GFSLR) at a coarser resolution (2-degree) to about 12 days. The rapid update cycle (RUC) model is updated hourly. Available from the Air Force Weather agency are MM5 forecasts at 15-km and 45-km resolution.

Except for the GFSLR, model output files are updated every 6 h with the most recent forecast data. Note that NOAA NCEP models are run at much higher spatial resolution than what is archived in a HYSPLIT compatible format on the ARL server. If calculations using higher resolution data are required, then the original ETA or AVN GRIB encoded files must be obtained and converted as described in that section's documentation. GRIB decoding can be difficult. Although much of the software is provided, no additional guidance except for the information contained in this report is available.

METEOROLOGY - Forecast Data FTP - ARL ASEAN

The ARL web server contains several meteorological model data sets already converted into a HYSPLIT compatible format on the public directories. Direct access via FTP to these data files is "hardwired" into the GUI. The data files are automatically updated on the server with each new forecast cycle. Only an email address is required for the password to access the server. The "FTP menu" is locked until the FTP process is completed, at which point a message box will appear indicating the file name that was just transferred.

A special extract of the GFS AVN forecast has been created for the Asian region that is of small size for quick transfer. The downloaded file name follows the convention: AVN{month}{day}{cycle}.

METEOROLOGY - Forecast Data FTP - NCEP AVN

Note that the NCEP models are run at much higher spatial resolution than what is archived on the ARL server. If calculations using the higher resolution date are desired, then the original GRIB encoded files must be obtained from the NCEP server and converted to the HYSPLIT compatible format.

The procedure under this menu tab accesses the NCEP server and uses the NOAA Global Aviation Model (AVN) GRIB decoder, avn2arl, to convert the GRIB data files to the HYSPLIT compatible format. The GRIB decoder interpolates global one-degree latitude-longitude data to a conformal map projection. The default HYSPLIT file is a 100 x 100 grid-point domain at 100-km resolution, centered about the point selected by the slider bars on the lower portion of the menu. AVN data consist of one GRIB file per forecast hour (0, +3, out to +72 hours). The size of each GRIB file is about 24Mb. The AVN decoder is run for each downloaded file, which creates a single-time forecast-file called *DATA.AVN*. This file is appended to AVN{YY}{MM}{DD} to create one file that contains all the forecast hours. The size of the final file is about 970 Kb per forecast time.

The menu is divided into two parts. The upper portion is used to select the initialization time and the duration of the forecast. The lower portion is used to set the output directory and center latitude-longitude of the extracted grid. If the "Save input global GRIB files" option is checked, the downloaded GRIB files are saved rather than deleted after each forecast hour is processed. In this way a different regional grid can be extracted from the global files without going through the FTP process again. The GRIB data files can be reprocessed through the "Convert to ARL" menu tab, which has more extensive conversion options available. Note that only the forecast files from the previous 24-h are available from the NCEP server. The server is not supported 24/7. If archive data are required, the file must be created manually be processing only the initialization times and +3 hour forecasts.

METEOROLOGY - Forecast Data FTP - NCEP ETA

Note that the NCEP models are run at much higher spatial resolution than what is archived on the ARL server. If calculations using the higher resolution date are desired, then the original GRIB encoded files must be obtained from the NCEP server and converted to the HYSPLIT compatible format.

The procedure under this FTP menu tab accesses the NCEP anonymous FTP server and transfers the ETA forecast GRIB files on the 40 km AWIPS 212 grid (15 Mb per file) or the 12 km AWIPS 218 grid (100 Mb per file). ETA forecast data consist of one forecast time (every 3 h) per file. The GRIB files are then converted on the same horizontal grid to a HYSPLIT compatible file which requires 3.5 Mb (212 grid) or 50 Mb (218 grid) per time period. Currently the 218 grid data available on NCEP's server do not have all the fields required for conversion to ARL format and hence this feature is disabled.

The menu is used to select the forecast initialization time (cycle), the duration of the forecast, the output directory, and file name. "Save GRIB files" option is checked, the downloaded files are saved rather than deleted after each forecast hour is processed. The downloaded GRIB data can be reprocessed through the "ETA forecast" menu tab. Note that only the forecast files from the previous 24-h are available from the NCEP server. The server is not supported 24/7. If archive data are required, the file can be created manually be processing only the initialization times and +3 hour forecasts with every 6-h initialization cycle. EDAS GRIB files are not available for download.

METEOROLOGY - Analysis Data FTP - ARL Current

The ARL web server contains several meteorological model data sets already converted into a HYSPLIT compatible format on the public directories. Direct access via FTP to these data files is "hardwired" into the GUI. The data files are automatically updated on the server with each new forecast cycle. Only an email address is required for the password to access the server. The "FTP menu" is locked until the FTP process is completed, at which point a message box will appear indicating the file name that was just transferred.

Pre-processed forecast data are available for NOAA's ETA model at 40-km resolution covering North America and the AVN run of the GFS, on hemispheric projections (northern and southern) at 191-km. The archive consists of a series of analysis (0-hour initialization) and short-time (+3 hr) forecast files that have been appended to each other to create a continuous data time series for the previous 48 hours from the most recent available forecast cycle.

Note that NOAA NCEP models are run at much higher spatial resolution than what is archived in a HYSPLIT compatible format on the ARL server. If calculations using higher resolution data are required, then the original EDAS or FNL GRIB files must be obtained and converted as described in the documentation.

METEOROLOGY - Analysis Data FTP - ARL archive

The ARL web server contains several meteorological model data sets already converted into a HYSPLIT compatible format on the public directories. Direct access via FTP to these data files is "hardwired" into the GUI. Only an email address is required for the password to access the server. The "FTP menu" is locked until the FTP process is completed, at which point a message box will appear indicating the file name that was just transferred.

The ARL analysis data archive consists of output from the Global Data Analysis System (GDAS) and the ETA Data Analysis System (EDAS) covering much of North America. Both data archives are available from 1997 in semi-monthly files. The EDAS (every 3-h) was saved at 80-km resolution through 2003, and then 40-km resolution (AWIPS 212 grid) starting in 2004. The GDAS (every 6-h) was saved on a hemispheric projection at 191-km resolution.

From the GUI it is necessary to enter the year, select the month and period (001 for the 1st through the 15th and 002 for the 16th through the end of the month). The file name will be automatically created.

Note that NOAA NCEP models are run at much higher spatial resolution than what is archived in a HYSPLIT compatible format on the ARL server. If calculations using higher resolution data are required, then the original EDAS or FNL GRIB encoded files must be obtained and converted as described in the documentation.

METEOROLOGY- Analysis Data FTP - CDC Reanalysis

Global NCEP/NCAR pressure level reanalysis data archives available on-line from NOAA's CDC (NOAA-CIRES Climate Diagnostics Center, Boulder, Colorado) were reprocessed into the HYSPLIT compatible format and are available on ARL's web site for 1948 through 2003. The global data files are on their native latitude-longitude grid (2.5 degree) and are archived as one file (122 Mb) per month. The GUI menu tab accesses the correct directory with the proper file name using anonymous ftp after the desired year and month have been selected.

The NCEP/NCAR Reanalysis Project is a joint project between the National Centers for Environmental Prediction (NCEP, formerly "NMC") and the National Center for Atmospheric Research (NCAR). The goal of this joint effort is to produce new atmospheric analyses using historical data (1948 onwards) and as well to produce analyses of the current atmospheric state (Climate Data Assimilation System, CDAS). Until recently, the meteorological community has had to use analyses that supported the real-time weather forecasting. More information about the reanalysis project and data can be found at the NCEP and CDC web sites.

A subset of this data (global pressure level only) is available from ARL in a format suitable for transport and dispersion calculations using HYSPLIT through the GUI. Both the global pressure-level data and a sigma-level regional extract over the U.S. are available through the web for on-line calculations using READY by selecting "Reanalysis" in the meteorological data set selection pull-down menu. Each directory contains data files with the following syntax: R{S|P}{YEAR}{MONTH}.{gbl|usa|tbd}, where R indicates "Reanalysis", S or P indicates that the data are on Sigma or Pressure surfaces, YEAR is a four digit year, and MONTH is a two digit month. The file suffix identifies the projection as either the 2.5 degree global latitude-longitude projection (gbl), or a regional conformal map projection (such as over the "usa"). Other regional projections are "to be determined (tbd)" later. The projection details are encoded in the file's index record and are processed by HYSPLIT during trajectory or dispersion computations. Note conformal regional projections (usa) are currently available only from READY's research directory pull-down menu.

METEOROLOGY - Analysis Data FTP - NCEP FNL

Note that the NCEP models are run at much higher spatial resolution than what is archived on the ARL server. If calculations using the higher resolution date are desired, then the original GRIB encoded files must be obtained from the NCEP server and converted to the HYSPLIT compatible format.

The procedure under this menu tab accesses the NCEP server for the GDAS to convert the GFS FNL GRIB data files to the HYSPLIT compatible format. The GRIB decoder (same as the AVN decoder) interpolates global one-degree latitude-longitude data to a conformal map projection. The default HYSPLIT file is a 100 x 100 grid-point domain at 100-km resolution, centered about the point selected by the slider bars on the lower portion of the menu. FNL data consist of one GRIB file per forecast hour (0, +3, +6, and +9 hours). The size of each GRIB file is about 24Mb. The FNL decoder is run for each downloaded file, which creates a single-time forecast-file called *DATA.AVN*. This file is appended to FNL{YY}{MM}{DD} to create one file that contains all the forecast hours. The size of the final file is about 970 Kb per forecast time.

The menu is divided into two parts. The upper portion is used to select the initialization time and the duration of the forecast. The lower portion is used to set the output directory and center latitude-longitude of the extracted grid. If the "Save input global GRIB files" option is checked, the downloaded GRIB files are saved rather than deleted after each forecast hour is processed. In this way a different regional grid can be extracted from the global files without going through the FTP process again. The GRIB data files can be reprocessed through the "Convert to ARL" menu tab, which has more extensive conversion options available. Note that only the forecast files from the previous 24-h are available from the NCEP server. The server is not supported 24/7. A longer-term archive file must be created manually be processing only the initialization times and +3 hour forecasts with each new 6-h forecast cycle.

METEOROLOGY - Convert to ARL - ETA forecast

This menu item is intended to convert existing ETA GRIB files saved on the local computer to the ARL packed data format. The processing program only converts ETA GRIB files that have been interpolated to the AWIPS 212 grid or 218 grids, selected by the resolution button. There are no processing options available. The entire grid is converted to ARL format. The "Set File Name" tab on the GUI is used to select any GRIB file within the forecast series. File names for individual forecasts are then constructed automatically by the script based upon the time interval between files and the forecast duration selected. File names follow the NCEP conventions, such that the base name depends upon the grid (base{HH}.tm00, where HH is the forecast hour).

METEOROLOGY - Convert to ARL - EDAS Archive

This menu item is intended to convert existing EDAS GRIB files to the ARL packed format. The same program is used to convert ETA forecast data as EDAS archive data. EDAS GRIB files must be on AWIPS 212 grid. The data files to process are determined by selecting any file in the GRIB directory. Subsequent file names will be constructed automatically between the starting and ending day at the hour interval set by the radio-button. Day and hour are assumed to be the last four digits of the file name: {base}{DD}{HH}. Time intervals of 12 hours or more should be avoided in transport and dispersion calculations. Only one month of data or less should be processed into one output file. HYSPLIT cannot properly position to the correct starting data record if there is a month transition included within a data file.

METEOROLOGY - Convert to ARL - AVN Forecast

This menu tab is intended to convert existing AVN GRIB forecast files to the ARL packed HYSPLIT compatible format. A similar program is used to convert GRIB data in the FTP menu tab. However, more options are available through this menu.

The base GRIB file is set by selecting one of the GRIB files in the "Set file name" menu tab. Subsequent forecast file names will be constructed automatically following the NCEP convention which assumes file names end with pgrbf{HH}.tm00, where HH is the forecast hour. GRIB conversions are only valid for data on the one-degree global latitude-longitude grid (*pgrb* base name).

Four different grid conversion are available through the menu. "Extract" is identical to option available through the FTP menu, such that the GRIB data are interpolated to a conformal map projection of 100x100 grid points at 100 km resolution at the center of the latitude and longitude selection of the slider bar. The Northern- or Southern Hemisphere options create a 95-km resolution polar stereo-graphic grid centered about the pole. The global check-button does no conformal interpolation, but just converts the original one-degree latitude-longitude data to the ARL packed format.

METEOROLOGY - Convert to ARL - FNL Archive

This menu tab is intended to convert existing one-degree latitude-longitude FNL GRIB data files to ARL format by selecting any file in the GRIB directory. Subsequent file names will be constructed automatically between the starting and ending day at the hour interval set by the radio-button. Day and hour are assumed to be the last four digits of the file name: {base}{DD}{HH}. Time intervals of 12 hours or more should be avoided in transport and dispersion calculations. Only one month of data or less should be processed into one output file. HYSPLIT cannot properly position to the correct starting data record if there is a month transition included within a data file.

Four different grid conversions are available through the menu. "Extract" is identical to option available through the FTP menu, such that the GRIB data are interpolated to a conformal map projection of 100x100 grid points at 100 km resolution at the center of the latitude and longitude selection of the slider bar. The Northern- or Southern Hemisphere options create a 95-km resolution polar stereo-graphic grid centered about the pole. The global options results in no conformal interpolation and just converts the original one-degree latitude-longitude data to the ARL packed format.

METEOROLOGY - Convert to ARL - ECMWF forecast

This menu tab is intended to convert existing ECMWF GRIB files to the ARL packed HYSPLIT compatible format. Subsequent file names will be constructed automatically between the starting and ending day at the hour interval set by the radio-button. Day and hour are assumed to be the last four digits of the file name: {base}{DD}{HH}. Input files could consist of multiple time period forecasts or single-time period archives. Time intervals of 12 hours or more should be avoided in transport and dispersion calculations. Only one month of data or less should be processed into one output file. HYSPLIT cannot properly position to the correct starting data record if there is a month transition included within a data file.

The ECMWF input data may be on pressure surfaces or its native hybrid coordinate system. The "number of levels", counted from the surface upwards, can be used to restrict the size of the output file. This may be particularly useful when the input file consists of 60 or more hybrid levels.

Four different grid conversions are available through the menu. The "Extract" option interpolates the data to a conformal map projection of 100x100 grid points at 100 km resolution at the center of the latitude and longitude selection of the slider bar. The Northern- or Southern Hemisphere options create a 95-km resolution polar stereo-graphic grid centered about the pole. The "Input" option results in no interpolation and the program just converts the original latitude-longitude data (global or regional) to the ARL packed format.

METEOROLOGY - Convert to ARL - ECMWF archive

This menu tab is intended to convert ECMWF reanalysis data archives (ERA-40) GRIB files to the ARL packed HYSPLIT compatible format. Data files must be independently downloaded from the ECMWF web page (http://data.ecmwf.int/data). The user can select which fields and time periods to extract. A GRIB file may contain multiple time periods. No more than one month's data should be included in a GRIB file if it is to be converted to ARL format. All the time periods from one input file will be processed into one output file. The ECMWF download menu permits copying all the data on the global grid or extracting regional sub-grids. The ARL conversion program can handle either option.

The conversion menu is designed to select three different GRIB file types: upper air data, surface data, and time invariant data. The upper air and surface data files should contain data for the same time periods. To be able to run HYSPLIT, the upper air data file must contain at a minimum the following variables: geopotential, temperature, u-velocity, v-velocity, w-velocity, and relative humidity. The surface variable file should contain the 2m temperature, and the 10m u- and v- velocity components. The invariant file should contain the surface geopotential field (terrain height). Terrain height or surface pressure is required for HYSPLIT to be able to interpolate pressure level data to its internal terrain following coordinate system.

The ECMWF input data may be on pressure surfaces or its native hybrid coordinate system. The "number of levels", counted from the surface upwards, can be used to restrict the size of the output file. This may be particularly useful when the input file consists of 60 or more hybrid levels.

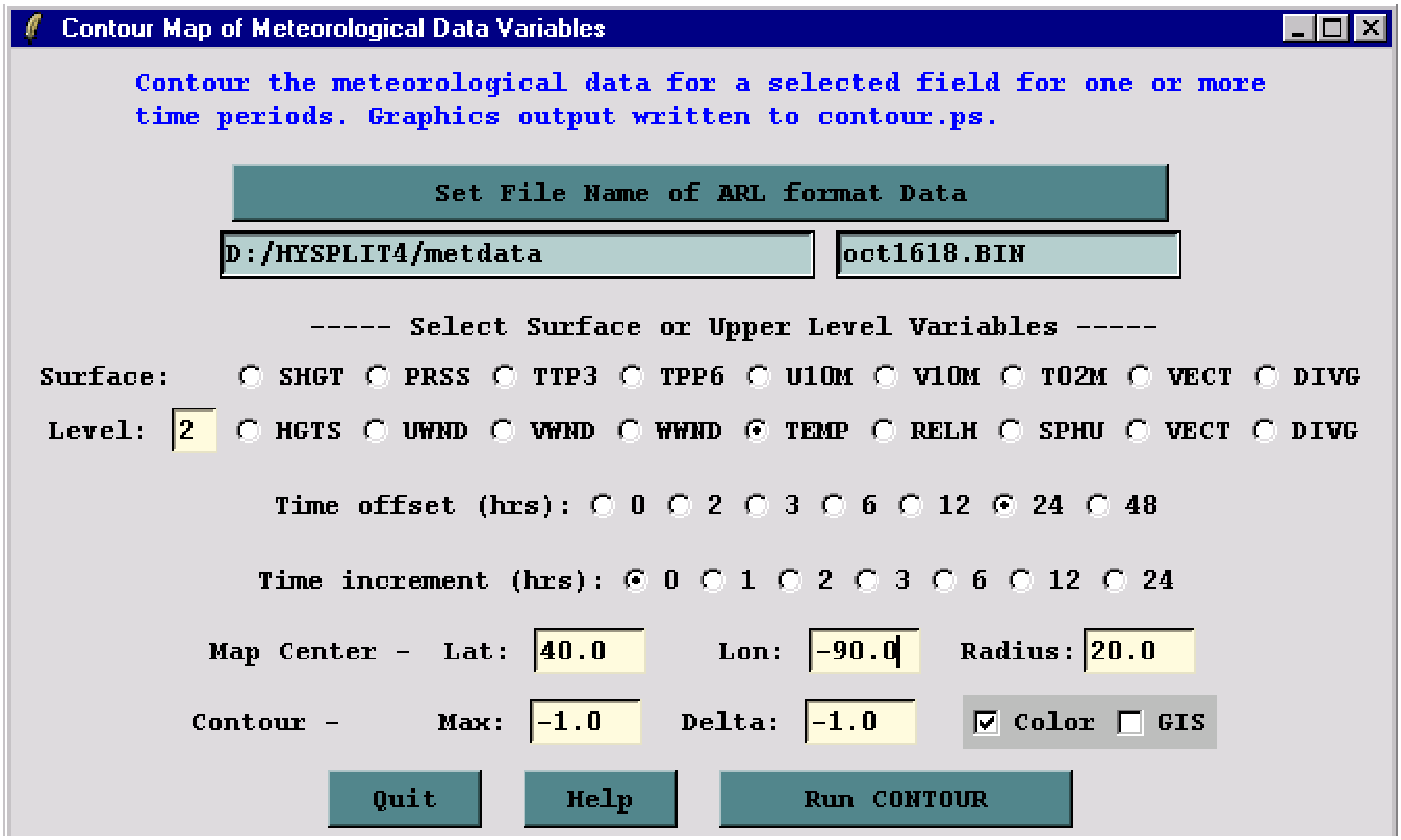
Four different grid conversions are available through the menu. The "Extract" option interpolates the data to a conformal map projection of 100x100 grid points at 100 km resolution at the center of the latitude and longitude selection of the slider bar. The Northern- or Southern Hemisphere options create a 95-km resolution polar stereo-graphic grid centered about the pole. The "Input" option results in no interpolation and the program just converts the original latitude-longitude data (global or regional) to the ARL packed format. The "Input" option must be used for latitude-longitude sub-grids because interpolation from input data sub-grids is not supported.

METEOROLOGY - Convert to ARL - MM5V3

This menu tab is intended to convert MM5 Version 3 output files, usually named MMOUT_DOMAIN{X}, to the ARL packed HYSPLIT compatible format. Input files can have any name. The reformatted output files are named by default to DATA{X}.ARL. Where "X" is the domain number. The output file can be renamed through the menu. Multiple time periods can be contained in one file. Note that the conversion program rounds the output times to the nearest 15 minutes. This is required because when MM5 is run on a multiprocessor system, output times may differ be a few minutes for each output cycle.

METEOROLOGY - Display data - Contour map

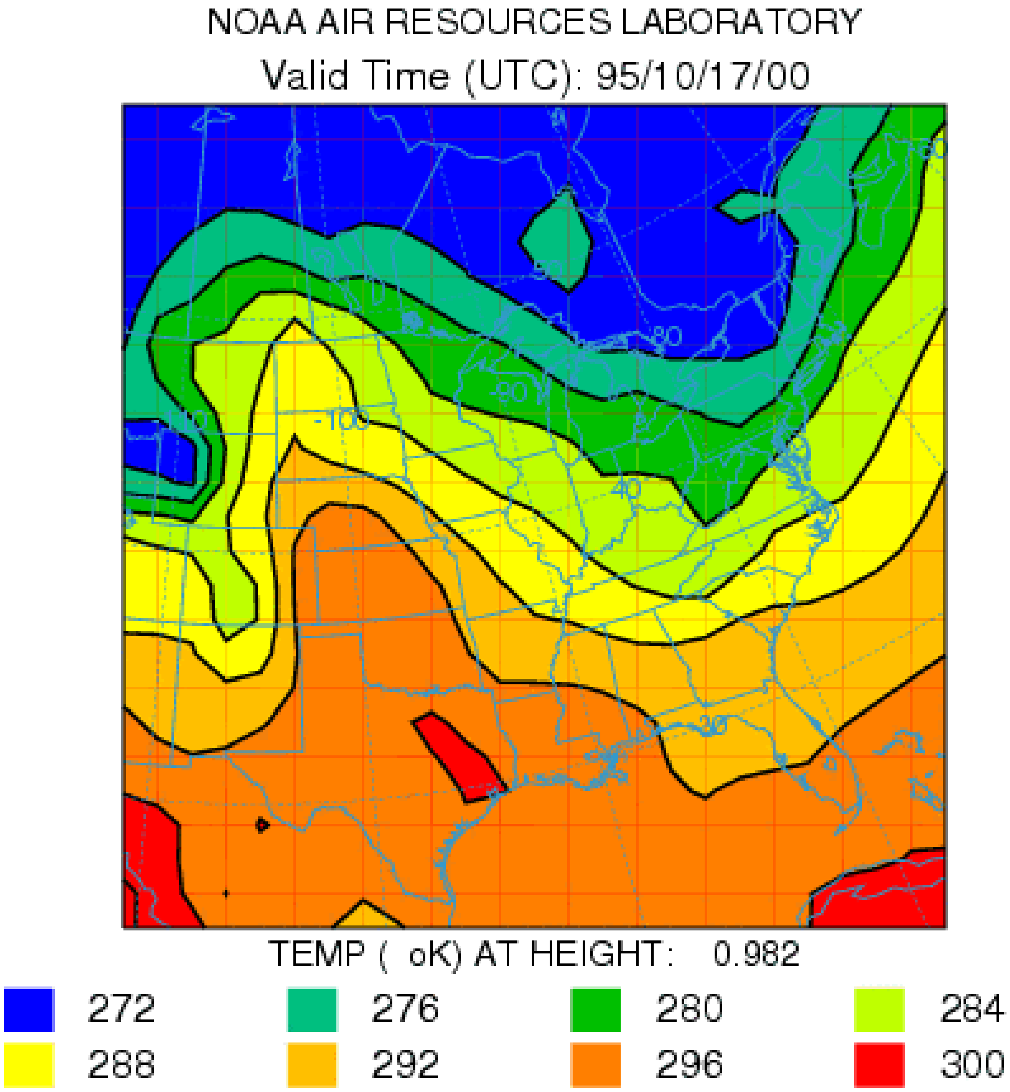
The contour display program, run from this menu, provides a simple tool to create a contour map for any variable at any time period in any ARL packed format meteorological data file. A simple configuration is available through the menu shown below. More options are available if the program is run through the command line.



In the example shown above, the sample meteorological data file has been selected to display the temperature (TEMP) variable on model level #2. The time period to be shown is offset 24 hours from the first time period in the file. No additional time periods are shown when the increment is set to zero. The map will be centered about 40N 90W with a radius of 20 degrees latitude. The maximum contour value and the temperature difference (Delta) between contours will be determined automatically (-1.0).

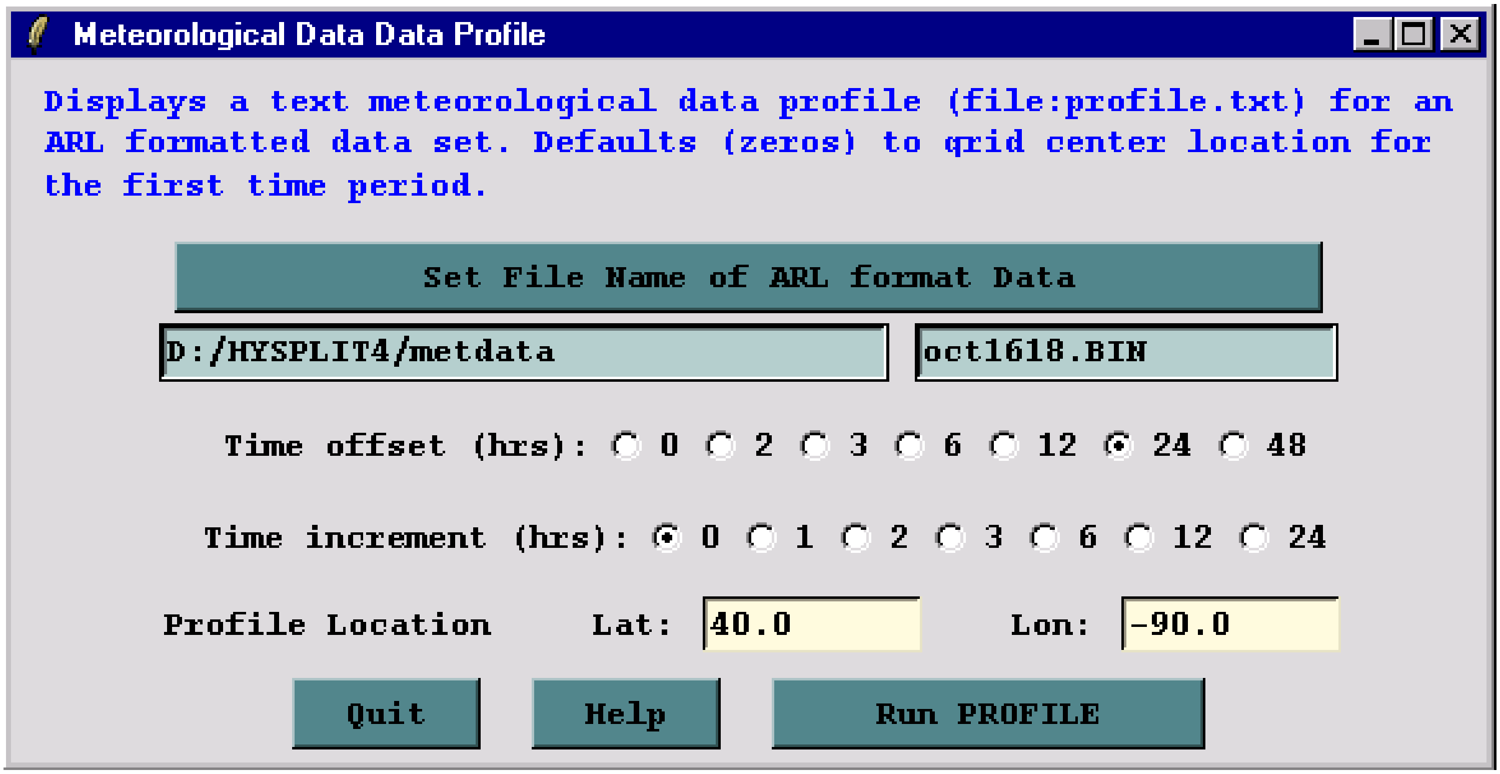
There are two additional derived display options available. Selecting the variable "VECT" will produce a wind vector plot and selecting "DIVG" will produce a plot of the wind field divergence. Both of these options require the U and V velocity fields at the selected level in the order of U followed by V.

The output file *contour.ps* is created in Postscript format. It is shown below, as displayed by the Ghostscript viewer, and may be converted to GIF through the Convert Postscript menu tab.


NOAA AIR RESOURCES LABORATORY
Valid Time (UTC): 95/10/17/00
TEMP (oK) AT HEIGHT: 0.982
272
276
280
284
288
292
296
300

METEOROLOGY - Display Data - Text Profile

The profile program is used to create a profile at a specific location of the meteorological variables in an ARL format packed data file. Output is written to the ASCII text file *profile.txt*. Note that the program does not interpolate to the selected point, but processes the data at the grid point nearest to the location selected. In the example shown below the sample meteorological data file has been selected to display the profile at 40N 90W. The time period to be shown is offset 24 hours from the first time period in the file. No additional time periods are shown when the increment is set to zero.



The output is written to a file called *profile.txt* and the following would be shown in the GUI's display window.

f

METEOROLOGICAL PROFILE LISTING ...
Meteorological Profile: oct1618.BIN
File start time : 95 10 16 0 0
File ending time: 95 10 18 0 0

Profile Time: 95 10 17 0 0
Profile Location: 40.00 -90.00 (21, 13)

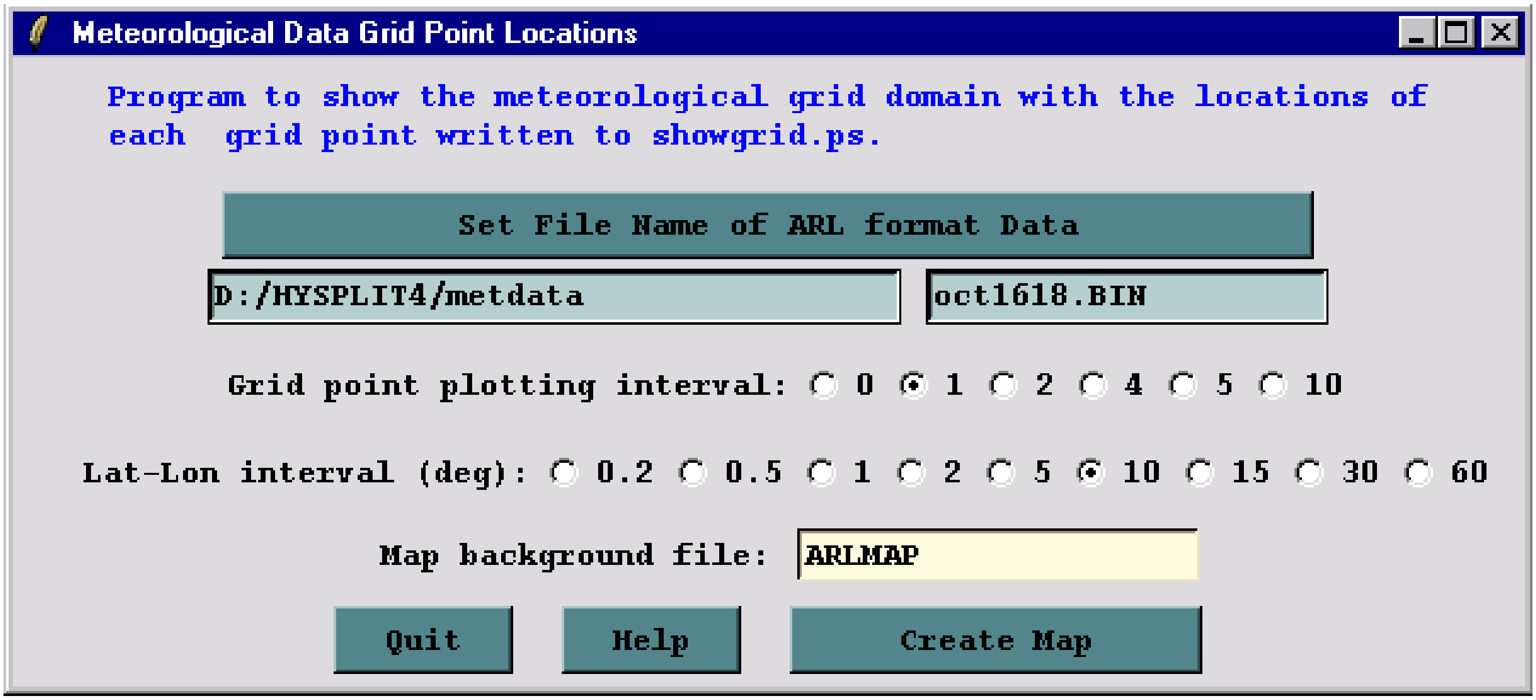
	TPPT	EXCO	HFLX	PRSS	ICWT	SNOW	SHGT	MXLR	WFLX	CPPT	MSLP
	mm			hPa							
998	0	1E-3	-5.6	998	0	0	184	1.00	1E-6	0	1020

	UWND	VWND	WWND	SPHU	TEMP	TPOT	UWND	VWND
	m/s	m/s	mb/h	g/kg	oC	oK	W->E	S->N
981	-1.5	5.0	-2.3	3.9	16.3	291.1	-0.2	5.3
942	-0.42	5.1	-0.19	3.8	13.3	291.5	0.9	5.1
895	2.3	4.4	1.6	2.8	10.6	292.9	3.3	3.6
842	6.7	1.7	1.4	2.7	10.6	298.0	6.9	-0.1
784	11.9	-1.5	0.15	4.3	8.4	301.9	11.1	-4.5
720	16.2	-5.7	0	4.8	5.6	306.2	14.2	-9.7
652	16.2	-8.8	2.5	3.1	0.77	309.6	13.4	-12.7
581	14.9	-8.3	4.3	1.5	-5.9	312.2	12.2	-11.9
508	16.3	-7.7	2.4	0.63	-13.2	315.5	13.7	-11.6
433	21.3	-6.2	-4.2	0.20	-20.0	321.5	21.3	-6.2

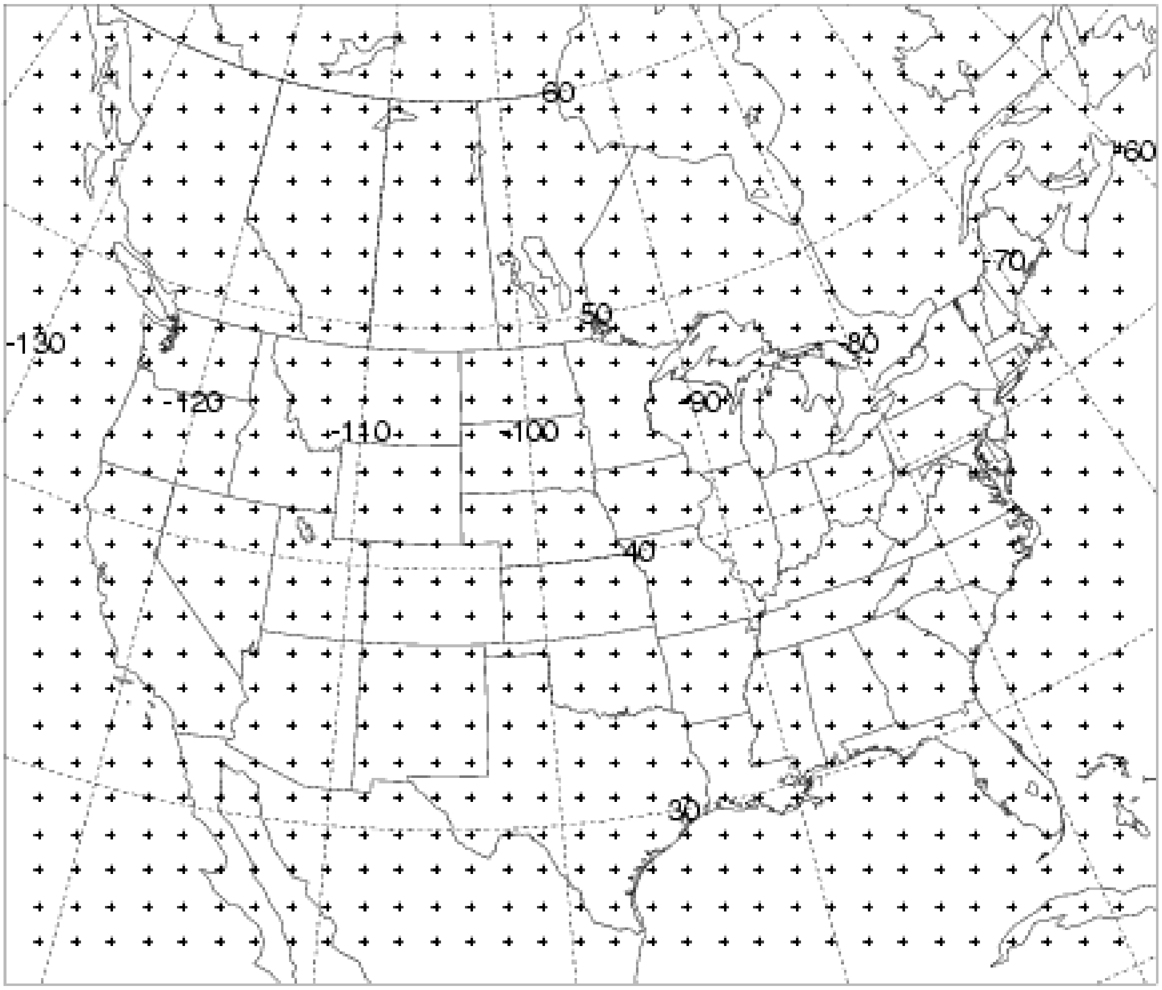
Exit

METEOROLOGY - Display Data - Grid Domain

The *showgrid* program is used to show the domain of the ARL packed format meteorological grid file. Output is to a Postscript file and contains a plus symbol at every grid point intersection as selected by the integer plotting increment. A non-zero value for the Lat-Lon interval draws latitude-longitude lines at that interval over the map background. The example below is shown for the sample meteorological data file.



The following graphic is produced:


60
60
-70
50
-130
-80
-120
-90
-110
-100
40
30

ALL MENUS - Convert Postscript [Files to Another Format]

A utility tab is provided in the meteorology, trajectory, and concentration menu's that will convert the Postscript graphic to another format. The menu shows a "dummy" name that should be replaced with the file name generated by that section's graphic program. All output files are assumed to reside in the \working directory. The file suffix represents the conversion format.

The conversion process uses Ghostscript to read the Postscript file and ImageMagick to convert that file to a variety of other supported graphical formats. These programs can be stand-alone, but they have been linked with the Tcl/Tk GUI menu. Conversion to GIF is the default. Proper installation of the previous 3rd party software is critical in the correct operation of the conversion process. This software is not included with the HYSPLIT installation executable. There are many sources of conversion failures. Make sure that the *Ghostscript* executable (gswin32.exe) is defined in the executable path of the *AUTOEXEC.BAT* file (e.g. c:\gs\gs6.01\bin). Also insure that *ImageMagick* has the proper delegates file defined in the *ImageMagick* directory (e.g. "*copy win95.mgk delegates.mgk*"), where the "from" file name depends upon the operating system. In all situations, if the conversion does not work from the command line, it will not work from the GUI. For instance, once a Postscript file has been created, then "*convert myfile.ps myfile.gif*" should work without error from the command line.

Depending upon the installation directories of *Ghostscript* and *ImageMagick* , for the proper operation of the conversion within the GUI, it may be necessary to edit the upper-level Tcl/Tk file ..*\guicode\hysplit4.tc*l to point to the correct directories. Various versions of these programs will install to different directory names. For instance the lines that might require editing would be similar to the following:

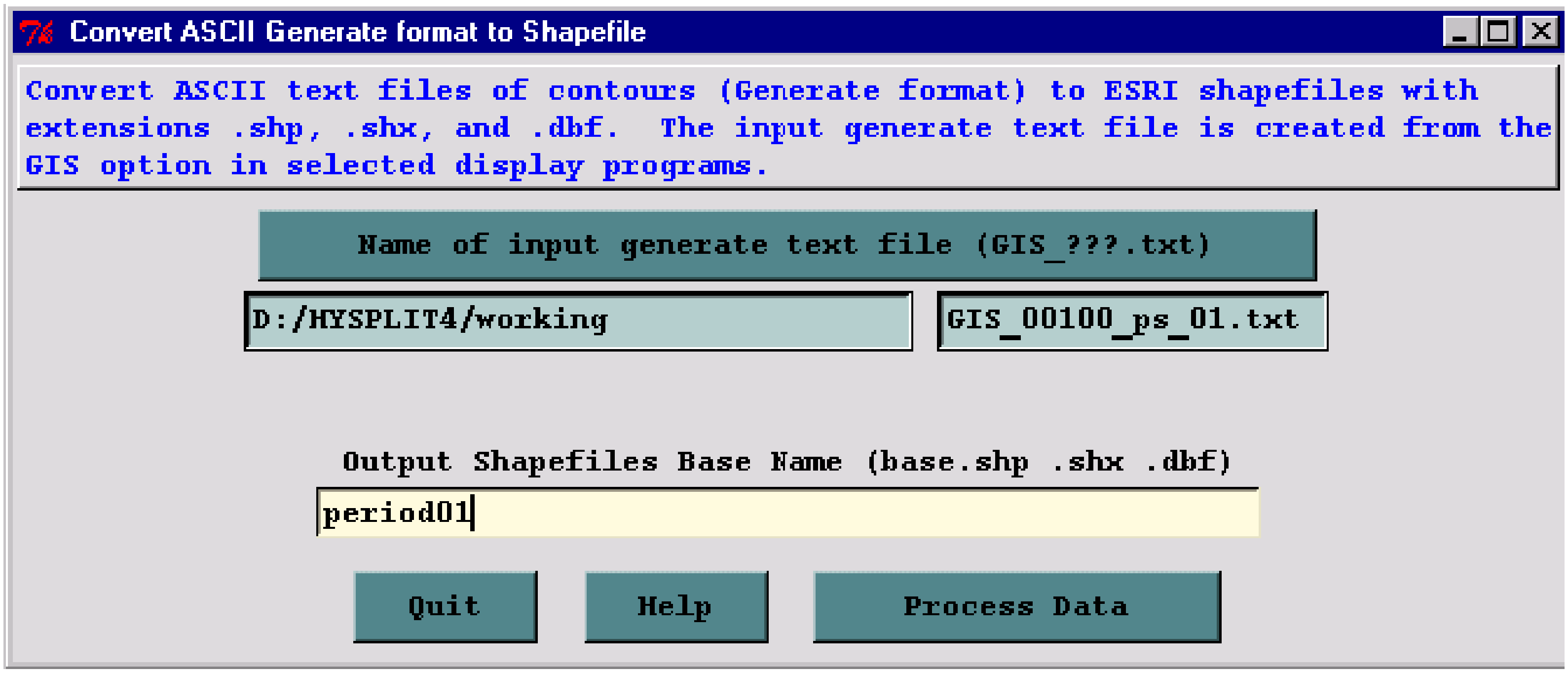
set magick_dir "c:/ImageMagick/convert.exe"

set gsc_dir "c/gs/gs8.00/bin/gswin32.exe"

set gsv_dir "c:/Ghostgum/gsview/gsview32.exe"

ALL Utility Menus – GIS to Shapefile Conversion

A utility tab is provided in the meteorology, trajectory, and concentration menu's that will convert an ESRI generate format text file to a Shapefile for import into ArcView or comparable GIS applications. The "ascii2shp" conversion software is available under the GNU license from the Free Software foundation. The conversion can be run from the command line or the GUI menu shown below.



Normally the fields will be blank when the menu is first invoked. Prior to creating the shapefiles it is necessary to create the "*Generate*" text file which contains the latitude-longitude points of each contour as created by the contouring programs available through the display menus. These files are created by checking the GIS box in the menu (-a1 option on the command line). All generate format files start with *GIS* and end with *.txt*. The remainder of the file name depends upon the application from which it was created. However in all applications, the two digit string prior to the *.txt* identifies the frame number. There is one GIS output file per frame. In the lower text entry box enter the base name of the output shapefiles. All output files will be created in the \working directory.

What is a Shapefile?

If you don't know, you probably don't need this library. The Shapefile format is a new working and interchange format promulgated by ESRI for simple vector data with attributes. It is apparently the only file format that can be edited in ARCView 2/3, and can also be exported and imported in Arc/Info. An excellent white paper on the shapefile format is available from ESRI, but it is .pdf format, so you will need Adobe Acrobat to browse it. The file format actually consists of three files.

```
XXX.shp - holds the actual vertices.
XXX.shx - hold index data pointing to the structures in the .shp file.
XXX.dbf - holds the attributes in xBase (dBase) format.
```

Release Notes

To get notification of new releases of Shapelib *subscribe* to the project at www.freshmeat.net. This is currently the only reliable way of finding out about new releases since there is no shapelib specific mailing list.

Release 1.2.9: Good support for reading and writing NULL fields in .dbf files, good support for NULL shapes and addition of the DBFGetFieldIndex() functions (all contributed by Jim Matthews). Bill Miller has contributed an upgraded shputils.c. Daniel Morissette contributed DBFGetNativeFieldType(). Better error checking for disk errors in dbfopen.c. Various other bug fixes and safety improvements.

Release 1.2.8: Added hacked libtool support (supplied by Jan) and "rpm ready" install logic.

Release 1.2.7: Fix record size (was 4 bytes too long). Modify SHPReadObject() to handle null shapes properly. Use atof() instead of sscanf(). Support .DBF as well as .dbf.

Release 1.2.6: Now available under old MIT style license, or at the user's option, LGPL. Added the contrib directory of stuff from Carl Anderson and the shptree.c API for quadtree based spatial searches.

Release 1.2.5: SHPOpen() now forcibly uses "rb" or "r+b" access string to avoid common mistakes on Windows. Also fixed a serious bug with .dbf files with a 'F' field type.

Release 1.2.4: DBFOpen() will now automatically translate a .shp extension to .dbf for convenience. SHPOpen() will try datasets with lower and uppercase extension. DBFAddField() now returns the field number, not TRUE/FALSE.

Release 1.2.3: Disable writing measures to multi-patches as ArcView seems to puke on them (as reported by Monika Sester). Add white space trimming, and string/numeric attribute interchangability in DBF API as suggested by Steve Lime. Dbfdump was updated to include several reporting options.

Release 1.2.2: Added proper support for multipatch (reading and writing) - this release just for testing purposes.

Release 1.2 is mostly a rewrite of the .shp/.shx access API to account for ArcView 3.x 3D shapes, and to encapsulate the shapes in a structure. Existing code using the shapefile library will require substantial changes to use release 1.2.

Release V1.1 has been built on a number of platforms, and used by a number of people successfully. V1.1 is the first release with the xBase API documentation.

Maintainer

This library is maintained by me (Frank Warmerdam) on my own time. Please send me bug patches and suggestions for the library. Email can be sent to warmerdam@pobox.com.

The current status of the Shapelib code can be found somewhere off my home page at http://pobox.com/~warmerdam.

The shputils.c module was contributed by Bill Miller (NC-DOT) who can be reached at bmiller@doh.dot.state.nc.us. I had to modify it substantially to work with the 1.2 API, and I am not sure that it works as well as it did when it was originally provided by Bill.

Credits

I didn't start this section anywhere near soon enough, so a lot of earlier contributors to Shapelib are lost in pre-history.

- Bill Miller (NY-DOT) for shputils.c
- Carl Anderson for the contents of the contrib directory, and the "tuple" additions to dbfopen.c.
- Andrea Giacomelli for patches for dbfopen.c.
- Doug Matthews for portability improvements.
- Jan-Oliver Wagner for convincing me to make it available under LGPL, shared library support, and various other patches.
- Dennis Christopher (of Avenza) for testing and bug fixes.
- Miko Syrjä (of 3D-system Oy) for a record size bug fix.
- Steven Lime and Curtis Hill for help with NULL shapes.
- Jim Matthews for support of NULL attributes in dbf files.

In Memorium

I would like to dedicate Shapelib to the memory of Sol Katz. While I never met him in person, his generous contributions to the GIS community took many forms, including free distribution of a variety of GIS translators with source. The fact that he used this Shapelib in some of his utilities, and thanked me was a great encouragement to me. I hope I can do his memory honour by trying to contribute in a similar fashion.

Portability

The Shapefile C Library should port easily to 32bit systems with ANSI C compilers. It should work on 64 bit architectures (such as the DEC AXP). Care should also be taken to pass the binary access flag into SHPOpen() and DBFOpen() when operating on systems with special text file translation such as MSDOS. The shputils.c module is contributed, and may not take the same approach to portability as the rest of the package. On Linux, and most unix systems it should be possible to build and install shapefile support as a shared library using the "lib" and "lib_install" targets of the Makefile. Note that this Makefile doesn't use autoconf mechanisms and will generally require some hand tailoring for your environment.

Limitations

- You can't modify the vertices of existing structures (though you can update the attributes of existing structures, and create new structures).
- Not written in such a way as to be particularly fast. This is particularly true of the 1.2 API. For applications more concerned with speed it may be worth using the V1.1 API.

- Doesn't set the last access time properly in the .dbf files.
- There is no way to synchronize information to the file except to close it.
- Poor error checking and reporting.
- Not professionally supported (well it can be, if you want to pay).
- Some aspects of xBase files not supported, though I believe they are not used by ESRI.
- The application must keep the .dbf file in sync with the .shp/.shx files through appropriate use of the DBF and SHP APIs.
- No support for the undocumented .sbn/.sbx spatial index files.

Copyright

The source for the Shapefile C Library is (c) 1998 Frank Warmerdam, and released under the following conditions. The intent is that anyone can do anything with the code, but that I do not assume any liability, nor express any warranty for this code.

As of Shapelib 1.2.6 the core portions of the library are made available under two possible licenses. The licensee can choose to use the code under either the Library GNU Public License (LGPL) described in LICENSE.LGPL or under the following MIT style license. Any files in the Shapelib distribution without explicit copyright license terms (such as this documentation, the Makefile and so forth) should be considered to have the following licensing terms. Some auxiliary portions of Shapelib, notably some of the components in the contrib directory come under slightly different license restrictions. Check the source files that you are actually using for conditions.

Default License Terms

Shapelib Modifications

I am pleased to receive bug fixes, and improvements for Shapelib. Unless the submissions indicate otherwise I will assume that changes submitted to me remain under the above "dual license" terms. If changes are made to the library with the intention that those changes should be protected by the LGPL then I should be informed upon submission. Note that I will not generally incorporate changes into the core of Shapelib that are protected under the LGPL as this would effectively limit the whole file and distribution to LGPL terms.

Opting for LGPL

For licensee's opting to use Shapelib under LGPL as opposed to the MIT Style license above, and wishing to redistribute the software based on Shapelib, I would ask that all "dual license" modules be updated to indicate that only the LGPL (and not the MIT Style license) applies. This action represents opting for the LGPL, and thereafter LGPL terms apply to any redistribution and modification of the affected modules.

METEOROLOGY - Help

Data Format

The following sections describe the ARL packed data format in a little more detail to permit the development of customized applications. Library routines are provided to simplify the task of creating a model compatible data file. A meteorological data file is composed of one or more time periods. Each time period begins with one or more ASCII index records that summarize the valid time, the grid definition, the variables, and level information. Each subsequent record contains one horizontal data field, consisting of 50 ASCII bytes of time, variable, and level information for that record, followed by X times Y bytes of data, where X and Y are the number of data points in the horizontal and vertical directions, respectively. Floating point or integer data are packed as one byte per variable. Precision is maintained by packing the differences between adjacent grid points rather than packing the absolute values. In any time period, although not required, the surface data precede the upper-level data fields. All records are of the same length to permit the model to read the file in a "direct access" mode. Data files can be read on most computing platforms without any transformation and appended to each other using routine operating system commands such as "cat" or "type". Only binary transfers or copies are permitted. All the routines discussed in this section can be found in the *\metdata\source* directory.

Valid Meteorological Data Types

Meteorological variables are identified to the model by a unique four-character identification field that is written to the first 50-byte header portion of each data record. Some of the variables that can be decoded by the model and their units are identified below.

Surface-Level Parameters Description Units Identification

Pressure at surface hPa PRSS

Pressure reduced to mean sea level hPa MSLP

Temperature at surface K TMPS

Total precipitation (6-h accumulation) m TPP6

Momentum flux, u-component at surface N/m^2 UMOF

Momentum flux, v-component at surface N/m^2 VMOF

Sensible heat net flux at surface W/m^2 SHTF

Latent heat net flux at surface W/m^2 LHTF

Sfc downward short wave radiation flux W/m^2 DSWF

Temperature at 2 m AGL K T02M

Relative humidity at 2 m AGL % RH2M

U-component of wind at 10 m AGL m/s U10M

V-component of wind at 10 m AGL m/s V10M

Total cloud cover, entire atmosphere % TCLD

Upper-level Parameters Description Units Identification

U-component of wind with respect to grid m/s UWND

V-component of wind with respect to grid m/s VWND

Geopotential height gpm* HGTS

Temperature K TEMP

Pressure vertical velocity hPa/s WWND

Relative humidity % RELH

Data may be obtained from any source, however there are certain minimum data requirements to run the model: surface pressure or terrain height, u-v wind components, temperature, and moisture. In addition, precipitation is required for wet removal calculations. Not required, but certainly necessary to improve vertical mixing calculations, would be some measure of low-level stability. This may take the form of a near-surface wind and temperature or the fluxes of heat and momentum.

It is also important to have sufficient vertical resolution in the meteorological data. Some of the NOAA higher resolution data files have five or more levels in the boundary layer (<850 hPa) in addition to wind and temperatures near the surface, usually at 2 and 10 m agl. These surface values are especially important when the data are only available on absolute pressure surfaces, rather than terrain following sigma surfaces, to avoid interpolation to the surface between data levels when the local terrain is well above sea-level.

Creation of a Meteorological Input Data File

One may prepare meteorological data from any number of different sources to be in a suitable format for the model using a series of routines described in this section. In general it is assumed one has access to a meteorological data source, either the data fields are already on a grid, such as those output from a meteorological model, or perhaps from observations that have been interpolated to a grid. Some example conversion programs are provided within the GUI to convert from either NOAA, or ECMWF GRIB format data files to HYSPLIT format. Other programs are also provided (see *avn2arl*, *ncr2arl*, *ecm2arl*, *and mm5toarl*I) in directory /data2arl. The directory also contains source code for reading the direct access of variable record length files, such as those that contain GRIB records, and special platform independent GRIB decoding subroutines.

The meteorological data are processed in time-sequence, calling the subroutines provided, to create a HYSPLIT compatible output file. These subroutines will pack the data and write the index record. The index record, which precedes the data records for each time period, can only be written after the data records are processed. The packing routines must first be initialized by setting the appropriate grid parameters and defining all the meteorological variables that will be written to the file. Multiple output grids may be defined and written simultaneously by invoking the *PAKSET* routine with a different unit number for each grid. The grid parameters are all defined in a configuration file, which should be in the directory from which the procedure is invoked:

CALL PAKSET (kunit,fname,krec,nx,ny,nz)

Kunit is the Fortran unit number to which the data records will be written. *Fname* is the character string of the name of the configuration file. It can be any name, but it will default to *metdata.cfg*. The file is opened internally on *kunit* to read the configuration file. This routine needs to be called once for each grid. *Krec* is the starting record number (of the index record) to which output will be written. It is normally set to 1 unless you want to start writing in the middle of a file. The remaining parameters (*nx, ny, nz*) are returned by the subroutine and define the horizontal and vertical grid dimensions. These values can be used to set variable dimensions. It is your responsibility to open *kunit* for output after having completed the *pakset* calls:

OPEN(file=myfile, unit=kunit, form='unformatted', access='direct', recl={50+nx*ny})

The individual data records are packed and written by a call to *PAKREC*, once for each variable at each level. The routine calculates the record offset from the index record according to the variable and level information provided in the arguments and writes the record according to the order specified in *metdata.cfg*. The data can be supplied in any order. Note that although the level indicator (LL) goes from 1 to the number of levels, one is subtracted before it is written to the 50 byte header to be consistent with the definition of surface data to be at level "0". All the records in a time period may be initialized according to the value of the *kini* flag. Initialization fills in the time variable for all records and assigns the variable identification field as *null*.

CALL PAKREC (kunit,rvar,cvar,nx,ny,nxy,kvar,iy,

im,id,ih,mn,ic,ll,kini)

kunit - integer unit number of the defined file

rvar - input array of real*4 data values

cvar - character*1 array of packed data values

nx,ny - integer horizontal grid dimensions

nxy - integer product nx*ny and length of cvar

kvar - character*4 descriptor of variable being written

iy,im - integer year and month

id,ih - integer day and hour

mn - integer minutes (usually 0)

ic - integer forecast hour (hours from initialization)

ll - integer level indicator (1 to NZ)

kini - integer initialization flag (0-no; 1-yes)

When all the data records for a time period have been written, it is necessary to close that time period by writing its index record:

CALL PAKNDX(kunit)

At this point your program can return to *PAKREC* if data records for additional time periods are to be added to the file.

The key to the process is creating the proper configuration file for the data set you want to create. Some of the sample data decoders provided in the *data2arl* directory dynamically configure the packing configuration file based upon the command line input information. A sample *metdata.cfg* file for NOAA's AVN model output created dynamically by the *avn2arl* decoder that is automatically invoked through the GUI using the "FTP NCEP AVN " tab of the *Meteorology* menu is shown below. An extract of the global one-degree latitude-longitude GRIB AVN model data have been interpolated to a 100 km resolution Lambert Conformal projection 100x100 point grid centered about 45N 90W. The configuration file format is such that the first 20 characters are a dummy identification field followed by the data.

Example Meteorological Packing Configuration File

20X,Format Column_1 ID Column_21 DATA

A4 Data Source: AVNX

I4 Grid Number: 99

I4 Z-Coordinate: 2

F10 Pole Latitude: 45.0

... Pole Longitude: -90.0

... Reference Latitude: 45.0

... Reference Longitude: -90.0

... Reference grid size: 100.0

... Orientation: 0.0

... Tangent Latitude: 45.0

... Synch point X: 50.5

... Synch point Y: 50.5

... Synch Latitude: 45.0

... Synch Longitude: -90.0

... Reserved: 0.0

I4 Number X points: 100

I4 Number Y points: 100

I4 Number of Levels: 16

F6.,I3,(1X,A4)

Level 1: .0000 6 PRSS MSLP TPP6 U10M V10M T02M

... Level 2: 1000. 6 HGTS TEMP UWND VWND WWND RELH

... Level 3: 975. 6 HGTS TEMP UWND VWND WWND RELH

... Level 4: 950. 6 HGTS TEMP UWND VWND WWND RELH

... Level 5: 925. 6 HGTS TEMP UWND VWND WWND RELH

... Level 6: 900. 6 HGTS TEMP UWND VWND WWND RELH

... Level 7: 850. 6 HGTS TEMP UWND VWND WWND RELH

... Level 8: 800. 6 HGTS TEMP UWND VWND WWND RELH

... Level 9: 750. 6 HGTS TEMP UWND VWND WWND RELH

... Level 10: 700. 6 HGTS TEMP UWND VWND WWND RELH

... Level 11: 600. 6 HGTS TEMP UWND VWND WWND RELH

... Level 12: 500. 6 HGTS TEMP UWND VWND WWND RELH

... Level 13: 400. 6 HGTS TEMP UWND VWND WWND RELH

... Level 14: 300. 6 HGTS TEMP UWND VWND WWND RELH

... Level 15: 200. 6 HGTS TEMP UWND VWND WWND RELH

... Level 16: 100. 6 HGTS TEMP UWND VWND WWND RELH

It is important that the information contained in this file is correct because it not only controls the writing of the packed meteorological data file, but much of the information is written into the index record of each time period. The model decodes this information to set up the internal processing of the meteorological data. Starting with HYSPLIT V4.5, the model is also capable of using meteorological data on a latitude-longitude grid. Previous versions were limited to data on a conformal map projection. Data on a regular latitude-longitude grid still need to be converted to the ARL packed format. Modifications to the packing configuration file required to support a latitude-longitude grid are noted below. A complete description of *metdata.cfg* format follows:

Description of the Meteorological Packing Configuration File

Record 1 consists of a four-character string that identifies the source of the meteorological data. This string will be passed through to many of the output graphics.

Record 2 is an optional integer identification of the meteorological data grid. It was used extensively in previous meteorological data file formats. It is not used in Hysplit4 applications.

Record 3 is an integer number that identifies the vertical coordinate system. Only four coordinate types are recognized: 1-pressure sigma; 2-pressure absolute; 3-terrain sigma; 4-hybrid sigma.

Records 4 & 5 identifies the pole position of the grid projection. Most projections will either be defined at +90 or -90 depending upon the hemisphere. The longitude would be the point 180 degrees from which the projection is cut. Lat-Lon Grids only: contains the latitude and longitude of the grid point with the maximum grid point value.

Records 6 & 7 is the reference position at which the grid spacing is defined. Lat-Lon Grids only: contains the grid spacing in degrees latitude and longitude.

Record 8 is the grid spacing in km at the reference position. Lat-Lon Grids only: a value of zero signals that the grid is a lat-lon grid.

Record 9 is the grid orientation or the angle at the reference point made by the y-axis and the local direction of north. Lat-Lon Grids only: value always = 0.

Record 10 is the angle between the axis and the surface of the cone. For regular projections it is equal to the latitude at which the grid is tangent to the earth's surface. A polar stereo-graphic grid would be tangent at either 90 or -90, while a Mercator projection is tangent at 0 latitude. A Lambert Conformal projection would be in between the two limits. An oblique stereo-graphic projection would have a cone angle of 90. Lat-Lon Grids only: value always = 0

Records 11 & 12 are used to equate a position on the grid with a position on the earth as given in the following two records:

Records 13 & 14. In this example, the position indicated is the center of the grid located over the North Pole. Any position is acceptable. It need not even be on the grid. Lat-Lon Grids only: contains the latitude and longitude of the 1,1 position grid point.

Record 15 is not currently used.

Records 16 & 17 identify the number of grid points in each direction.

Record 18 is the number of levels in the vertical, including the surface level.

Record 19, through the number of levels, identifies the height of each level in appropriate units according the definition of the vertical coordinate, the number of variables at that level, and the four character identification string for each variable. The height units are as follows for each coordinate:

1-sigma (fraction)

2-pressure (mb)

3-terrain (fraction)

4-hybrid (mb: offset.fraction)

Decoding Meteorological Data Files

One may want to develop other applications for HYSPLIT compatible meteorological data files. For these situations, some lower-level routines are provided in the source code library. The key to reading the meteorological files is decoding the index record. The format for this record is given below. Complete descriptions are similar to the variables in the discussion above.

FORMAT DESCRIPTION

A4 Data Source

I3 Forecast hour (>99 the header forecast hr = 99)

I2 Minutes associated with data time

12F7. 1) Pole Lat, 2) Pole Lon, 3) Tangent Lat, 4) Tangent Lon,

5) Grid Size, 6) Orientation, 7) Cone Angle, 8) X-Synch point,

9) Y-Synch point, 10) Synch point lat, 11) Synch point long,

12) Reserved

3I3 1) Number x points, 2) Number y points, 3) Number levels

I2 Vertical coordinate system flag

I4 Length in bytes of the index record, excluding the first 50 bytes

LOOP through the number of data levels

F6. height of the first level

I2 number of variables at that level

LOOP through the number of variables

A4 variable identification

I3 rotating checksum of the packed data

1X Reserved space for future use

Once the index record has been read and decoded you have sufficient information to read and decode the data records. A data un-packer is provided to convert the packed character*1 array to a real*4 array. It can also be used to extract a sub-grid from the full domain through specification of the sub-grid lower left corner:

CALL PAKINP (rvar,cvar,nx,ny,nx0,ny0,lx,ly,prec,nexp,var1,ksum)

rvar - real output array of integer dimensions lx,ly

cvar - character*1 packed input array of length nx*ny

nx,ny- integer dimensions of the full grid

nx0 - integer sub-grid position of left edge in nx units

ny0 - integer sub-grid position of lower edge in ny units lx - integer first dimension of sub-grid length

ly - integer second dimension of sub-grid length

prec - real precision of packed data array

nexp - integer scaling exponent of packed data array

var1 - real value of array at position (1,1)

ksum - integer rotating checksum of packed data array

If the entire grid is to be unpacked then *nx0=ny0=1* and *nx=lx*, *ny=ly*. The checksum (*ksum*) that is returned should be compared with the corresponding value in a table generated from reading the index record. If you are not going to compare the checksum, set *ksum = -1*, this will save a little computer time. Due to the sub-grid option the checksum cannot be computed in the regular unpacking loop, but requires a second pass through the data. The checksum pass is enabled when *ksum=0*. It will then return a non-zero value. If you don't reset it to zero, no further checksums will be computed.

If you want to create your own packed data by converting a real*4 array to the character*1 packed data array use the following:

CALL PAKOUT(rvar,cvar,nx,ny,nxy,prec,nexp,var1,ksum)

Although the structure of the packed data may seem complex, unpacking is a very simple process, the basic elements of which are summarized in the Fortran code shown below. The value of each element is the sum of the initial value and the difference stored in that element location.

```
SUBROUTINE UNPACK(CPACK,RVAR,NX,NY,NXY,NEXP,VAR1)

CHARACTER*1 CPACK(NXY)

REAL*4 RVAR(NX,NY)

SCALE=2.0**(7-NEXP)

VOLD=VAR1

INDX=0

DO J=1,NY

DO I=1,NX

INDX=INDX+1

RVAR(I,J)=(ICHAR(CPACK(INDX))-127.)/SCALE+VOLD

VOLD=RVAR(I,J)

END DO
```

```
VOLD=RVAR(1,J)

END DO

RETURN
```

METEOROLOGY - Help

Sample Meteorological Programs

The source code for many different meteorological data applications can be found in the */metdata* and /*data2arl* directories. A summary of these programs and some others where only the executable is provided are given below. Some of these are available for execution through the GUI, others must be run from the command line. For a few programs, only the PC executable is available for distribution.

GRIB records

The various programs and library routines that are required to convert GRIB formatted meteorological data files to ARL HYSPLIT compatible format can be found in the */data2ar*l directory. These are summarized below:

ncr2arl - decodes NCEP/NCAR re-analysis GRIB fields on pressure surfaces. A newer program, grib2arl, may be used instead.

rsm2arl and prsm2arl - decodes the Regional Spectral Model (RSM) GRIB output fields on sigma and pressure surfaces to ARL packed format.

ecm2arl - decodes ECMWF global model GRIB fields on a latitude-longitude grid. Requires the ECMWF supplied GRIBEX subroutines. A newer program, grib2arl, may be used instead.

content - decodes individual GRIB sections in a record for diagnostic purposes. This program does not unpack the data but only lists the contents of the GRIB file.

inventory - decodes all the records within a GRIB file (without unpacking) providing file content information.

unpacker - decodes GRIB records in a data file to a real data array.

sample - creates a packed meteorological file using dummy fields hardwired into the program. The input meteorological data subroutines should be replaced by routines reading user supplied meteorological data files.

grib2arl - a generic program to convert global data from ECMWF or NOAA on a global latitude longitude grid using the ARL provided GRIB decoder. ECMWF data may be on the native hybrid sigma or pressure surfaces. The program options are similar to avn2arl, but with some significant differences. The grib2arl version is capable of decoding both NOAA and ECMWF GRIB files. For computational purposes, HYSPLIT requires either surface pressure or terrain height as a surface field in the meteorological data file. The default is that surface pressure is assumed to be available in the input GRIB file, otherwise the terrain height is required. This option is set using the "-p" flag. ECMWF GRIB files may contain upper-level variables in one file, surface variables in another file, and invariant data in a third file. In this situation, the upper-level data are considered the primary file "-i" and the surface data are the supplemental file "-s", and the invariant data are the constant file "-c".

Usage: grib2arl [-options]

-i[primary grib data: file name {required}]

-s[supplemental grib data: file name {optional}]

-c[constant grib data: file name {optional}]

-x[subgrid extract center longitude {-80.0}]

-y[subgrid extract center latitude {60.0}]

-g[output projection 0 :conformal extract

1 :fixed northern hemisphere polar

2 :fixed southern hemisphere polar

{3}:lat-lon global grid (as input)

4 :lat-lon extract grid

-n[number of (x:y) extract grid points {100}]

-k[number of output levels including sfc {16}]

-p[surface defined by {1}:pressure or 0:terrain height]

-q[analyze grib file {0} or use saved configuration: 1]

-z[zero initialization of output file 0:no {1}:yes]

avn2arl - a more customized program to convert global data only from NOAA on a global latitude longitude grid. Selecting extract (-g0) interpolates the global latitude-longitude GRIB data to a conformal map projection of 100x100 grid points (-n100) at 100 km resolution at the center of the latitude (-y) and longitude (-x) selection of the slider bar. Checking the North (-g1) or South (-g2) hemisphere option, automatically creates the appropriate hemispheric polar stereo-graphic projection centered about the pole. The global check-button (-g3) does no conformal re-interpolation, but just converts the original one-degree latitude-longitude grid data to the ARL packed format. The GRIB converter can also be run from the command line with the following options:

Usage: avn2arl [-options]

-i[GRIB input file]

-x[extract center longitude]

-y[extract center latitude]

-g[0:conformal 1:north-hemis 2:south-hemis 3:global]

-n[number of x,y extract grid points]

afwa2arl - decodes ETA model fields on pressure surfaces and on the AWIPS-212 grid. The ETA GRIB converter can also be run from the command line with the following options:

Usage: afwa2arl [file_1] [file_2] [...],

where file_1, etc, are all the GRIB files that contain data for the same time period. The GRIB decoder only processes data for one time period per execution. Multiple input files may be required as some meteorological centers split surface, upper-air, and diagnostic variables, into different files. The decoder is identical in functionality to the *eta40arl* program, which also converts GRIB data files on the AWIPS-212 grid. However the afwa2arl program produces a more condensed file with only the variables required for HYSPLIT simulations.

mm5toarl - program to convert MM5 V3 format files.

ffsubs - library routines required for direct access to variable length records

ll2grid - library routines to convert from latitude-longitude to a conformal map projection

packing - library routines to read and write ARL packed data records.

w3arl - library routines of the ARL version of the NCEP w3lib GRIB record decoder.

ARL Packed Format

The various programs and library routines that are required to manipulate or display meteorological data already in the ARL packed format can be found in the */metdata* directory. Note that when these programs are run from the command line there will be a typical two line prompt, the first for the directory and second for the file name. A directory name should always have the proper terminator (/ or \). The following programs can be found:

chk_rec - program to dump the first 50 bytes of each meteorological data record. Those bytes contain ASCII data describing the packing.

chk_file - program to examine header and data records of an ARL packed meteorological data file. The program uses the same I/O subroutines common to HYSPLIT code. If this program does not work with a data file, neither will HYSPLIT.

chk_data - a simple program that shows how to read and unpack the ARL format packed meteorological data.

contour - creates postscript file of meteorological data contoured and color filled for a single variable at a specified time. The output written to *contour.ps*

display - creates postscript file of meteorological data contoured and color filled for a single variable at a specified time. The output written to *display.ps*. This program is prompted interactive version of the command line program *contour*.

edit_miss - a simple missing data interpolation program to fill in gaps.

profile - creates text file of the profile of meteorological variables at a specified location and time. The output is written to the screen and to the file *profile.txt*.

showgrid - shows the extent of the meteorological grid domain with a "+" symbol at the intersection of each node. The output is written to *showgrid.ps*

xtrct_grid - extracts a subgrid from the full grid meteorological data file. It permits selection by lat-lon corners and number of levels from the ground. Creates output file called "extract.bin"

*xtrct_tim*e - extracts data between two selected time periods from the designated meteorological file. Creates an output file called "*extract.bin*"

/source - contains remaining library routines not found in the */data2arl* directories.

TRAJECTORY - Help

The trajectory menu tab is composed of five main sections: setting up the simulation, executing the model, displaying the trajectory, converting the graphic to other formats, and configuring special simulations. The model is configured and executed through the menu and results displayed. However for experienced users, each component may be run independently from the command line and the CONTROL file could be created using any text editor.

In the Trajectory Setup menu the entire purpose of the GUI is to configure the model's input *CONTROL* file. This is a text file that configures the simulation parameters. Once the input parameters are set to their desired value, the model is executed from the Run Standard Model menu tab. When complete, the output window is closed and the Trajectory Display menu is used to draw and display the trajectory from the endpoints output file. The Special Simulations menu is used to configure several different customized simultaneous multiple trajectory simulations.

For inexperienced users, a review of the Quick Start Help is highly suggested, which goes through the trajectory computation step-by-step using the example meteorological data file.

TRAJECTORY - Quick Start Help

Upon installation the first time the model will be configured to run the example case discussed in more detail below. The "Quick Start" menu tab can be used to run any simulation in one-step. The last configuration will be used for the simulation. The "Quick Start" menu brings up a global map with the current source location shown as a blue dot. A "right-click" of the mouse button will write that source location to the control file, close the map, run the model simulation, create the output file, and display the graphics. A "left-click" of the mouse button will set a new source location. A "left-click-hold-drag-release" will define a new "zoomed" domain and the map will be redrawn. An "exit" from the window's menu bar will close the map without updating the control file or running the model. For more detailed simulation configurations, follow the steps below.

The easiest way to run the model is to use the GUI menu to create the model's input *Control* file. For the purposes of this demonstration appropriate meteorological files are provided. If for some reason the menu system is not available, perhaps because Tcl/Tk was not installed, the Control file can be created manually

Step 1 - start the GUI menu system using *\working\hysplit4.tcl* or the desktop shortcut to *Hysplit4*. A widget will appear with the *HYSPLIT* graphic and three button options: *Menu*, *Help* and *Exit*. On some systems the graphic may be scrambled or the colors may be flat. Switching the PC display to VGA and 256 colors usually solves these problems. However it can be left alone as the faulty graphic will not affect any of the other widgets or display graphics. **Click on *Menu* tab.**

Step 2 - The four main menus of the Hysplit4 GUI will appear: Meteorology , Trajectory , Concentration , and Advanced. An additional small widget underneath the main menu gives the current Hysplit4 version information. Do not delete this widget as it will terminate the GUI. It provides the reference frame for the model's standard output and messages. **Click on the *Trajectory* tab**.

Step 3 - Five options appear under this item: Trajectory Setup, Run Standard Model, Trajectory Display , Utility Programs , *and* Special Simulations. Normally these are run in sequence, however any item can be selected and run if the appropriate input files were created during a previous simulation. **Click on the *Trajectory Setup* tab**.

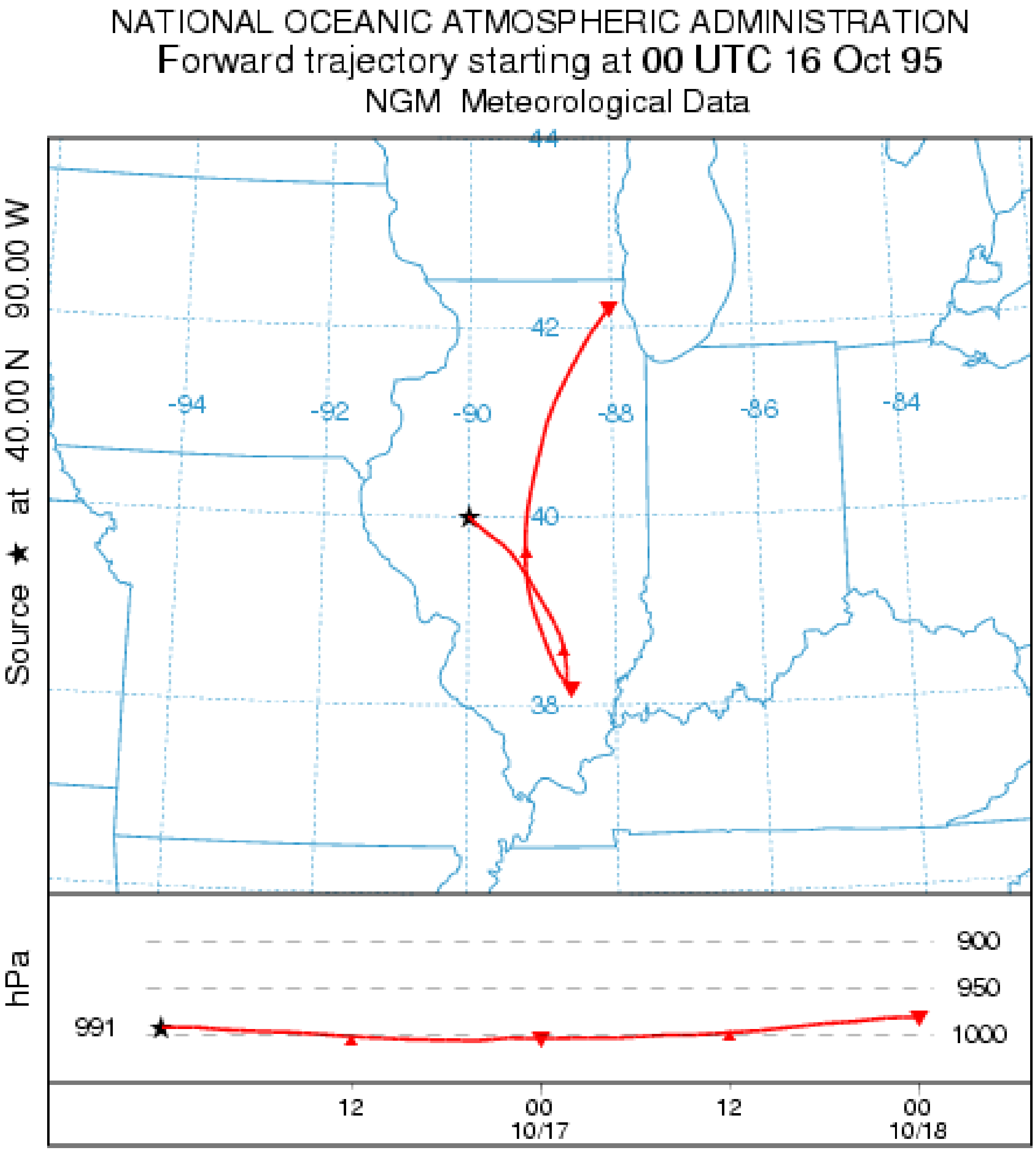
Step 4 - *Trajectory Setup* is used to enter the basic model simulation parameters: the starting time of the calculation; starting location in terms of latitude, longitude, and height; the run-time or duration of the trajectory calculation; and the names and locations of all required files. When modifications to this menu are complete, click on *Save*. However for this example, you will use the Retrieve option for predefined configurations, so do nothing here and **go on to Step 5**.

Step 5 - The example calculation is configured by **clicking on *Retrieve*** and then ***entering the text: sample_traj***, which is the name of the example simulation control file that was created for this demonstration. Then **click on *OK.* A**fter the data entry widget is closed, **click on *Save*** and the setup menu will close.

Step 6 - **Click on *Run Standard Model***, which first copies the setup configuration (default_traj) saved in the previous step, to the model's input *Control* file. The model calculation is then started. A series of messages will appear on standard output text window showing the progress of the calculation. When the simulation is completed, the trajectory end-points output file is ready to be converted to a graphical display. Under Windows 95/98 the standard output widget will not show any output until the end of the calculation and the *Trajectory* menu items will be locked until the calculation completes.

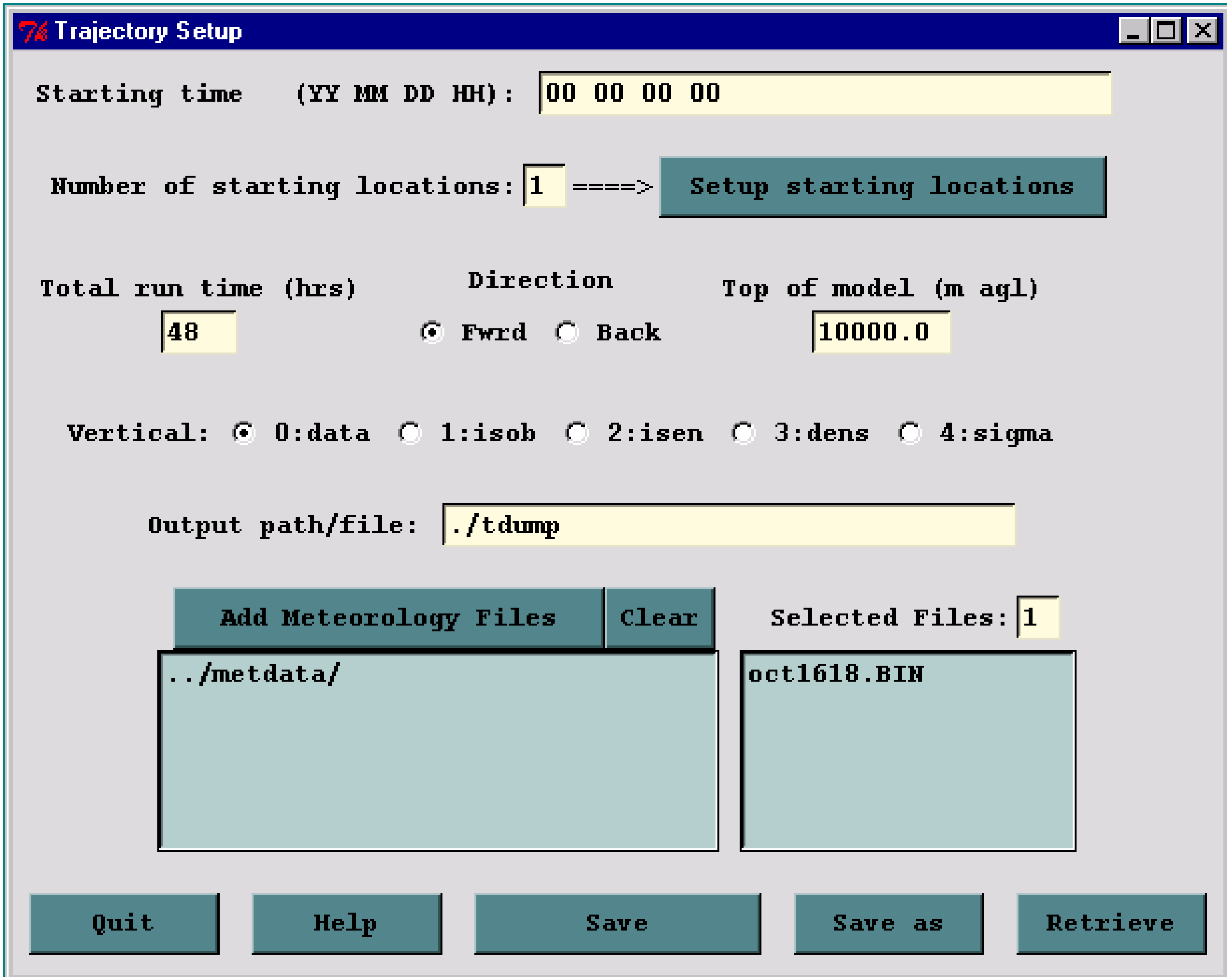
After completion **click on Exit** to close the window.

Step 7 - Selecting Trajectory Display will run a special program that converts the text file of trajectory end-point positions into a high quality Postscript file (*trajplot.ps*) suitable for printing. The conversion widget provides options for the frequency of the labels on the trajectory, a variable zoom factor, and color or black and white graphics options. If the Postscript viewer (Ghostscript / Ghostview) has been installed and associated with the .ps file suffix, then it will be invoked by the GUI. If the viewer does not automatically open, it may be necessary to manually edit *../guicode/hysplit4.tcl* to change the directory location associated with the program *gsview32.exe*. The Utility Programs tab contains only one option - converting the Postscript file to other graphical formats. The format is specified by the file name suffix: e.g. trajplot.ps to trajplot.gif. After **clicking on Execute Display**, the following graphic will appear in the *Ghostview* window:

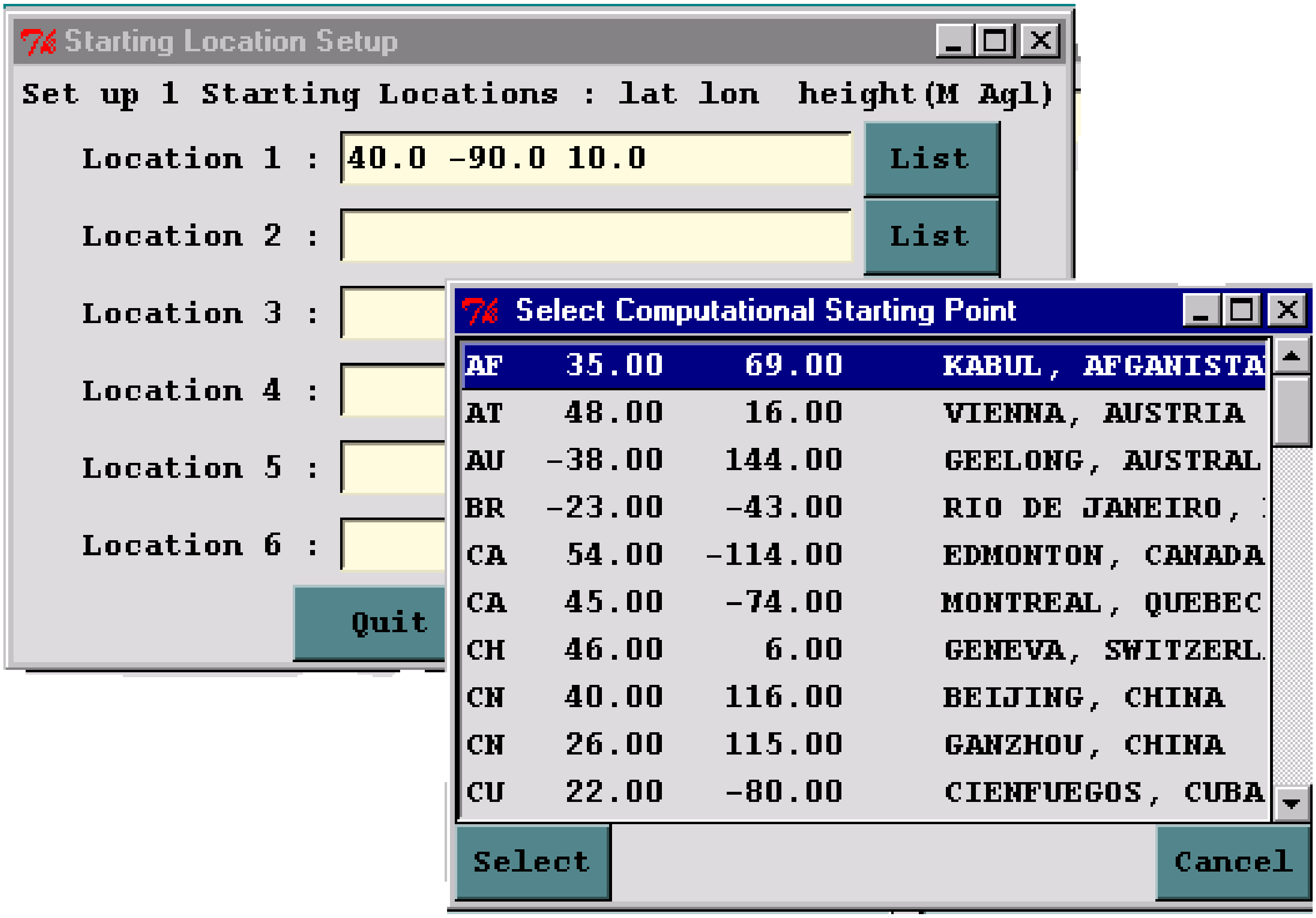


TRAJECTORY - Setup Menu

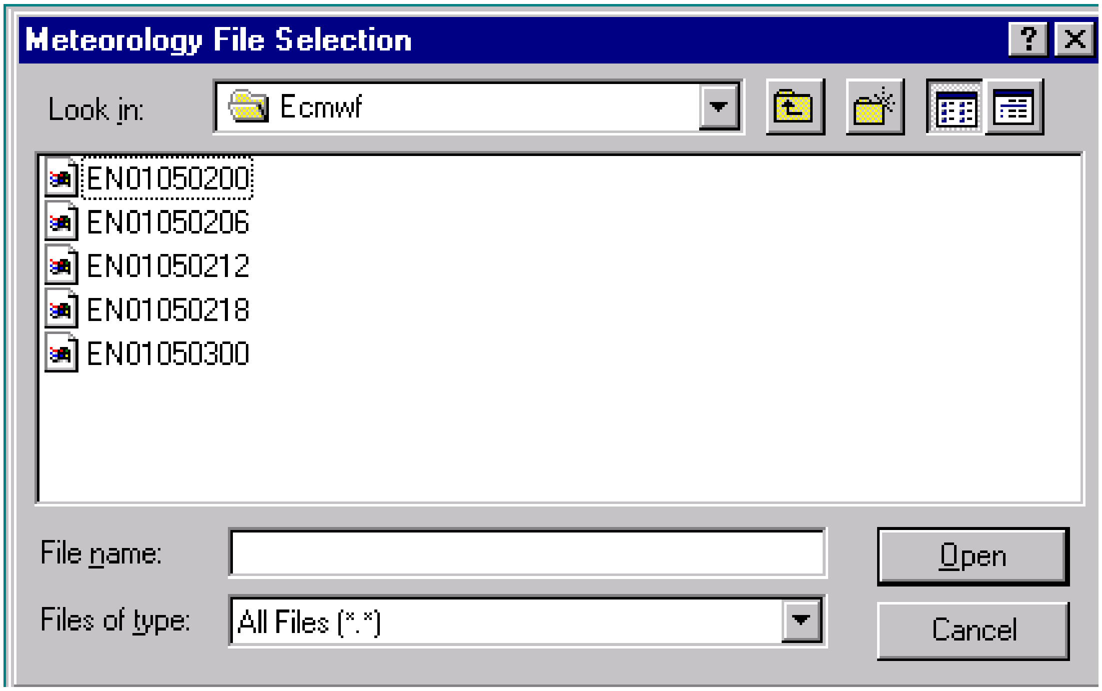
When the *Trajectory Setup* tab is executed, the *default_traj* file is read and the current parameter values are loaded. The menu for the example simulation (*sample_traj*) is shown below. The options shown on the menu correspond with the various lines in the CONTROL file. See the discussion of the control file format for a more detailed description for each of these parameters. However there are some features of the GUI that require additional explanation.



Clicking the *Setup Starting Locations* tab brings up the menu shown below. If the number of starting locations were to have been changed from the default value of one on the main menu, then there would be that number of starting location lines set on the menu, all with the same latitude and longitude. These could then be manually edited for different locations or starting heights. Another possibility would be to click on the *List* tab, which brings up a list of pre-selected starting locations from a file called *plants.txt*, which can be found in the *../hysplit4* directory. This file can be edited to reflect starting locations of interest to the user.



Another important feature of the main menu is how to select or add meteorological data files. The *Clear* button will erase all file selections, then pressing the *Add Meteorology Files* tab brings up the file selection dialog shown below. Select a file and click *Open* and the file will be added to the main menu. For each additional file, it is necessary to click again on the *Add Meteorology Files* tab. With each new file the selected files number is incremented by one.

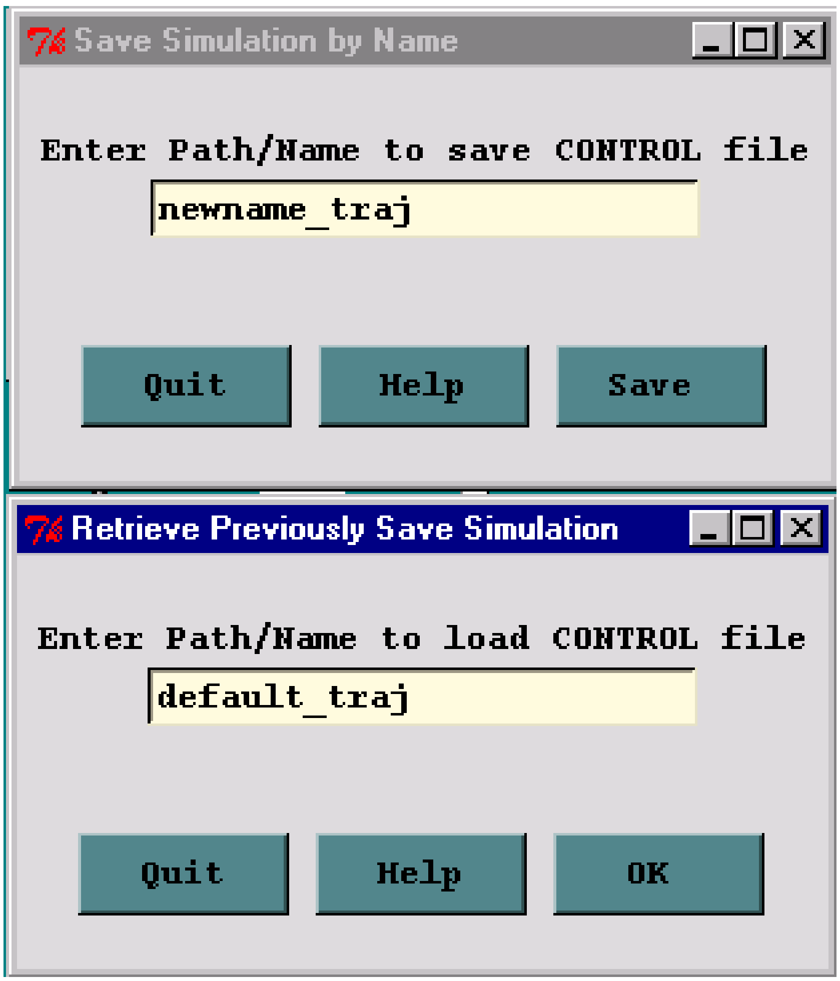


Once the simulation is configured as required, click the *Save* menu tab. This causes the GUI menu to over-write the values in *default_traj*. Note that the format of *default_traj* is identical to the *CONTROL* file. Clicking on *Save* closes the menu and then when the Run Standard Model tab is executed, *default_traj* is first copied to *CONTROL*, and then the trajectory model is executed.

When the GUI menu system is restarted, it loads the last values stored in *default_traj*. The *default_traj* file may also be saved to another name to permit future similar simulations to be set up more quickly. This option is available through the Save As and Retrieve menu tab of the setup menu.

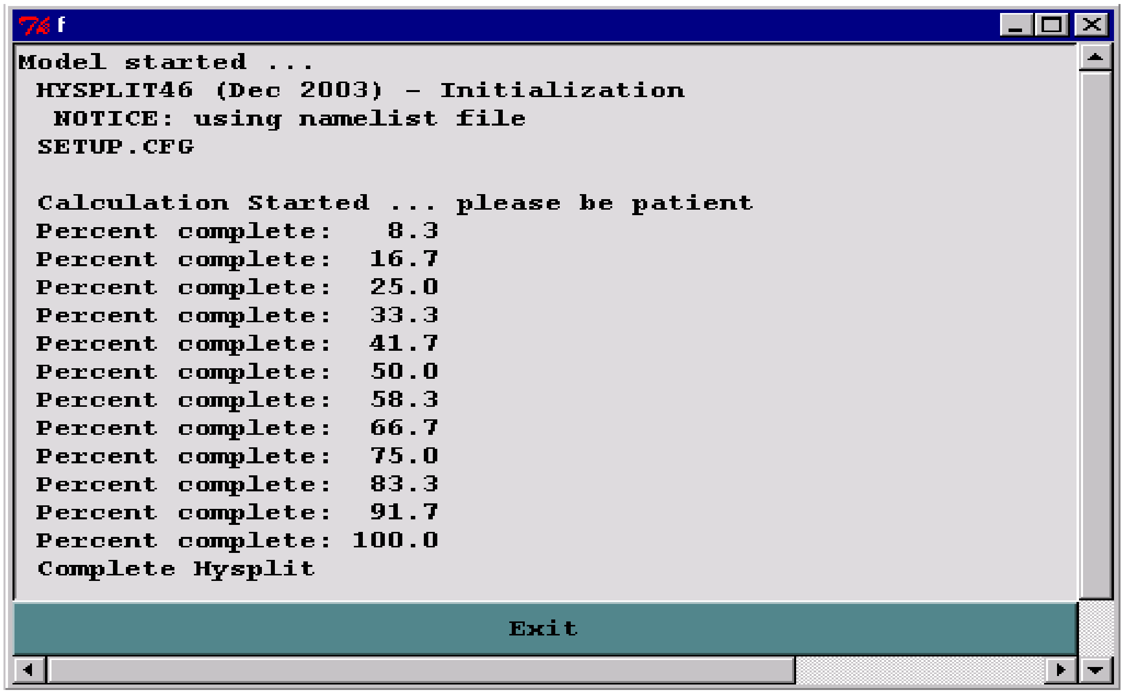
TRAJECTORY and CONCENTRATION - Saving and Retrieving

The trajectory and concentration setup menus for the *CONTROL* file permit a *Save As* or *Retrieve* option. Each of these menus is shown in the panel below. After a simulation has been configured, run, displayed, and assuming that the results are satisfactory and no more changes are required, then go back to the setup menu tab and *Save As* the control file by giving it a unique name, such as *newname_traj* in the example below. There are no restrictions on the naming convention. If the simulation is to be rerun, perhaps with some minor variation, the *Retrieve* option is used to load those values back into the setup menu.



TRAJECTORY - Run Standard Model

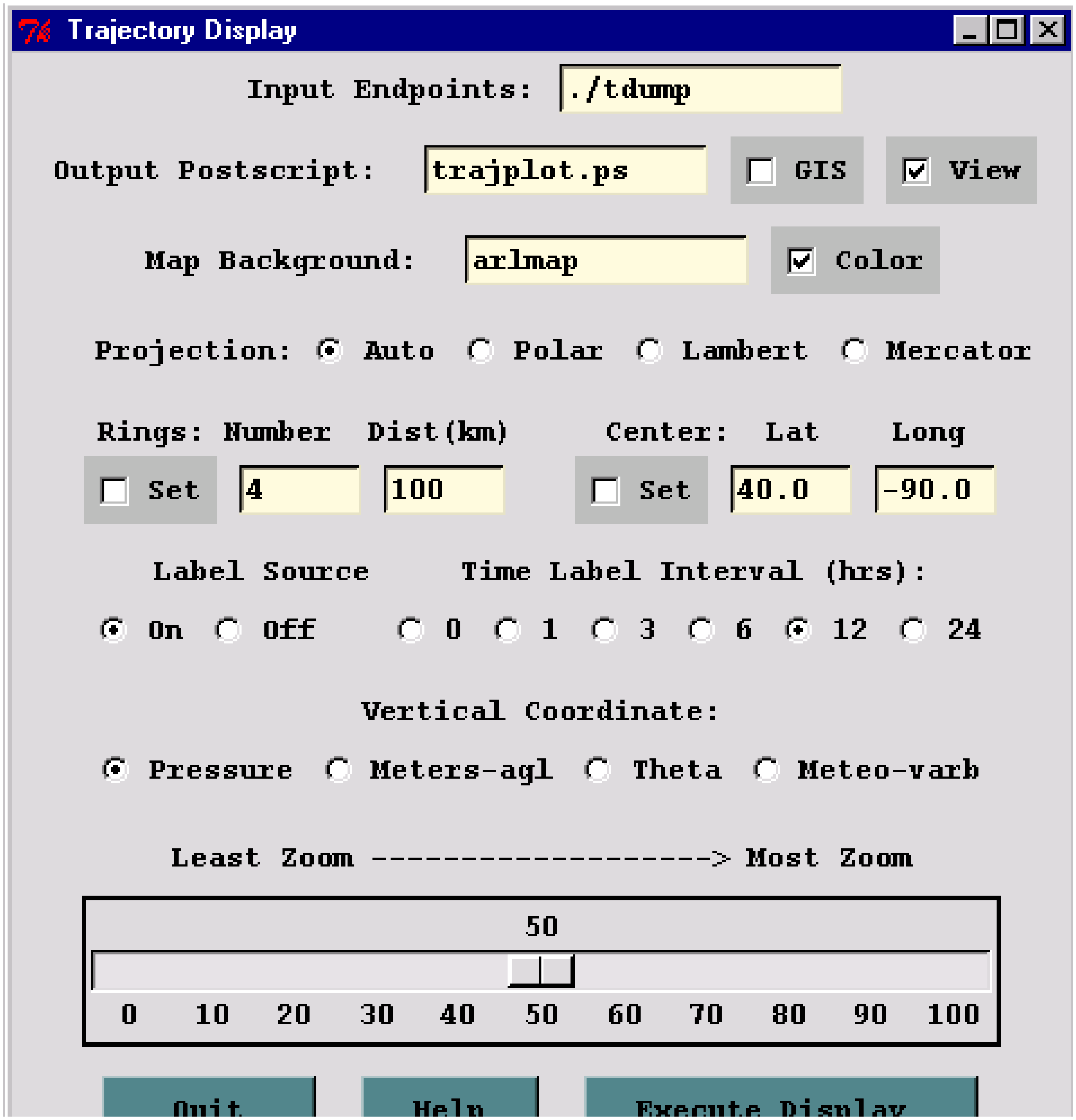
Once the Trajectory Setup menu has been closed with the *Save* button, the changes to the simulation parameters are copied to *default_traj*. Clicking on the *Run Standard Mode*l menu tab first copies *default_tra*j to *CONTROL* and then runs the trajectory model executable, *hymodelt*. The executable, by default, attempts to open a file named *CONTROL* to read all the required input parameters. If not found, the model will prompt to standard output for values from standard input. This condition should not occur running the model through the GUI. When the model execution starts, output messages are written to a special window, an example of which is shown in the illustration below:



Successful completion of a simulation will show a similar message. Additional run-time diagnostic messages and other error messages are always written to a file called *MESSAGE*. This file may be viewed through one of the Advanced Menu tabs. Depending upon the nature of the error message, perhaps a failure in the model initialization process, error messages may also appear in above window. Once the model has completed, **press Exit** to close the window.

TRAJECTORY - Display

The trajectory model generates a text output file of end-point positions. The end-point position file is processed by *trajplot* to produce the Postscript display file. *Trajplot* can be accessed through the GUI or run directly from the command line. The display program has a variety of command line options, most of which are available through the GUI. There is a one-to-one relationship between GUI options, an example of which is shown below, and the command line options. There are several features particular to the GUI. First the *Trajectory Display* menu will not open unless the *Trajectory Setup* menu was first opened and *Saved*. This procedure sets all the GUI parameters to their *default_traj* values. These are the values shown in the *Display* menu. Normally the map projection is computed automatically based upon the location and length of the trajectory. However some trajectory combinations may give no maps or improperly scaled maps. In these situations one would want to over-ride the default projection and try forcing a selection.



All mapping programs use the same text map background file, *arlmap*, which normally would be located in the ../*graphics* directory. However, all graphics programs first search the local start directory first (../*working* if running the GUI), then the ../*graphics* directory. Customized map background files can be read instead of the default file for specialized applications. Some higher resolution map background files are available from the HYSPLIT download web page. These different map files may be accessed implicitly by putting them in the "../hysplit4 (the startup directory), or explicitly through the GUI by entering the name of the customized map background file. The map background file format consists of three or more records per graphic line with the following format:

2I5 - line number, number of latitude-longitude points in the line

10F6.2 - latitude points in line

10F7.2 - longitude points in line

Trajplot Command Line Options

The Postscript conversion program *trajplot* can be found in the ../*trajmdl* directory. The program reads the trajectory endpoints output file, calculates the most optimum map for display, and creates the output file - *trajplot.ps*. When executed from the command line, there are several Unix style command line options. There should be no space between the option and any arguments: trajplot -[options (default)]

-a[Arcview GIS: 0-no dump 1-dump to files]

Selecting the GIS checkbox creates an ASCII text file for each output time period that consists of the value and latitude-longitude points along each contour. This file can be converted to an Arcview Shapefile or converted for other GIS applications through the utility menu "GIS to Shapefile". The view checkbox would be disabled to do just the GIS conversion without opening the the Postscript viewer.

-f[Frames: (0)-all frames in one file 1-one frame per file]

= 0 - all files plotted on one frame

= 1 - one input file per output frame

-g [Circle overlay: ( )-auto, #circ (4), #circ:dist(km)]

= {blank} - draws four distance rings about source

point, automatically scaled

= {#) - draws the indicated number of rings about

the source point

= {#1}:{#2} - draws the number {#1} of rings about the source point each {#2} kilometers apart. In the

special case where #1 is zero, the rings are not

drawn, but the size of the plot is scaled as if one

ring of radius {#2} is to be drawn.

-h [Hold map at center lat-lon: (source point), lat:lon]

= latitude:longitude - forces the map center to be at

the selected location instead of the source point. In

conjunction with the -g1:{km} option it is possible to

create a constant map projection for each execution.

-i[Input files: name1+name2+name3+... (tdump)]

= tdump - default file name.

= {user defined}

= {name1+name2+...} - to overlay multiple tdump files on the same plot based upon file name1 limits.

= +filename - filename is a file of filenames to be plotted following the same convention as the previous argument.

-j[Map background file: (arlmap)]

-k[Kolor: 0-B&W (1)-Color]

= 0 - for black and white output

= 1 - for color differentiation of multiple trajectories

-l[Label interval: 0, (6), 12, ... hrs]

= 0 - for no labels along the trajectory

= 6 - for labels every 6 hours

= 12 - for labels every 12 hours

= {X} - any hour selection permitted

= {-X} - negative value turns off source point labels

-m[Map projection: (0)-auto 1-polar 2-lambert 3-mercator]

-o[Output file name: (trajplot.ps)]

= trajplot.ps is the default Postscript output file name, otherwise it may be {user defined}.

-p[Process file name suffix: (ps) or process ID}

-v[Vertical: 0-pressure (1)-agl 2-isentropic 3-meteorology]

= 0 - The vertical trajectory plot coordinate is in hPa (absolute pressure units).

= 1 - The vertical trajectory plot coordinate is meters above ground level.

= 2 - An isentropic coordinate display requires the trajectory to have been run in those coordinates.

=3 - Forces the display of one of the selected meteorological variables along the trajectory rather than a trajectory vertical position coordinate. The meteorological variable is selected in the "Trajectory" tab of the "Advanced Configuration" menu. Only one meteorological variable can be selected for display. If multiple variables are selected, then only the last variable will be plotted.

-z[Zoom factor: 0-least zoom, (50), 100-most zoom]

= 50 - for standard resolution maps.

= 100 - for high resolution maps (maximum zoom)

Additional Map Label Customization

Many of the Postscript graphics programs have label information that can be customized to some extent. This is accomplished by placing a file called *Labels.cfg* in the startup directory which contains the following valid entries (all in single quotes terminated by &) replacing the new string with the desired text. A sample file called *Labels.bak* may be found in the relevant directory. Not all label strings are valid with every plotting program. For instance with *trajplot*, only the title entry would be used to replace the top label line of the plot.

'TITLE&','NATIONAL WEATHER SERVICE&'

'MAPID&','PROBABILITY&'

'LAYER&','BOUNDARY LAYER AVERAGE&'

'UNITS&','BQ&'

'VOLUM&','/M3&'

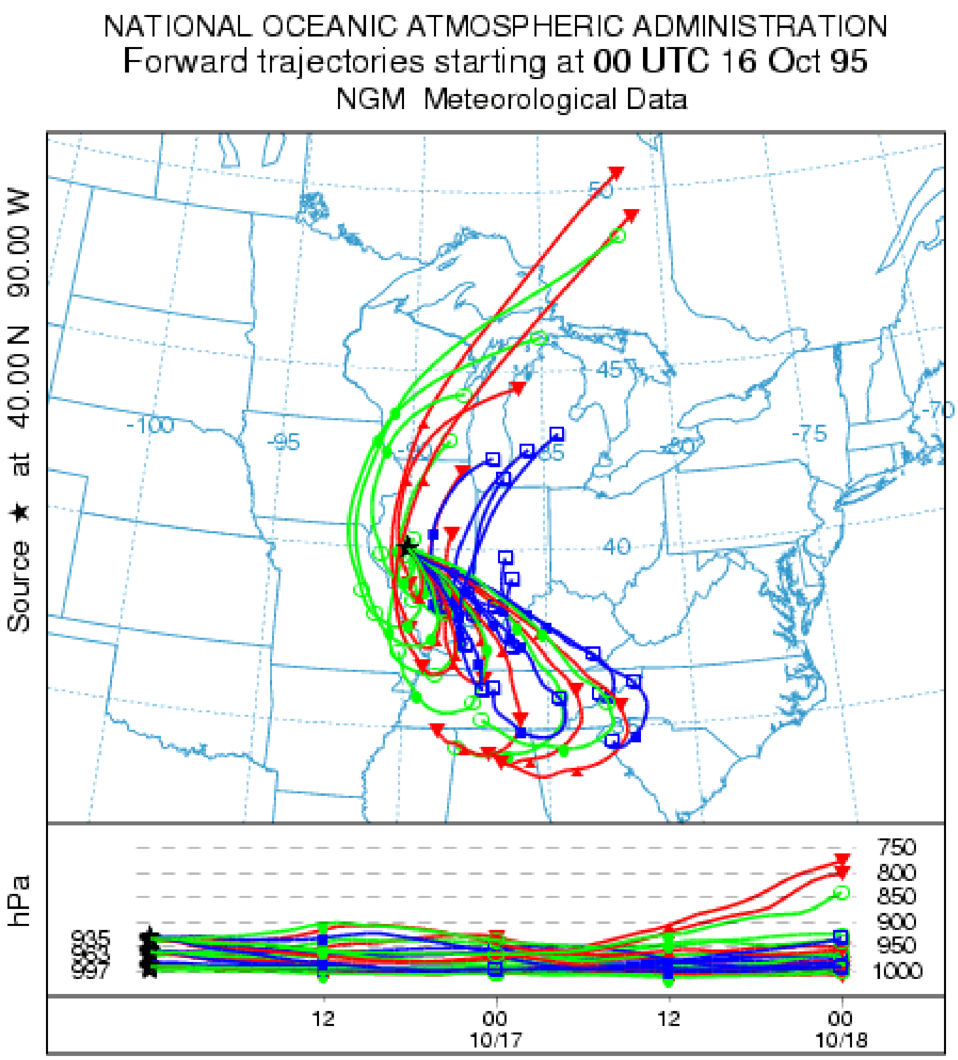
Additional supplemental text may be added at the bottom of the graphic by creating a file called *MAPTEXT.CFG*, also to be located in the startup directory. This is a generic file used by all plotting programs but each program will used different lines in its display. The file can be created and edited through the Advanced menu tab.

TRAJECTORY - Special Simulations

The *Special simulations* menu tab is required because certain options may require a different executable file, modifications to the Control file that are not supported by the GUI, or interactions with other items under the Advanced Menu tab. More information is provided below for each special simulation. Some *Special Simulations* may not be available for all operating systems.

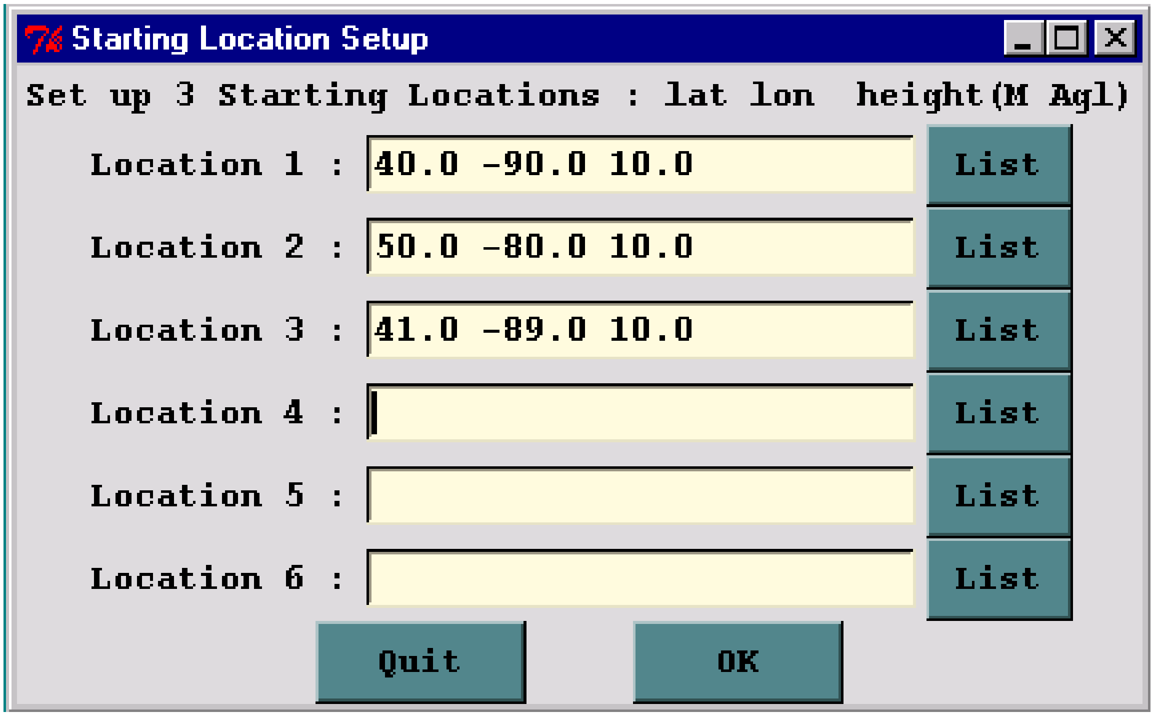
Run Ensemble

The ensemble form of the model automatically starts multiple trajectories from the selected starting point. Each member of the trajectory ensemble is calculated by offsetting the meteorological data by a fixed grid factor as defined in the Advanced Trajectory Configuration tab. The default offset is one meteorological grid point in the horizontal and 0.01 sigma units in the vertical. The results in 27 members for all-possible offsets in X, Y, and Z. After the model calculation has completed, use the normal Trajectory Display tab to view the results. Because the offset is computed in both directions in the vertical from the starting location, a starting location at the ground would not provide an optimal configuration for this type of simulation. The default vertical offset is about 250 m. Therefore this should be the minimum starting height for ensemble trajectories, unless the default offset is changed in the namelist file. An example of the ensemble trajectory using the example meteorological data is shown in the illustration below.

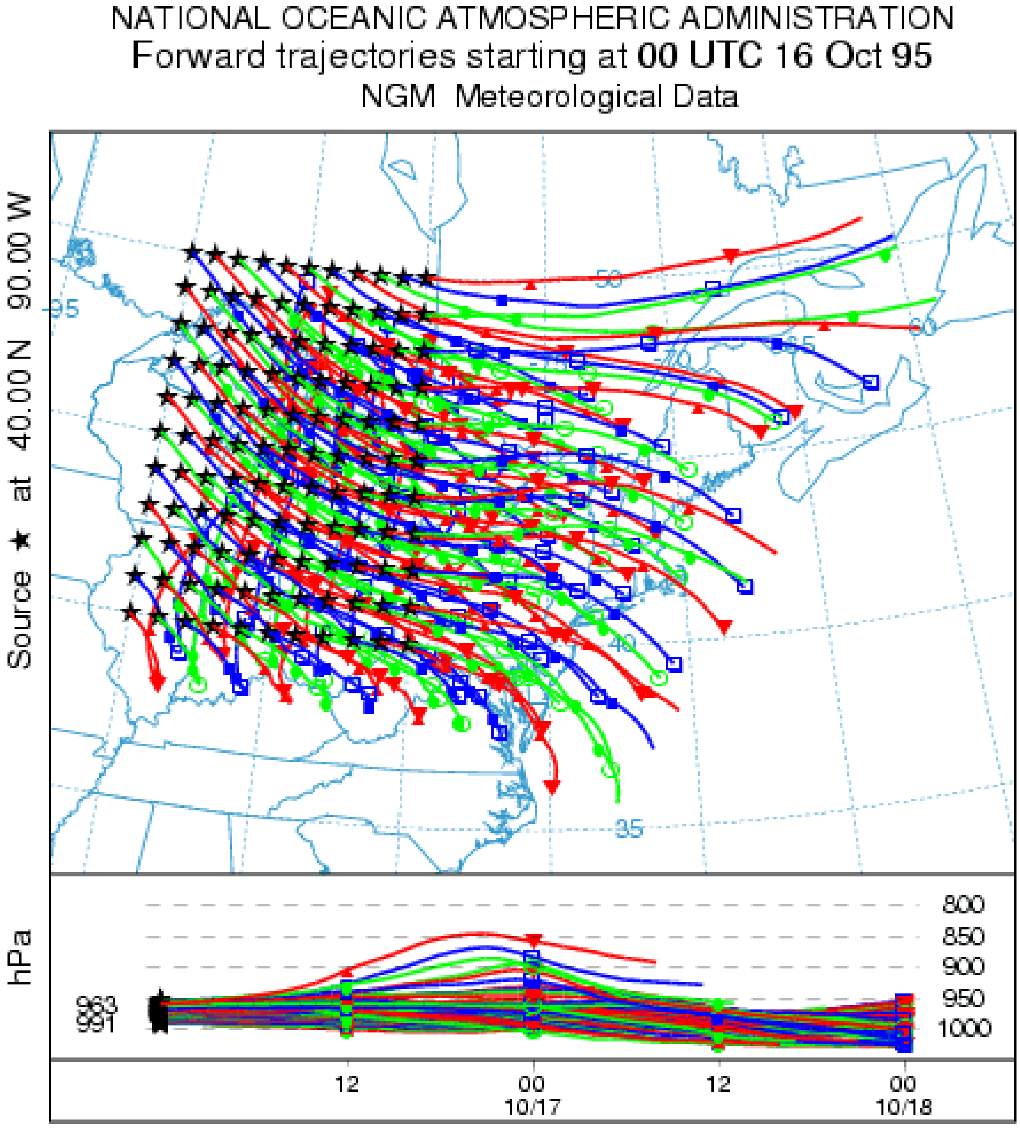


Run Matrix

The matrix calculation is a way to set up the *CONTROL* file for multiple starting locations that may exceed the GUI limit of 6 under the Trajectory Setup tab. Hundreds or thousands of starting points may be specified. The *Run Matrix* tab just executes the model using a *CONTROL* file that is dynamically created from the *default_traj* file using a special program called *latlon* that is called from within the GUI. The program reads a *CONTROL* file that is required to have three starting locations and then re-writes the same file with multiple locations. The multiple starting locations are computed by the *latlon* program based upon the number of starting points that fall in the domain between starting point 1 and starting point 2. Each new location is offset by an amount equal to the difference between starting locations 1 and 3. An example would be more illustrative. If the original control file has three starting locations: #1 at 40N, 90W, #2 at 50N, 80W, and #3 at 41N, 89W; then the matrix processing results in a CONTROL file with 121 starting locations, all one degree apart, all between 40N 90W and 50N 80W.

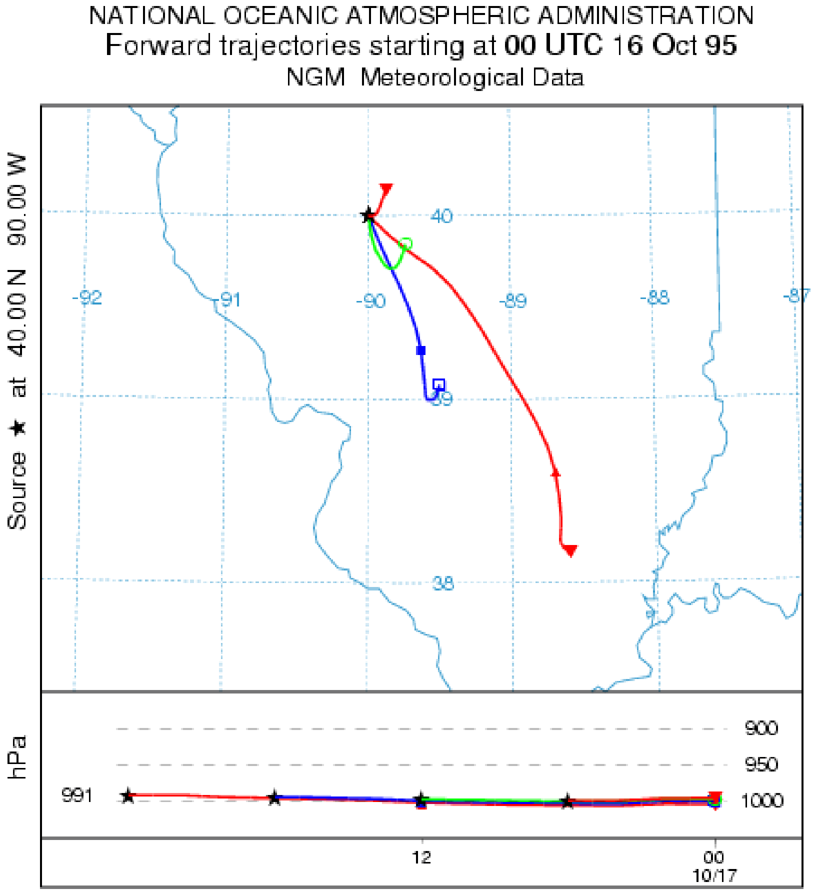


The reason for this approach is that only the *CONTROL* file is modified and not *default_traj*. The GUI never reads *CONTROL*, therefore the file does not need to conform to the GUI limits. The final matrix trajectory using the example meteorological data for this configuration is shown in the illustration below.



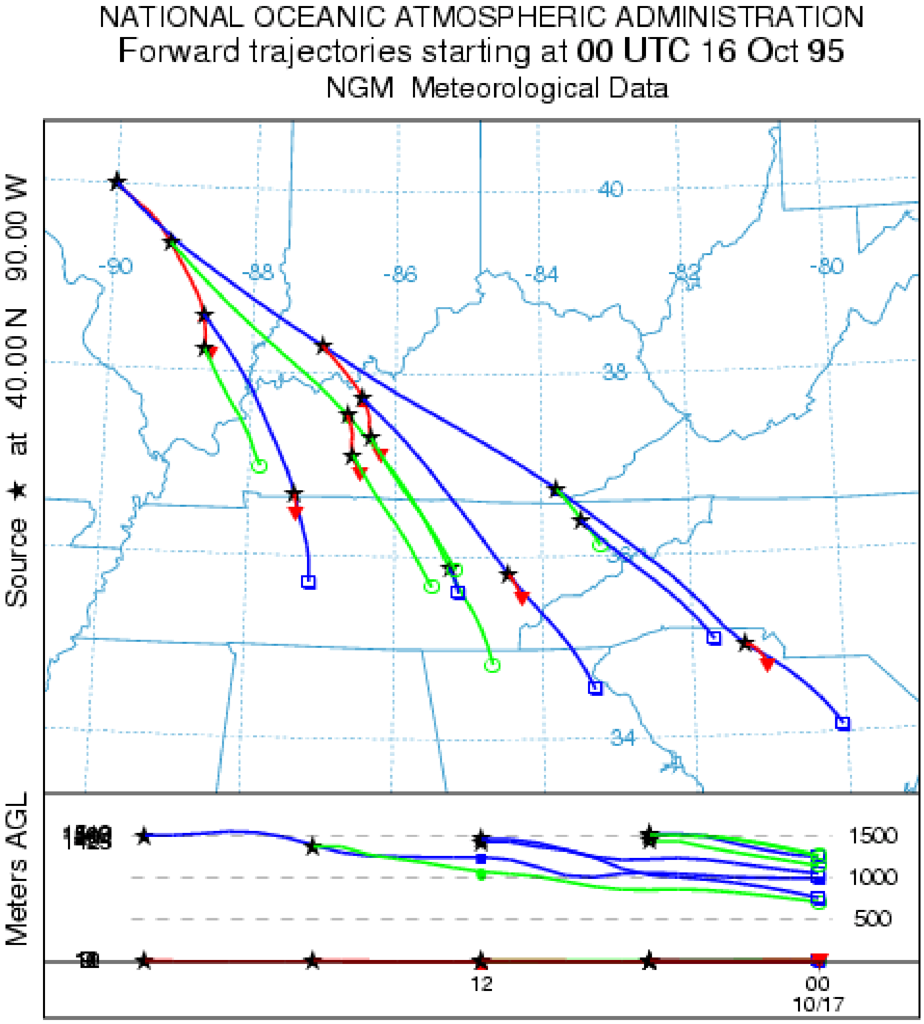
Multiple Trajectories in Time

In the normal simulation configuration, all trajectories start at time that was defined in the first line of the Trajectory Setup menu. Trajectories starting at a different time would require an independent simulation. However, there is shortcut to permit the calculation of multiple trajectories in time from the same starting location. Setup the simulation for a single trajectory then go to the Advanced Trajectory configuration menu. Under the *Temporal Trajectory Restart* line, edit the temporal interval in hours from the default of zero (no restarts) to the desired value. For example, if this value is set to 6 hours (corresponding to *NSTR=6* in the namelist file *SETUP.CFG*), then in addition to the normal trajectory that starts at the initial time, new trajectories would be started every six hours for the duration of the simulation. The result of this simulation, using the example meteorological data, is shown in the illustration below.



Multiple Trajectories in Space and Time

The standard setup options of the GUI permit the calculation of multiple trajectories in space or height by simply specifying new locations in the *CONTROL* file. A variation of that procedure was shown above with respect to setting up a matrix of starting locations. The model also permits the start of new trajectories at different locations along an existing trajectory. This can be considered a special case of multiple trajectories-in-time because when starting a new trajectory along an existing trajectory, the new starting location differs in time and space from the original trajectory. In this variation, new trajectories are started at multiple levels at the temporal restart interval. The multiple restart levels are assumed to be the same multiple starting heights specified in the original setup and therefore the number of restart heights must equal the number of starting locations. Go the Advanced Configuration menu and setup the example as in the previous case to restart trajectories every 6 hours. Then change the *Number of Levels* parameter from its default value of zero (no new trajectories) to the number of new trajectories will be started. For example, set the number of levels to 2 (sets the *NVER*=2 parameter in the *SETUP.CFG* namelist file). The number of starting locations should also be set to two, at the same location, but with release heights of 10 and 1500 m. Two trajectories, at 10 and 1500 m, will start every 6 hours. The result is shown in the illustration below.


NATIONAL OCEANIC ATMOSPHERIC ADMINISTRATION
Forward trajectories starting at 00 UTC 16 Oct 95
NGM Meteorological Data
Source ★ at 40.00 N 90.00 W
Meters AGL
40
-90
-88
-86
-84
-82
-80
38
36
34
1500
1000
500
12
00
10/17

TRAJECTORY - Setup Menu - Control File Format

The trajectory model input control file can be created using any text editor. However if the GUI is not being used, it would be easier to let the model create the initial file based upon standard output prompts. These are described in more detail below. When data entry is through the keyboard (a file named *CONTROL* is not found), a *Startup* file is created. This contains a copy of the input, and which later may be renamed to *Control* to permit direct editing and model execution without data entry. If you are unsure as to a value required in an input field, just enter the forward slash (/) character, and the indicated default value will be used. This default procedure is valid for all input fields except directory and file names. An automatic default selection procedure is also available for certain input fields of the *Control* file when they are set to zero. Those options are discussed in more detail below. Each input line is numbered (only in this text) according to the order it appears in the file. A number in parenthesis after the line number indicates that there is an input loop and multiple entry lines may be required depending upon the value of the previous entry.

1. Enter starting time (year, month, day, hour)

Default: 0 0 0 0

Enter the two digit values for the UTC time that the calculation is to start. Use 0's to start at the beginning (or end) of the file according to the direction of the calculation. All zero values in this field will force the calculation to use the time of the first (or last) record of the meteorological data file. In the special case where year and month are zero, day and hour are treated as relative to the start or end of the file. For example, the first record of the meteorological data file usually starts at 0000 UTC. An entry of "00 00 01 12" would start the calculation 36 hours from the start of the data file.

2- Enter number of starting locations

Default: 1

Simultaneous trajectories can be calculated at multiple levels or starting locations. The GUI menu can accommodate up to six simultaneous starting locations. Specification of additional locations can be accomplished through the Special Simulations menu tab, or through manual editing of the *Control* file and running the model outside of the GUI. When multiple starting locations are specified, all trajectories start at the same time. A multiple trajectory in time option is available through the Advanced menu through a namelist file parameter setting.

3(1)- Enter starting location (lat, lon, meters)

Default: 40.0 -90.0 50.0

Trajectory starting position in degrees and decimal (West and South are negative). Height is entered as meters above ground-level. An option to treat starting heights as relative to mean-sea-level is available through the Advanced menu through a namelist file parameter setting.

4- Enter total run time (hours)

Default: 48

Specifies the duration of the calculation in hours. Backward calculations are entered as negative hours. A backward trajectory starts from the trajectory termination point and proceeds upwind. Meteorological data are processed in reverse-time order.

5- Vertical motion option (0:data 1:isob 2:isen 3:dens 4:sigma 5:diverg 6:eta)

Default: 0

Indicates the vertical motion calculation method. The default "data" selection will use the meteorological model's vertical velocity fields; other options include isobaric, isentropic, constant density, constant internal sigma coordinate, computed from the velocity divergence, and a special transformation to correct the vertical velocities when mapped from quasi-horizontal ETA surfaces to HYSPLIT's internal terrain following sigma coordinate.

6- Top of model domain (internal coordinates m-agl)

Default: 10000.0

Sets the vertical limit of the internal meteorological grid. If calculations are not required above a certain level, fewer meteorological data are processed thus speeding up the computation. Trajectories will terminate when they reach this level. A secondary use of this parameter is to set the model's internal scaling height - the height at which the internal sigma surfaces go flat relative to terrain. The default internal scaling height is set to 25 km but it is set to the top of the model domain if the entry exceeds 25 km. Further, when meteorological data are provided on terrain sigma surfaces it is assumed that the input data were scaled to a height of 20 km (RAMS) or 34.8 km (COAMPS). If a different height is required to decode the input data, it should be entered on this line as the negative of the height. HYSPLIT's internal scaling height remains at 25 km unless the absolute value of the domain top exceeds 25 km.

7- Number of input data grids

Default: 1

Number of simultaneous input meteorological files. The following two entries (directory and name) will be repeated this number of times. A simulation will terminate when the computation is off all of the grids in either space or time. Trajectory calculations will check the grid each time step and use the finest resolution input data available at that location at that time. When multiple meteorological grids have different resolution, there is an additional restriction that there should be some overlap between the grids in time, otherwise it is not possible to transfer a trajectory position from one grid to another.

8(1)- Meteorological data grid # 1 directory

Default: (\main\sub\data\ )

Directory location of the meteorological file on the grid specified. Always terminate with the appropriate slash (\ or /).

9(2)- Meteorological data grid # 1 file name

Default: file_name

Name of the file containing meteorological data. Located in the previous directory.

10- Directory of trajectory output file

Default: (\main\trajectory\output\ )

Directory location to which the text trajectory end-points file will be written. Always terminate with the appropriate slash (\ or /).

11- Name of the trajectory endpoints file

Default: file_name

The trajectory end-points output file is named in this entry line.

TRAJECTORY - Endpoint File Format

The trajectory model generates its own text output file of ASCII end-point positions. The trajectory display program processes the end-point file. The format of the file is given below:

Record #1

I6 - Number of meteorological grids used in calculation

Records Loop #2 through the number of grids

A8 - Meteorological Model identification

5I6 - Data file starting Year, Month, Day, Hour, Forecast Hour

Record #3

I6 - number of different trajectories in file

A8 - direction of trajectory calculation (FORWARD, BACKWARD)

A8 - vertical motion calculation method (OMEGA, THETA, ...)

Record Loop #4 through the number of different trajectories in file

4I6 - starting year, month, day, hour

2F8.3 - starting latitude, longitude

F8.3 - starting level above ground (meters)

Record #5

I6 - number (n) of diagnostic output variables

nA8 - label identification of each variable (PRESSURE, THETA, ...)

Record Loop #6 through the number of hours in the simulation

I6 - trajectory number

I6 - meteorological grid number or antecedent trajectory number

5I6 - year month day hour minute of the point

I6 - forecast hour at point

F8.1 - age of the trajectory in hours

2F8.3 - position latitude and longitude

F8.1 - position height in meters above ground

nF8.1 - n diagnostic output variables (1st to be output is always pressure)

CONCENTRATION - Help

The concentration menu tab is composed of six main sections: setting up the simulation, executing the model, displaying the concentrations, various utility programs for converting the output to other formats, configuring special simulations, and simulations in a multi-processor environment. The model can be entirely configured and executed through the menu. However for experienced users, each component may be run independently from the command line.

In the Concentration Setup menu, the entire purpose of the GUI is to configure the model's input *CONTROL* file. This is a text file that configures the simulation parameters. Once the input parameters are set to their desired value, the model is executed from the Run Standard Model menu tab. When complete, the output window is closed and Display Options menu is used to draw and display the concentration contours from the model's binary concentration output file. The Special Simulations menu is used to configure several different customized simultaneous for ensemble applications, source-receptor matrices, and some simple chemistry simulations. The Multi-processor tab invokes some of the same special simulations but will only run under a multi-processor-computing environment. Normally this is not an option available under MS Windows.

For inexperienced users, a review of the Quick Start Help menu is highly suggested, which goes through a concentration computation step-by-step using the example meteorological data file.

CONCENTRATIONS - Quick Start Help

Upon installation the first time the model will be configured to run the example case discussed in more detail below. The "Quick Start" menu tab can be used to run any simulation in one-step. The last configuration will be used for the simulation. The "Quick Start" menu brings up a global map with the current source location shown as a blue dot. A "right-click" of the mouse button will write that source location to the control file, close the map, run the model simulation, create the output file, and display the graphics. A "left-click" of the mouse button will set a new source location. A "left-click-hold-drag-release" will define a new "zoomed" domain and the map will be redrawn. An "exit" from the window's menu bar will close the map without updating the control file or running the model. For more detailed simulation configurations, follow the steps below.

The easiest way to run the model is to use the GUI menu to edit the model's input *Control* file. For the purposes of this demonstration appropriate meteorological files are provided. If for some reason the menu system is not available, the *Control* file can be created manually.

Step 1 - start the GUI menu system using *\working\hysplit4.tcl* or the desktop shortcut to *Hysplit4*. A widget will appear with the *HYSPLIT* graphic and three button options: *Menu*, *Help* and *Exit*. On some systems the graphic may be scrambled or the colors may be flat. Switching the PC display to VGA and 256 colors usually solves these problems. However it can be left alone as the faulty graphic will not affect any of the other widgets or display graphics. **Click on *Menu*.**

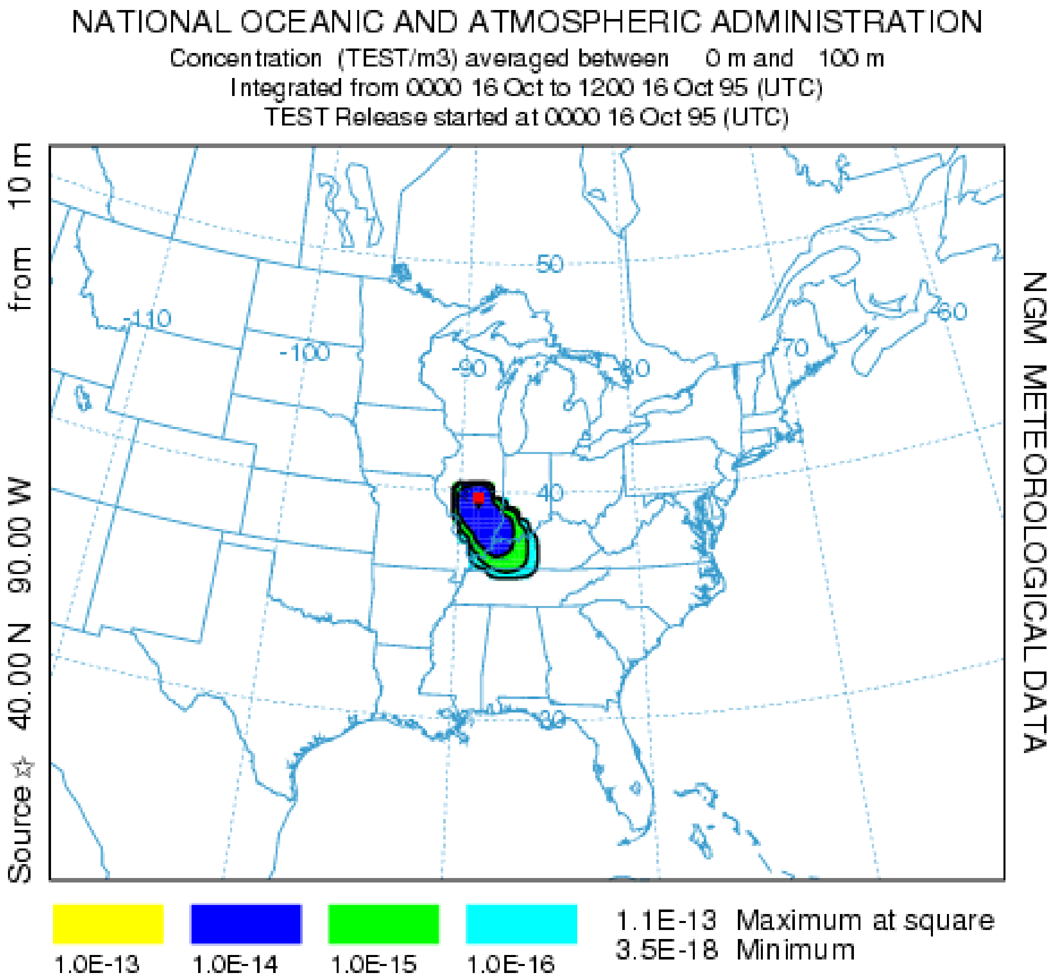
Step 2 - The four main menus of Hysplit4 will appear: Meteorology, Trajectory, Concentration, and the optional Advanced menu. An additional small widget underneath the main menu gives the current Hysplit4 version information. Do not delete this widget as it will terminate the GUI. It provides the reference frame for the model's standard output and messages. **Click on *Concentration*.**

Step 3 - Under the concentration menu there are also five options: concentration setup, run standard model, concentration display, utility programs, and special simulations. In general, they should be executed in sequential order. **Click on *Concentration Setup*.**

Step 4 - The *Concentration Setup* menu brings up similar starting information requirements as with the trajectory menu. There are three additional sub-menus: Pollutant - that can be used set the emission rate, duration, and start time of the emission; Grids - to set the location, resolution, levels, and averaging times of the concentration output grid; and Deposition - to set the characteristics of each pollutant. **Click on *Retrieve***, enter name of sample pre-configured control file: ***sample_conc***, then **click on *OK***. After the data entry widget is closed, **click on *Save*** and the setup menu will close.

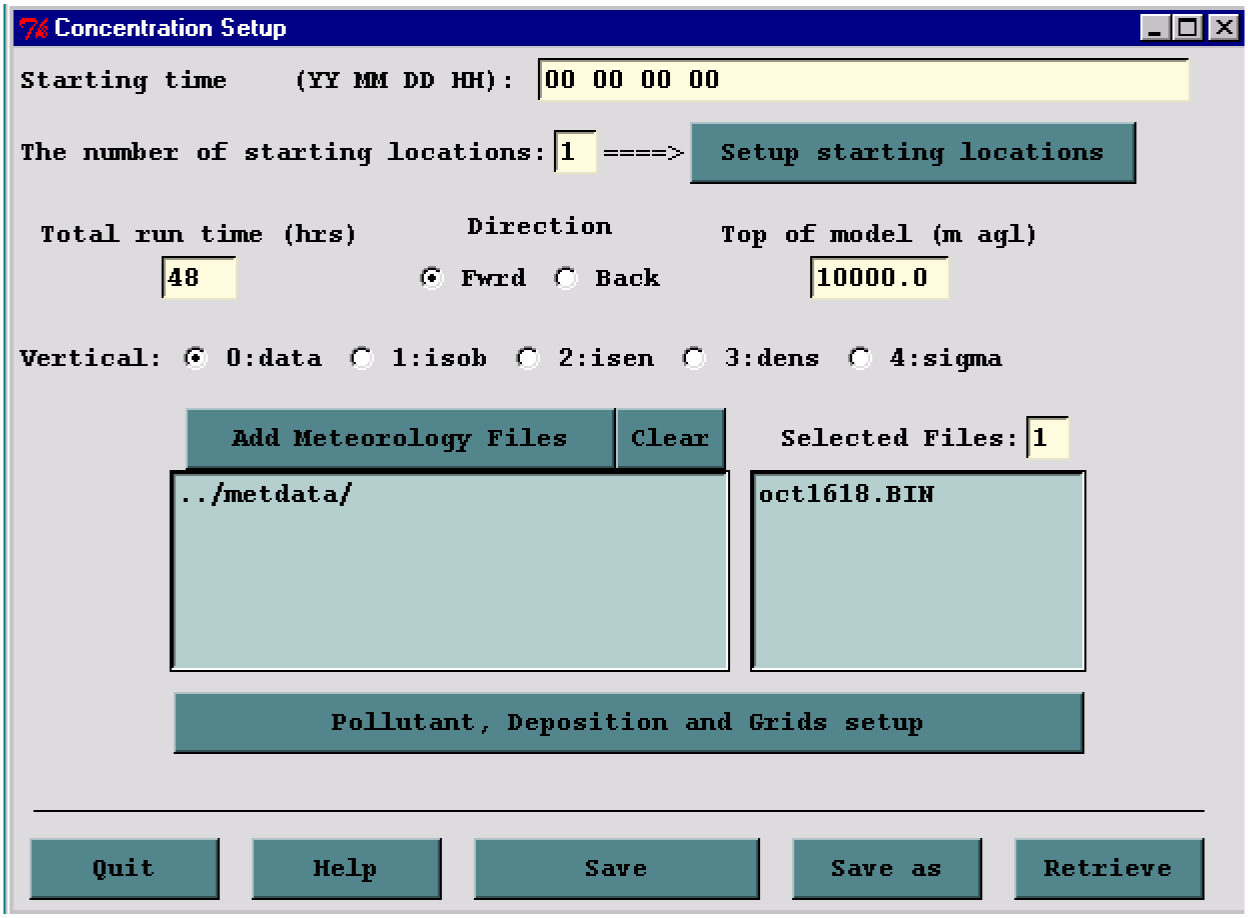
Step 5 - From the main concentration menu tab **select** Run Standard Model, which copies the setup configuration to the model's input *Control* file and starts the model simulation. Messages will appear on standard output showing the progress of the calculation after the calculation has completed. Be patient as concentration calculations may take considerably longer than trajectory calculations. **Click on Exit** to close the message window. At that point the binary concentration output file is ready to be converted to a graphical display.

Step 6 - **Click on *Display Options*** and then **select** Concentration to run a special program that converts the binary concentration file to the Postscript file *concplot.ps*, suitable for printing. The display widget contains multiple options for different pollutants (if defined), data grids, levels, and contour options. These are discussed in more detail in the graphics section. For this example, accept the defaults and just click **on *Execute Display***. If the Ghostview Postscript viewer has been installed and properly associated with .ps files, then it will be automatically invoked by the GUI. If the viewer does not open, it may be necessary to manually edit the file *hysplit4.tcl* for the directory entry associated with the program *gsview32.exe.* The output file can be printed directly on any Postscript device or printed through *Ghostscript/Ghostview*. The first frame (12 hour average) of the four frame simulation (48 hour duration) is shown in the illustration below.



CONCENTRATION - Setup Menu

The initial setup menu for the concentration model is identical to the trajectory setup menu in terms of starting time, location, and meteorology. These items will not be discussed again except to note the differences when applied to a dispersion simulation. The meaning of the entries in the CONTROL file that correspond to this setup menu are discussed in more detail below and should be reviewed to appreciate how the change in the simulation type (from trajectory to concentration) changes the meaning and context of the same input parameters. These parameters consist of the initial entries of the CONTROL file. The initial concentration setup menu is shown in the illustration below.



The entries in the *Control* file for air concentration simulations consist of four groups of input data. The first data group is almost identical to the trajectory simulation and is described in the next section. The other three groups define the pollutant emission characteristics, the concentration grid in terms of spacing and integration interval, and the pollutant characteristics relevant to computing deposition and removal processes. These latter three entries are accessed through the "Pollutant, Deposition, and Grids Setup" tab. Each of the these sections contains a more detailed description of the input parameters as well as the corresponding CONTROL file values that need to be set for command line simulations.

Initial CONTROL File Section

The concentration model input control file can be created using any text editor. However if the GUI is not being used, it would be easier to let the model create the initial file based upon standard output prompts. These are described in more detail below. When data entry is through the keyboard (a file named *CONTROL* is not found), a *Startup* file is created. This contains a copy of the input, and which later may be renamed to *Control* to permit direct editing and model execution without data entry. If you are unsure as to a value required in an input field, just enter the forward slash (/) character, and the indicated default value will be used. This default procedure is valid for all input fields except directory and file names. An automatic default selection procedure is also available for certain input fields of the *Control* file when they are set to zero. Those options are discussed in more detail below. Each input line is numbered (only in this text) according to the order it appears in the file. A number in parenthesis after the line number indicates that there is an input loop and multiple entry lines may be required depending upon the value of the previous entry.

1- Enter starting time (year, month, day, hour)

Default: 0 0 0 0

Enter the two digit values for the UTC time that the calculation is to start. Use 0's to start at the beginning (or end) of the file according to the direction of the calculation. All zero values in this field will force the calculation to use the time of the first (or last) record of the meteorological data file. In the special case where year and month are zero, day and hour are treated as relative to the start or end of the file. For example, the first record of the meteorological data file usually starts at 0000 UTC. An entry of "00 00 01 12" would start the calculation 36 hours from the start of the data file.

2- Number of starting locations

Default: 1

Single or multiple pollutant sources may be simultaneously tracked. The emission rate that is specified in the pollutant menu is assigned to each source. If multiple sources are defined at the same location, the emissions are distributed vertically in a layer between the current emission height and the previous source emission height. The effective source will be a vertical line source between the two heights. When multiple sources are in different locations, the pollutant is emitted as a point source from each location at the height specified. Point and vertical line sources can be mixed in the same simulation. The GUI menu can accommodate up to 6 simultaneous starting locations. Specification of additional locations requires manual editing of the *Control* file. Area source emissions can be specified from an input file: *emission.txt*. When this file is present in the root directory, the emission parameters in the *Control* file are superceded by the emission rates specified in the file. More information on this file structure can be found in the advanced help section.

3(1)- Enter starting location (lat, lon, meters, Opt-4, Opt-5)

Default: 40.0 -90.0 50.0

Position in degrees and decimal (West and South are negative). Height is entered as meters above ground level unless the mean-sea-level flag has been set.

The optional 4th (emission rate - units per hour) and 5th (emission area - square meters) columns on this input line can be used to supercede the value of the emission rate (line 12-2) when multiple sources are defined, otherwise all sources have the same rate as specified on line 12-2. The 5th column defines the virtual size of the source: point sources default to "0".

4- Enter total run time (hours)

Default: 48

Sets the duration of the calculation in hours. Backward calculations are configured by setting the run time to a negative value. See the discussion in the advanced help section on backward "dispersion" calculations.

5- Vertical motion option (0:data 1:isob 2:isen 3:dens 4:sigma 5:diverg 6:eta)

Default: 0

Indicates the vertical motion calculation method. The default "data" selection will use the meteorological model's vertical velocity fields; other options include isobaric, isentropic, constant density, constant internal sigma coordinate, computed from the velocity divergence, and a special transformation to correct the vertical velocities when mapped from quasi-horizontal ETA surfaces to HYSPLIT's internal terrain following sigma coordinate.

6- Top of model domain (internal coordinates m-agl)

Default: 10000.0

Sets the vertical limit of the internal meteorological grid. If calculations are not required above a certain level, fewer meteorological data are processed thus speeding up the computation. Trajectories will terminate when they reach this level. A secondary use of this parameter is to set the model's internal scaling height - the height at which the internal sigma surfaces go flat relative to terrain. The default internal scaling height is set to 25 km but it is set to the top of the model domain if the entry exceeds 25 km. Further, when meteorological data are provided on terrain sigma surfaces it is assumed that the input data were scaled to a height of 20 km (RAMS) or 34.8 km (COAMPS). If a different height is required to decode the input data, it should be entered on this line as the negative of the height. HYSPLIT's internal scaling height remains at 25 km unless the absolute value of the domain top exceeds 25 km.

7- Number of input data grids

Default: 1

Number of simultaneous input meteorological files. The following two entries (directory and name) will be repeated this number of times. A simulation will terminate when the computation is off all of the grids in either space or time. Calculations will check the grid each time step and use the finest resolution input data available at that location at that time. When multiple meteorological grids have different resolution, there is an additional restriction that there should be some overlap between the grids in time, otherwise it is not possible to transfer a particle position from one grid to another.

8(1)- Meteorological data grid # 1 directory

Default: (\main\sub\data\ )

Directory location of the meteorological file on the grid specified. Always terminate with the appropriate slash (\ or /).

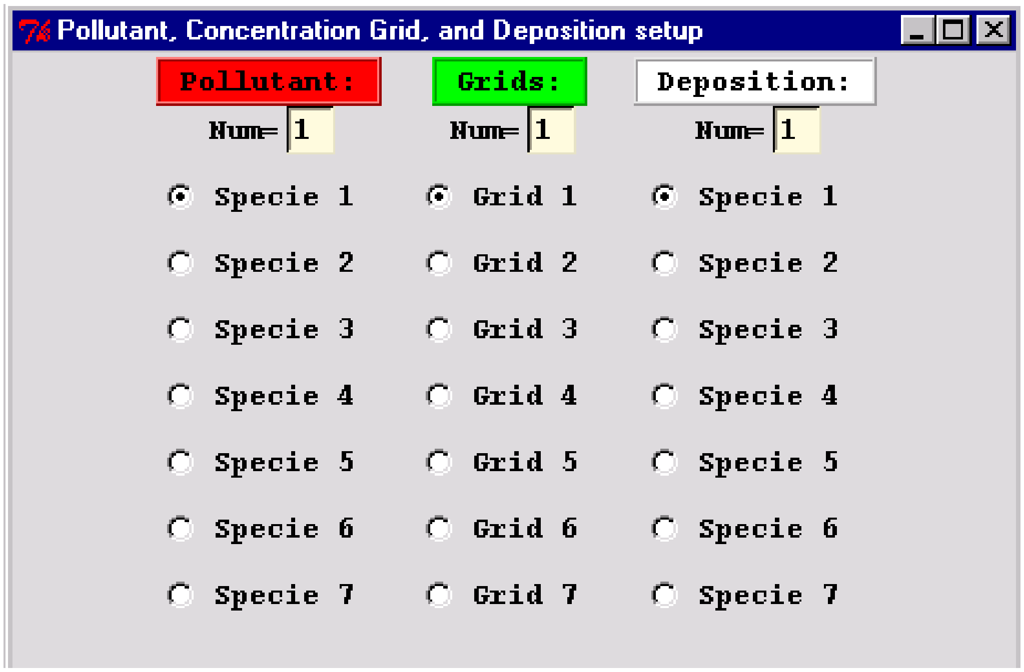
9(2)- Meteorological data grid # 1 file name

Default: file_name

Name of the file containing meteorological data. Located in the previous directory.

CONCENTRATION - Setup: Pollutant, Deposition, Grids

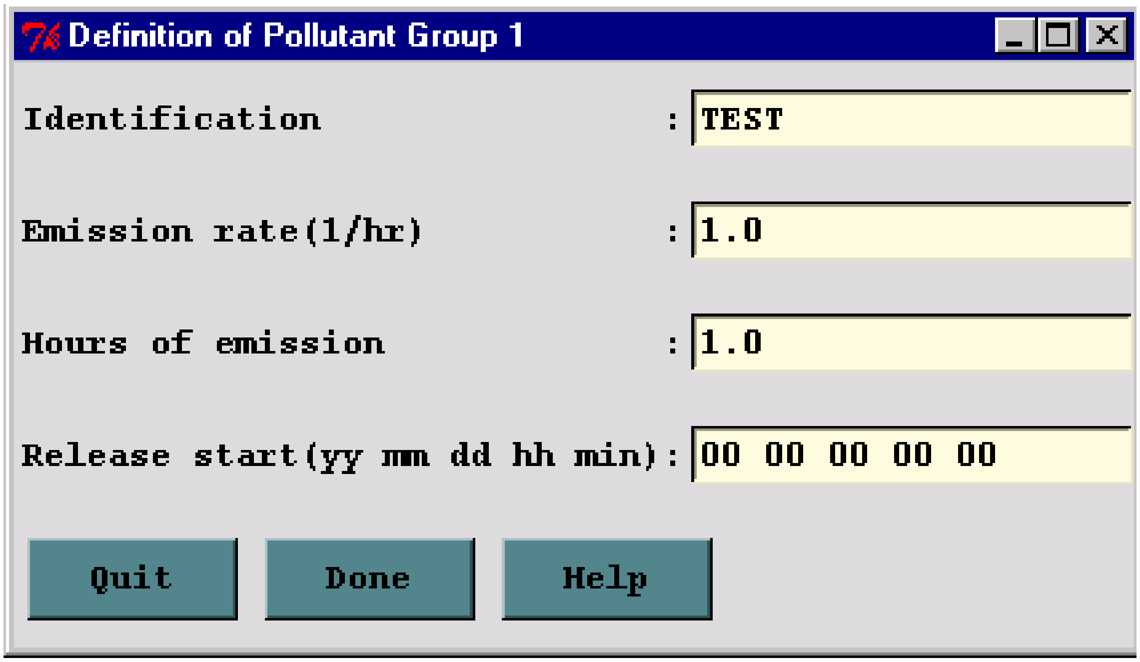
There are three menu choices under this tab: Pollutant, Grid, and Deposition. The first permits editing of the emission rate parameters, the second defines the concentration output grid, and the third the deposition characteristics of the pollutant, if that feature is enabled. This menu is illustrated below. To edit the parameters for a specific species entry just select the appropriate checkbox.



Note that up to 7 pollutant species may be defined by changing the *Num* parameter. However, in the current version, the pollutant and deposition menus must both reference the same number of species. Multiple species simulations are calculated independently, hence there is no computational benefit by doing two different simulations or combining both species in one simulation. The multiple species option is primarily used for chemical transformation simulations. A simple example is available from the configuration menu checkbox of "10% per hour", which transforms species #1 to species #2 at a rate of 10% per hour. The transformation occurs on the same particle and is discussed in more detail in the Advanced Applications section. More complex transformations require a linkage with compatible external modules. None are available at this time for public distribution.

CONCENTRATION - Pollutant Definition Setup

There are four *CONTROL* file entries, lines 11(1) through 14(4), that correspond with each line of the pollutant menu shown in the illustration below.



-

10- Number of different pollutants

Default: 1

Multiple pollutant species may be defined for emissions. Each pollutant is assigned to its own particle or puff and therefore may behave differently due to deposition or other pollutant specific characteristics. Each will be tracked on its own concentration grid. The following four entries are repeated for each pollutant defined.

11(1)- Pollutant four Character Identification

Default: TEST

Provides a four-character label that can be used to identify the pollutant. The label is written with the concentration output grid to identify output records associated with that pollutant and will appear in display labels. Additional user supplied deposition and chemistry calculations may be keyed to this identification string.

12(2)- Emission rate (per hour)

Default: 1.0

Mass units released each hour. Units are arbitrary except when specific chemical transformation subroutines are associated with the calculation. Output air concentration units will be in the same units as specified on this line. For instance an input of kg/hr results in an output of kg/m^3. When multiple sources are defined this rate is assigned to all sources unless the optional parameters are present on line 3(1).

13(3)- Hours of emission

Default: 1.0

The duration of emission may be defined in fractional hours. An emission duration of less than one time-step will be emitted over one time-step with a total emission that would yield the requested rate over the emission duration.

14(4)- Release start time: year month day hour minute

Default: [simulation start]

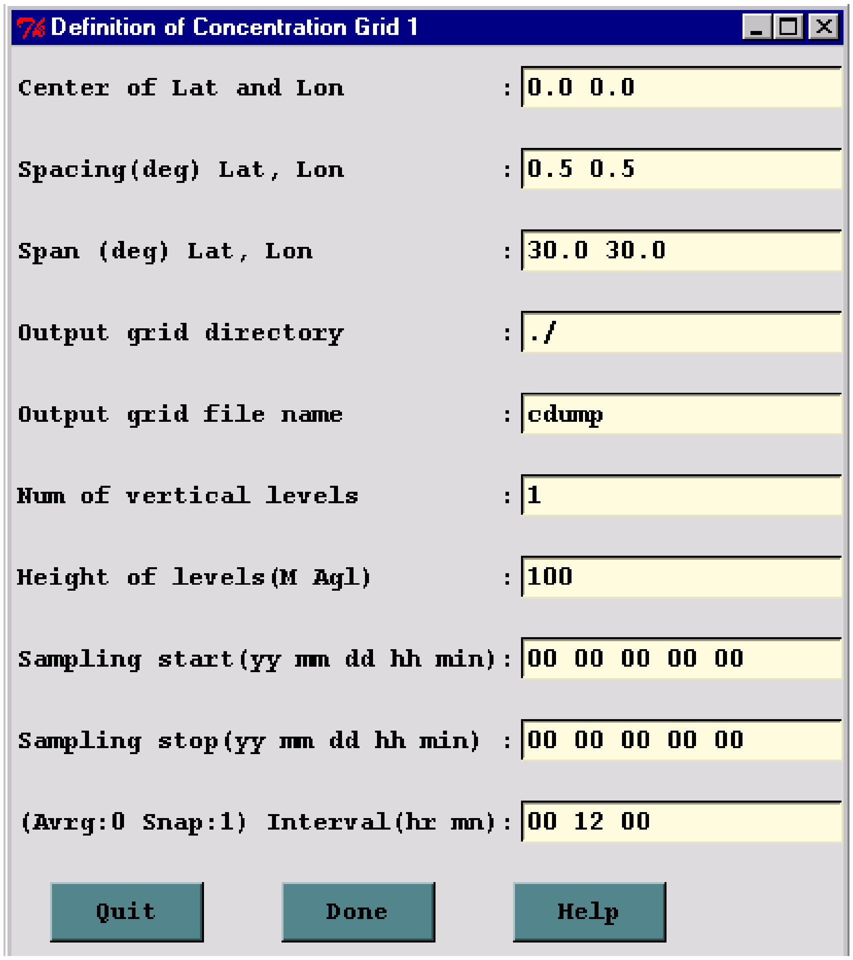
The previously specified hours of emission start at this time. An entry of zero's in the field, when input is read from a file, will also result in the selection of the default values that will correspond with the starting time of the meteorological data file. Day and hour are treated as relative to the file start when month is set to zero.

Temporal or Area Emission Variations

This menu is only designed to input point source emission rates. Unless additional input values are provided in the control file after each emission location, the same rate will apply to all defined point sources for the duration of the emission. When more complex emission scenarios are required, emission data can be read in from a file that defines a diurnal emission cycle for any number of pollutants at any number of locations. If the emission rate is to vary in time beyond one diurnal cycle, there is another input file that can be defined to set a new rate with each emission cycle. In this latter scenario the rate as well as the location may be changed.

CONCENTRATION - Setup Menu - Grid Definition

This section is used to define the grid system to which the concentrations are summed during the integration and subsequently for post-processing and display of the model's output. There are 10 entries in the CONTROL file for each concentration grid that has been defined. The lines 16(1) through 25(10) correspond with each of the menu items shown in the illustration below.



Dispersion calculations are performed on the computational (meteorological) grid without regard to the definition or location of any concentration grid. Therefore it is possible to complete a simulation and have no results to view if the concentration grid was in the wrong location. In addition, very small concentration grid spacing will reduce the model's integration time step and may result is substantially longer simulation clock times.

15- Number of simultaneous concentration grids

Default: 1

Multiple or nested grids may be defined. The concentration output grids are treated independently. The following 10 entries will be repeated for each grid defined.

16(1)- Center Latitude, Longitude (degrees)

Default: [source location]

Sets the center position of the concentration sampling grid in degrees and decimal. Input of zero's will result in selection of the default value, the location of the emission source. Sometimes it may be desirable to move the grid center location downwind near the center of the projected plume position.

17(2)- Grid spacing (degrees) Latitude, Longitude

Default: 1.0 1.0

Sets the interval in degrees between nodes of the sampling grid. Puffs must pass over a node to contribute concentration to that point and therefore if the spacing is too wide, they may pass between intersection points. Particle model calculations represent grid-cell averages, where each cell is centered on a node position, with its dimensions equal to the grid spacing. Finer resolution concentration grids require correspondingly finer integration time-steps. This may be mitigated to some extent by limiting fine resolution grids to only the first few hours of the simulation.

18(3)- Grid span (deg) Latitude, Longitude

Default: [180.0] [360.0]

Sets the total span of the grid in each direction. For instance, a span of 10 degrees would cover 5 degrees on each side of the center grid location. A plume that goes off the grid would have cutoff appearance, which can sometimes be mitigated by moving the grid center further downwind.

19(4)- Enter grid # 1 directory

Default: (\main\sub\output\ )

Directory to which the binary concentration output file for this grid is written. As in other directory entries a terminating (\) slash is required.

20(5)- Enter grid # 1 file name

Default: file_name

Name of the concentration output file for each grid. See Section 6 for a description of the format of the concentration output file.

21(6)- Number of vertical concentration levels

Default: 1

The number of vertical levels in the concentration grid including the ground surface level if deposition output is required.

22(7)- Height of each level (m)

Default: 50

Output grid levels may be defined in any order for the puff model as long as the deposition level (0) comes first (a height of zero indicates deposition output). Air concentrations must have a non-zero height defined. A height for the puff model indicates the concentration at that level. A height for the particle model indicates the average concentration between that level and the previous level (or the ground for the first level). Therefore heights for the particle model need to be defined in ascending order. Note that the default is to treat the levels as above-ground-level (AGL) unless the MSL (above Mean-Sea-Level) flag has been set (see advanced configuration).

23(8)- Sampling start time: year month day hour minute

Default: [simulation start]

Each concentration grid may have a different starting, stopping, and output averaging time. Zero entry will result in setting the default values. "Backward" calculations require that the stop time should come before the start time.

24(9)- Sampling stop time: year month day hour minute

Default: 12 31 24 60

After this time no more concentration records are written. Early termination on a high resolution grid (after the plume has moved away from the source) is an effective way of speeding up the computation for high resolution output near the source because once turned-off that particular grid resolution is no longer used for time-step computations.

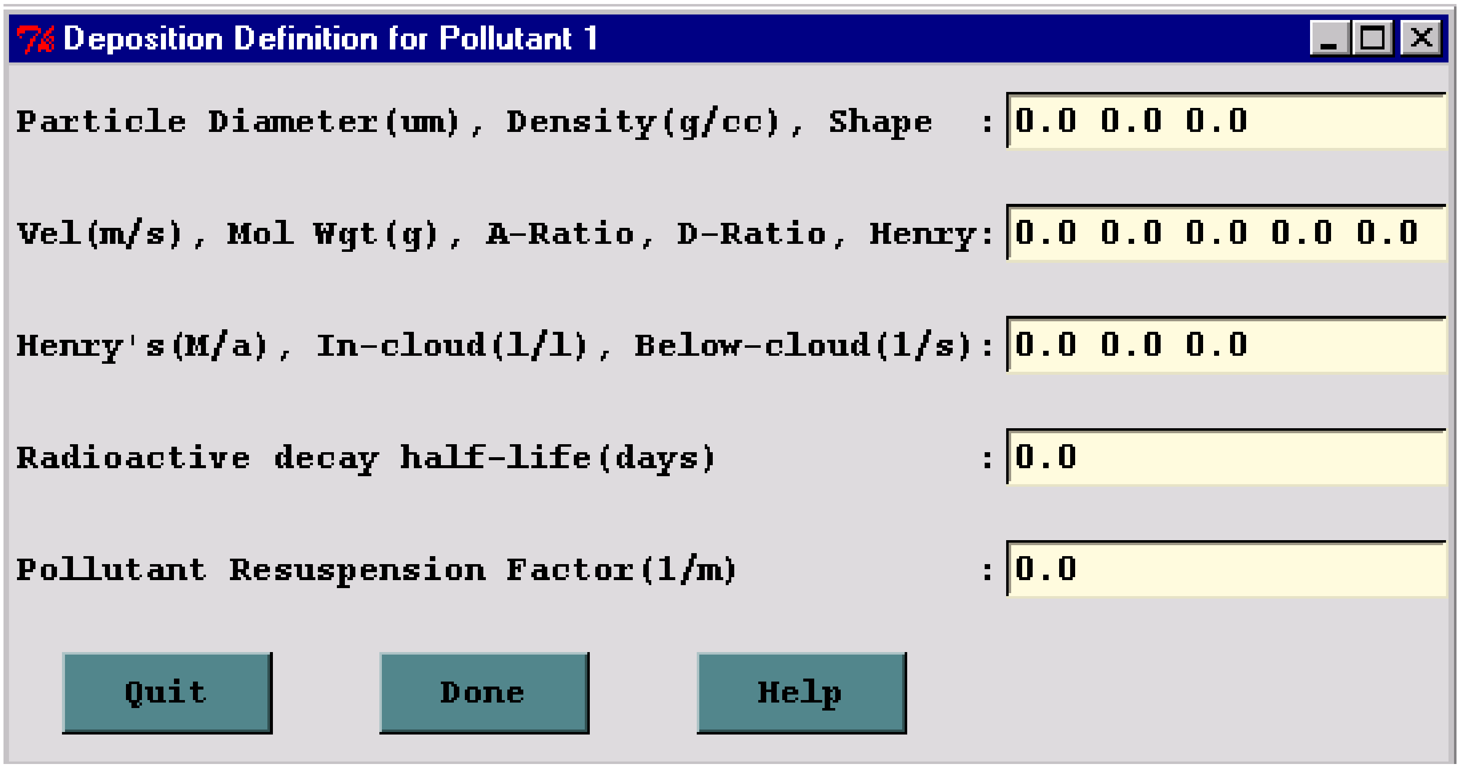
25(10)- Sampling interval: type hour minute

Default: 0 24 0

Each grid may have its own sampling or averaging interval. The interval can be of two different types: averaging (type=0) or snapshot (type=1). Averaging will produce output averaged over the specified interval. Snapshot will give the instantaneous output at the output interval. For instance you may want to define a concentration grid that produces 24-hour average air concentrations for the duration of the simulation which for the default case of a 2-day simulation will result in 2 output maps, one for each day. Each defined grid can have a different output type and interval. There is one special case exception when the snapshot value is less than zero, it represents the averaging time in hours and the output interval time represents the interval at which the average concentration is output. For instance a setting of {-1 6 0} would output a one-hour average concentration every six hours.

CONCENTRATION - Setup Menu - Deposition Definitions

This section is used to define the deposition parameters for emitted pollutants. The number of deposition definitions must correspond with the number of pollutants released. There is a one-to-one correspondence. There are 5 entries in the *CONTROL* file for each defined pollutant. The lines 27(1) through 31(5) correspond with each of the menu items shown in the illustration below.



-

26- Number of pollutants depositing

Default: number of pollutants on line # 10

Deposition parameters must be defined for each pollutant species emitted. Each species may behave differently for deposition calculations. Each will be tracked on its own concentration grid. The following five lines are repeated for each pollutant defined. The number here must be identical to the number on line 10. Deposition is turned off for pollutants by an entry of zero in all fields.

<u>27(1)- Particle: Diameter (µm), Density (g/cc), and Shape</u>

Default: 0.0 0.0 0.0

These three entries are used to define the pollutant as a particle for gravitational settling and wet removal calculations. A value of zero in any field will cause the pollutant to be treated as a gas. All three fields must be defined (>0) for particle deposition calculations. However, these values only need to be correct only if gravitational settling or resistance deposition is to be computed by the model. Otherwise a nominal value of 1.0 may be assigned as the default for each entry to define the pollutant as a particle. If a dry deposition velocity is specified as the first entry in the next line (28), then that value is used as the particle settling velocity rather than the value computed from the particle diameter and density.

28(2)- Deposition velocity (m/s), Pollutant molecular weight (Gram/Mole), Surface Reactivity Ratio, Diffusivity Ratio, Effective Henry's Constant

Default: 0.0 0.0 0.0 0.0 0.0

Dry deposition calculations are performed in the lowest model layer based upon the relation that the deposition flux equals the velocity times the ground-level air concentration. This calculation is available for gases and particles. The dry deposition velocity can be set directly for each pollutant by entering a non-zero value in the first field. The velocity can also be calculated by the model using the resistance method which requires setting the remaining four parameters (molecular weight, surface reactivity, diffusivity, and the effective Henry's constant). See the table below for more information.

Table I. Some typical pollutant constants from Wesely (1989) and Walmsley and Wesely (1996).

Chemical	Symbol	D_{rx}	H^* (M/atm) effective	H(M/atm) Actual	f_o
Sulfur dioxide	SO_2	1.9	$1x10^5$	1.24	0.0
Ozone	O_3	1.6	0.01	0.013	1.0
Nitrogen dioxide	NO_2	1.6	0.01	0.01	0.1
Nitric oxide	NO	1.3	$3x10^{-3}$	$1.9x10^{-3}$	0.0
Nitric acid	HNO_3	1.9	$1x10^{14}$	$2.1x10^5$	0.0
Hydrogen peroxide	H_2O_2	1.4	$1x10^5$	$1.0x10^5$	1.0
Ammonia	NH_3	0.97	$2x10^{14}$	62	0.0
Peroxyacetyl nitrate	PAN	2.6	3.6	5	0.1
Nitrous acid	HNO_2	1.6	$1x10^5$	$2.1x10^5$	0

D_{rx} - Diffusivity ratio; H - Henry's constant; f_o - Surface reactivity ratio

Wesely, M.L., 1989, Parameterizations of surface resistances to gaseous dry deposition in regional-scale numerical models, *Atmos. Environ.*, 23, 1293-1304.

Walmsley, J. L., and M.L. Wesely, 1996, Modification of coded parameterizations of surface resistances to gaseous dry deposition, *Atmos. Environ.*, 30, 1181-1188.

29(3)- Wet Removal: Actual Henry's constant, In-cloud (L/L), Below-cloud (1/s)

Default: 0.0 0.0 0.0

Suggested: 0.0 $3.2x10^5$ $5x10^{-5}$

Henry's constant defines the wet removal process for soluble gases. It is defined only as a first-order process by a non-zero value in the field. Wet removal of particles is defined by non-zero values for the in-cloud and below-cloud parameters. In-cloud removal is defined as a ratio of the pollutant in air (g/liter of air in the cloud layer) to that in rain (g/liter) measured at the ground. Below-cloud removal is defined through a removal time constant.

30(4)- Radioactive decay half-life (days)

Default: 0.0

A non-zero value in this field initiates the decay process of both airborne and deposited pollutants.

31(5)- Pollutant Resuspension (1/m)

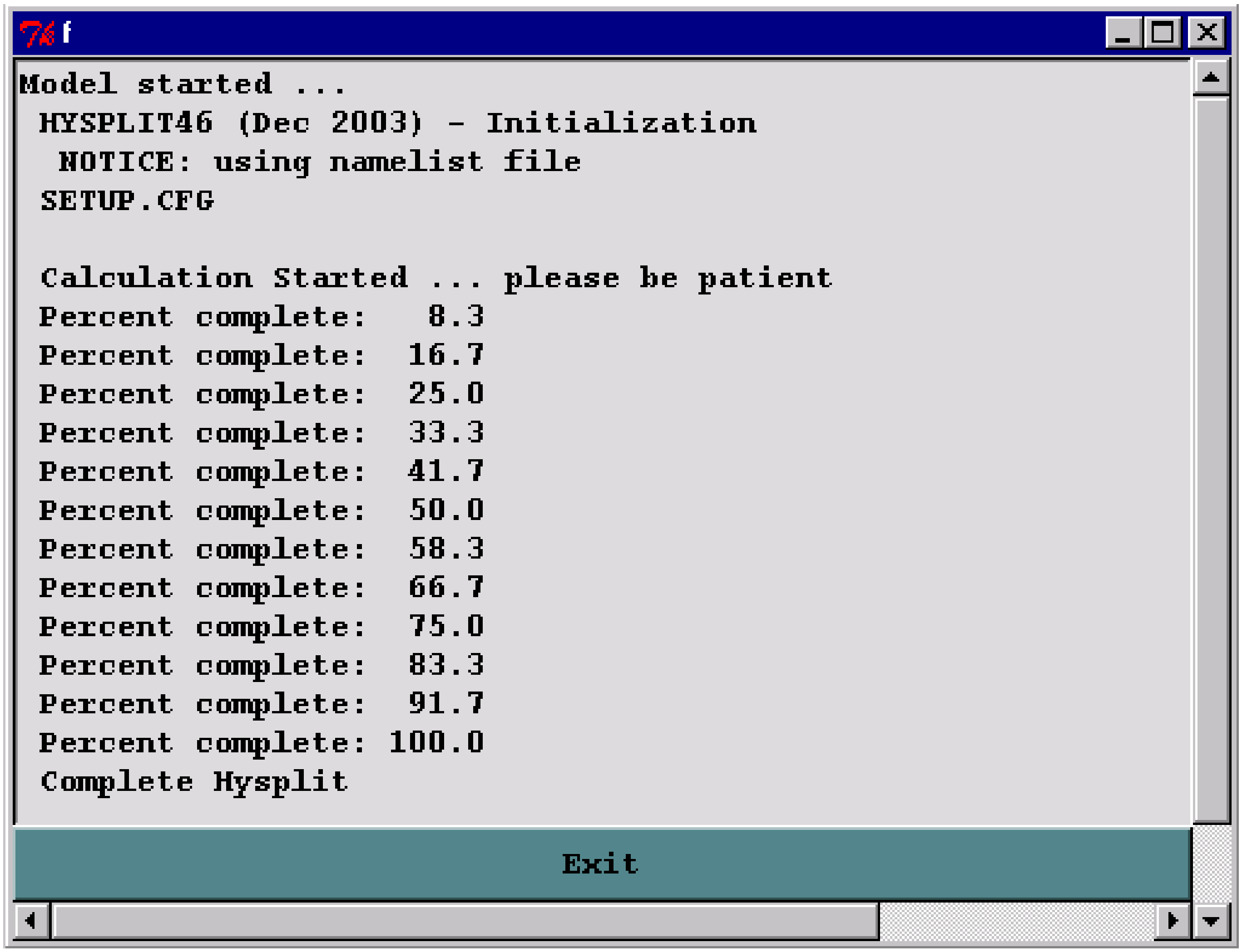
Default: 0.0

Suggested :10^{-6}

A non-zero value for the re-suspension factor causes deposited pollutants to be re-emitted based upon soil conditions, wind velocity, and particle type. Pollutant re-suspension requires the definition of a deposition grid, as the pollutant is re-emitted from previously deposited material. Under most circumstances, the deposition should be accumulated on the grid for the entire duration of the simulation. Note that the air concentration and deposition grids may be defined at different temporal and spatial scales.

CONCENTRATION - Run Standard Model

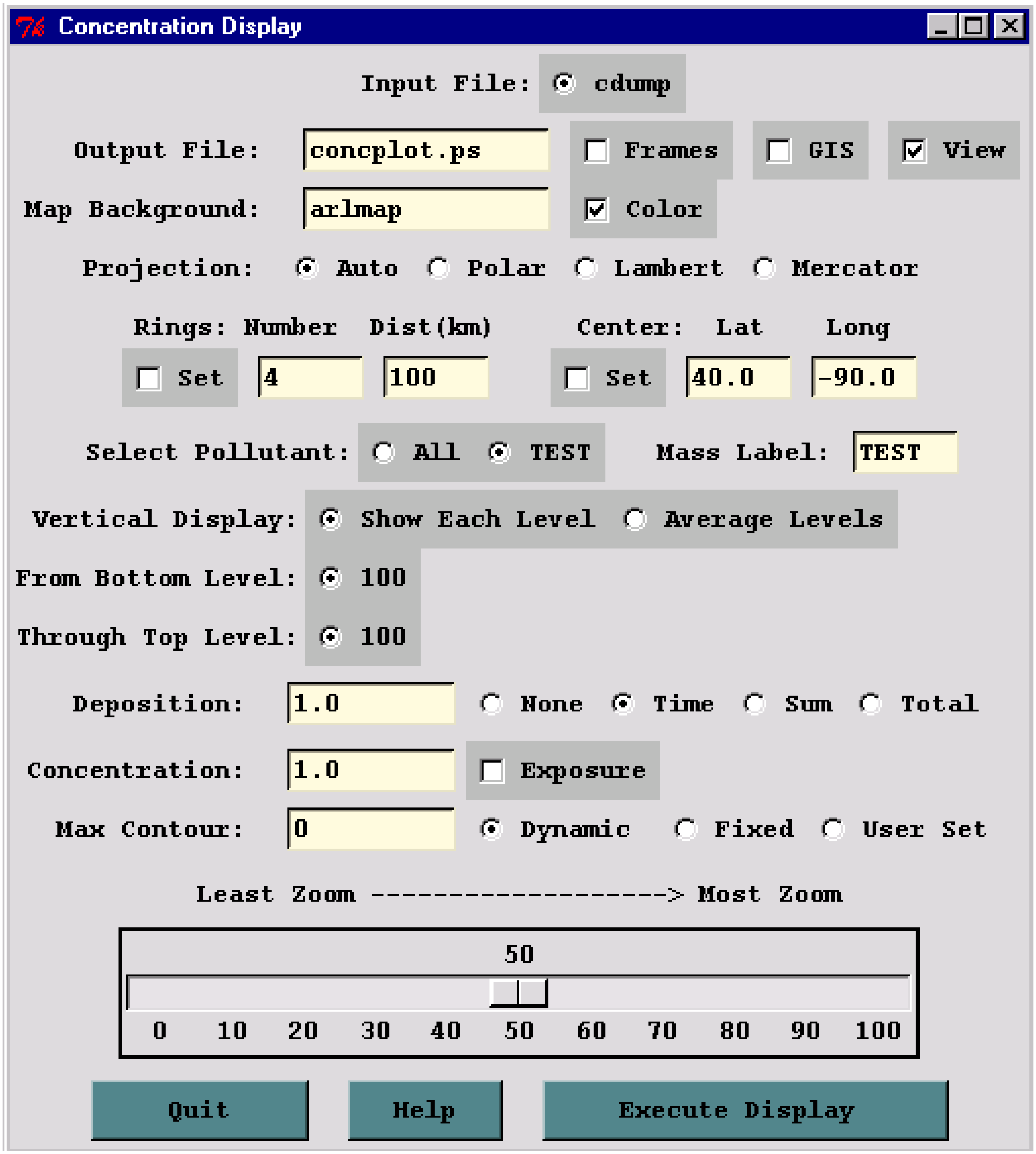
Once the Concentration Setup menu has been closed with the *Save* button, the changes to the simulation parameters are copied to *default_conc*. Clicking on the *Run Standard Model* menu tab first copies *default_conc* to *CONTROL* and then runs the trajectory model executable, *hymodelc*. The executable, by default, attempts to open a file named *CONTROL* to read all the required input parameters. If not found, the model will prompt to standard output for values from standard input. This condition should not occur running the model through the GUI. When the model execution starts, output messages are written to a special window, an example of which is shown in the illustration below:



Successful completion of a simulation will show a similar message. Additional run-time diagnostic messages and other error messages are always written to a file called *MESSAGE*. This file may be viewed through one of the Advanced Menu tabs. Depending upon the nature of the error message, perhaps a failure in the model initialization process, error messages may also appear in above window. Once the model has completed, **press Exit** to close the window.

CONCENTRATION - Display Options - Concentration

The concentration model generates a binary (big-endian) output file on a regular latitude-longitude grid, which is read by the other programs to produce various displays and other output. The plotting program, *concplot*, can be accessed through the GUI, which is shown in the illustration below, or it can be run directly from the command line. Most, but not all, of the command line options are available through the GUI.



Normally only one input file is shown, unless multiple files have been defined in the Concentration Setup menu. The default output file name is shown and unless the box is checked all frames (time periods and/or levels) will be output to that one file. The program uses the map background file, *arlmap*, which by default is located in the *\graphics* directory. Other customized map background files could be defined. Some of these higher resolution map background files are available from the HYSPLIT download web page. For multiple pollutant files, only one pollutant may be selected for display by individual levels, or averaged between selected levels. These levels must have been predefined in the Concentration Setup menu. Multipliers can be defined for deposition or air concentrations. Normally these values would default to 1.0, unless it is desired to change the output units (for instance, g/m^3 to ug/m^3 would require a multiplier of 10^6). Contours can be determined dynamically by the program, changing with each map, or fixed to be the same for all maps. A user can also set a maximum value as the base 10 exponent.

Concplot Command Line Options

The Postscript conversion program (*concplot*), found in the *\concmdl* directory, reads the binary concentration output file, calculates the most optimum map for display, and creates the output file *concplot.ps*. Multiple pollutant species or levels can be accommodated. Most routine variations can be invoked from the GUI. More complicated conversions should be run from the command line using the following optional parameters: concplot -[options (default)]

-a[Arcview GIS: 0-no dump 1-dump to files]

Selecting the GIS checkbox creates an ASCII text file for each output time period that consists of the value and latitude-longitude points along each contour. This file can be converted to an Arcview Shapefile or converted for other GIS applications through the utility menu "GIS to Shapefile" ". The view checkbox would be disabled to do just the GIS conversion without opening the the Postscript viewer.

-b[Bottom display level: (0) m]

= 0 - Represents the height (meters) below which no data will be processed for display. The level information is interpreted according to the display (-d) definition.

-c[Contours: (0)-dynamic, 1-fixed, {X}-10**X]

= 0 - Dynamic contour values are optimized for each map.

= 1 - Fixed contour values are the same for all maps.

= X - Set Limit, where {X} is the integer power of 10 of the maximum contour.

-d[Display: (1)-by level, 2-levels averaged]

= 1 - All output levels that fall between the bottom and top display heights are shown as individual frames. A single level will be displayed if both bottom and top heights equal the calculation level or they bracket that level. Deposition plots are produced if level zero data are available in the concentration file and the display height is set to 0.

= 2 - The concentrations at all levels between the specified range are averaged to produce one output frame per time period. If deposition data is available and a plot is required in addition to the air concentrations, then the bottom height should be set to 0. Deposition is not averaged with air

concentrations.

-e[Exposure units flag: (0)-concentrations, 1-exposure]

= 1 - A custom output format in which all the air concentrations have been converted to time-integrated units and vertically averaged for all levels between the bottom and top heights.

-f[Frames: (0)-all frames one file, 1-one frame per file]

= 0 - All output frames (one per time period) in one file.

= 1 - Each time period is written to a file: concplot{frame number}.ps

-g[Graphic circle overlay: ( )-auto, 0-numb, numb:dist(50) km]

= ( ) - Auto selection procedure draws four circles with the distance between them determined by the program algorithm.

= # - Specifies the number of circles with the default (50 km) distance interval.

= number:distance - specifies the number of circles and the distance interval between circles. For the special case of zero circles with a distance specified (e.g. -0:1500) the program will fix the map with the top and bottom edge at that distance from the center.

-h[Hold map at center lat-lon: (source point), lat:lon]

= lat:lon - Forces the center of the map to be at the specific latitude-longitude point rather than the default source location. This is normally used in conjunction with the -g option to get the same map each time or when there are multiple source locations.

-i[Input file name: (cdump)]

= Name of the binary concentration file for which the graphics will be produced. Default name {cdump} or user defined}.

-j[Graphics map background file name: (arlmap)]

The program first searches the local directory, then the ..\graphics directory for the name of the default map background file (arlmap). Set this parameter to select the directory/name of any map background file of compatible format.

-k[Kolor: 0-B&W, (1)-Color]

= 0 - Uses gray shade patterns for the contour color fill.

= 1 - Uses the default four color fill pattern.

-l[Label options: ascii code, (73)-open star]

The default plot symbol over the source location is an open star. This may be changed to any value defined in the psplot ZapfDingbats library. For instance a blank, or no source symbol would be defined as -l32

-m[Map projection: (0)-auto, 1-polar, 2-lambert, 3-mercator]

Normally the map projection is automatically determined based upon the size and latitude of the concentration pattern. Sometimes this procedure fails to produce an acceptable map and in these situations it may be necessary to force a map projection.

-n[Number of time periods: (0)-all, number, min:max, -increment]

= 0 - All time periods in the input file are processed.

= # - Sets the number of time periods to be processed starting with the first.

= #1:#2 - Processes time periods, including and between #1 and #2.

= [-#] - Sets the increment between time periods to be processed. For instance -n-6 would only process every 6^{th} time period.

-o[Output file name: (concplot.ps)]

The name of the Postscript output file defaults to concplot.ps unless it is set to a {user defined} value.

-p[Process file name suffix: (ps)]

The suffix defines the character string that replaces the default "ps" in the output file name. A different suffix does not change the nature of the file. It remains Postscript. The suffix is used in multi-user environments to maintain multiple independent output streams.

-q[Quick data plot: ( )-none, filename]

By defining a text file with a comma or space delimited format consisting of one or more records of the four real numbers, identity, latitude, longitude, and value, the program will plot the values at the given locations. The same values are plotted on all maps.

-r[Removal: 0-none, (1)-each time, 2-sum, 3-total]

= 0 - No deposition plots are produced even if the model produced deposition output.

= 1 - One deposition plot is produced for each time period.

= 2 - The deposition is summed such that each new time period represents the total accumulation.

= 3 - Similar to =2, deposition is accumulated to the end of the simulation and but only one plot is produced at the end.

-s[Species: 0-Sum (1)-Single Pollutant {N}-Species Number]

= {Species Number} - Only one pollutant species may be displayed per plot sequence if multiple species were created during a simulation. However, an entry of "0" will cause all species concentrations to be summed for display.

-t[Top display level: (99999) m]

= 99999 - Represents the height (m) above which no data will be processed for display. The level is interpreted according to the display definition.

-u[Units label for mass: (mass), also see "labels.cfg" file]

Defines the character string for the units label. Can also be modified using the labels.cfg file.

-v[Values for fixed contours: val1+val2+val3+val4]

If the contour values are fixed for all plots (-c1), then it is also possible to define the four individual contours explicity through this option. For instance -v4+3+2+1, would define the contours 4, 3, 2, and 1.

-w[Weight for contour smoothing: (0.25), none=0.0]

= 0.25 - default

= X - User selected value

= 0.0 - none

-x[Concentration multiplier: (1.0)]

= 1.0 - No units conversion.

= X - where {X} is the multiplier applied to the air concentration input data before graphics processing.

-y[Deposition multiplier: (1.0)]

= 1.0 - No units conversion.

= X - where {X} is the multiplier applied to the deposition input data before processing.

-z[Zoom factor: 0-least zoom, (50), 100-most zoom]

= 50 - Standard resolution.

= 100 - High resolution map (less white space around the concentration pattern)

Additional Map Label Customization

Many of the Postscript graphics programs have label information that can be customized to some extent. This is accomplished by placing a file called *Labels.cfg* in the startup directory which contains the following valid entries (all in single quotes terminated by &) replacing the new string with the desired text. A sample file called *Labels.bak* may be found in the relevant directory. Not all label strings are valid with every plotting program.

'TITLE&','NATIONAL WEATHER SERVICE&'

'MAPID&','PROBABILITY&'

'LAYER&','BOUNDARY LAYER AVERAGE&'

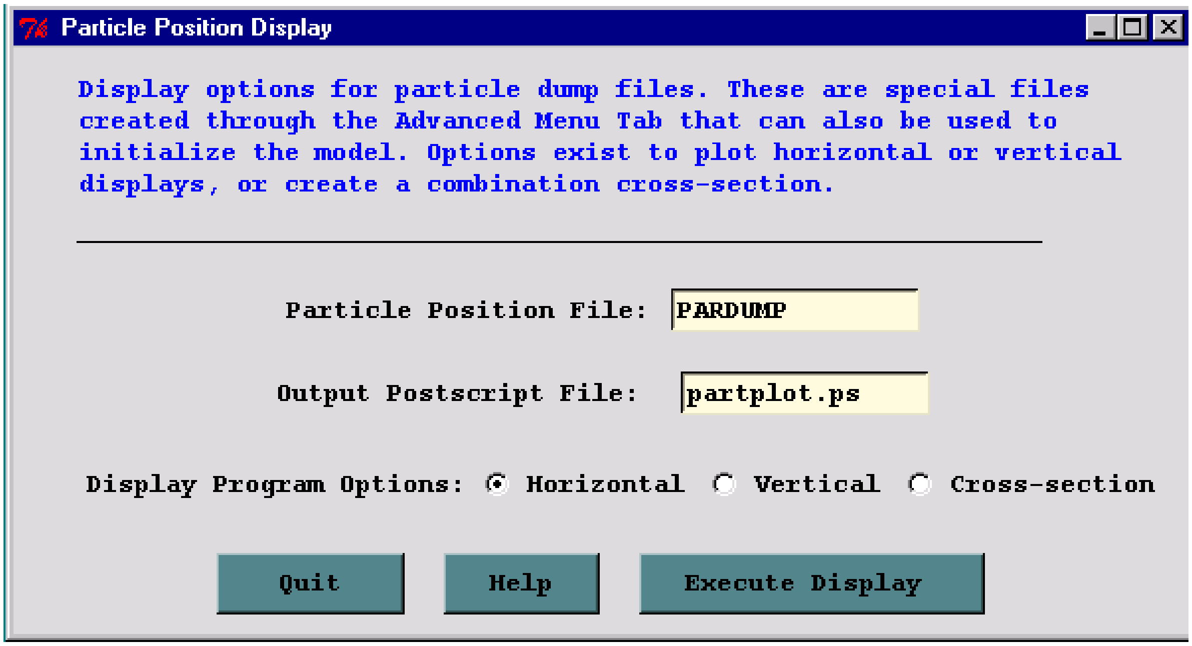
'UNITS&','BQ&'

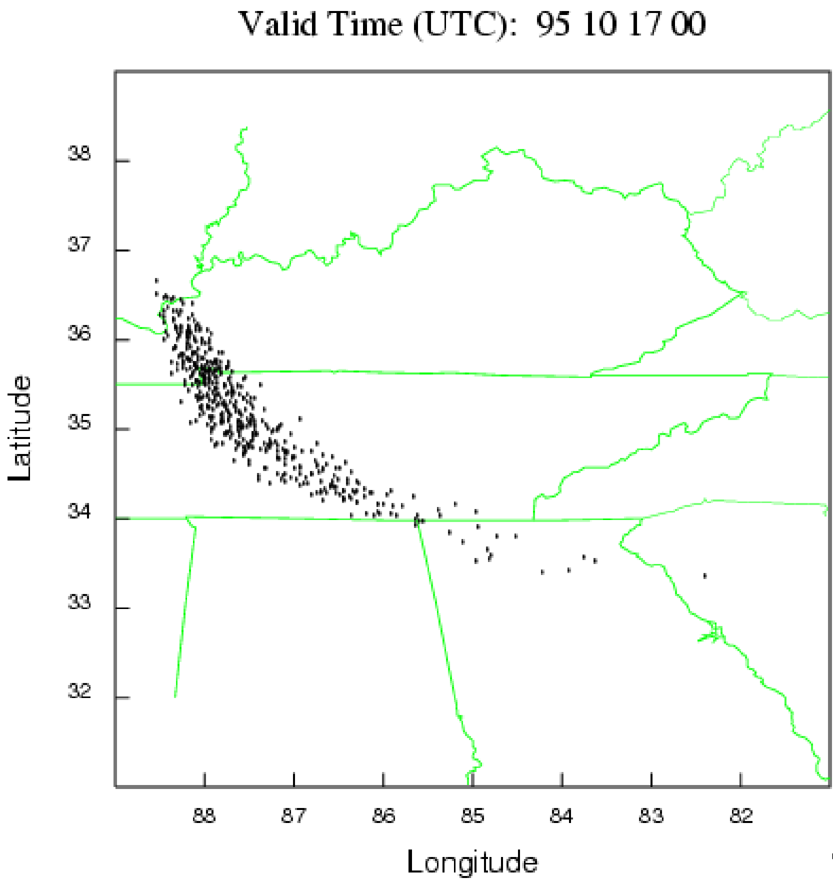
'VOLUM&','/M3&'

Additional supplemental text may be added at the bottom of the graphic by creating a file called *MAPTEXT.CFG*, also to be located in the startup directory. This is a generic file used by all plotting programs but each program will used different lines in its display. The file can be created and edited through the Advanced menu tab.

CONCENTRATION - Display Options - Particles

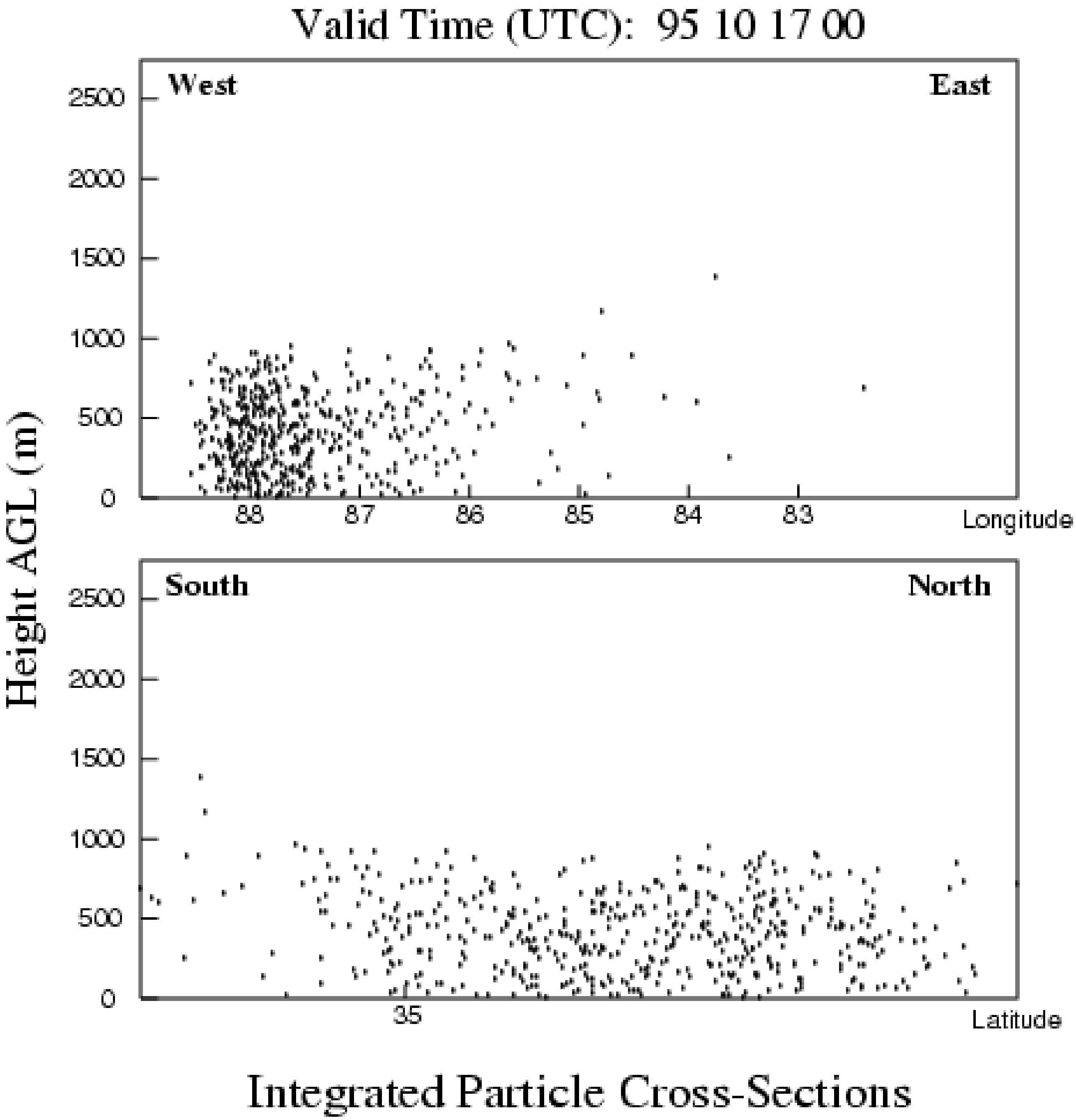
In addition to the normal display programs for air concentrations, which are designed to display and contour the actual values, the particle display programs only show the instantaneous positions of the pollutant particles or puffs that are integrated in time by the model to produce the air concentration fields. To generate the particle position graphics from this menu it is necessary to generate a particle dump file (default name: PARDUMP), which may contain one or more time periods of output. The creation of this file is controlled by the parameters set in the menu tab: "Advanced / Configuration Setup / Concentration." The particle display menu has only three choices, the name of the input file, the Postscript output file, and the type of display requested.



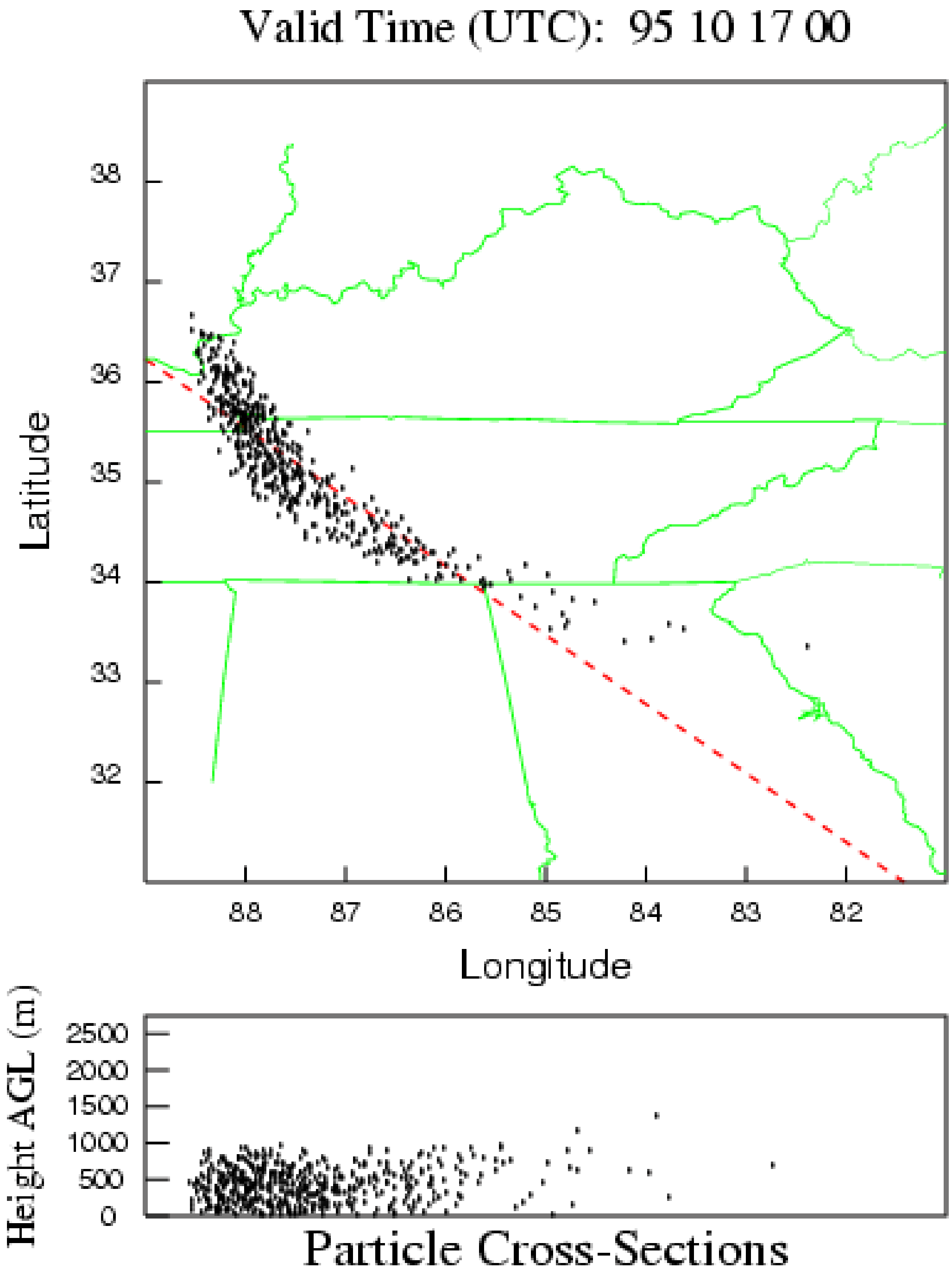


The "Horizontal" option results in the conventional graphic shown below. There is one black dot for each particle. The size of the dot varies according to the pollutant mass assigned to the particle. All examples are 24 hours after the start of the test simulation.

The "vertical" option shows an integrated particle distribution view from the south looking north (top panel) and from the east looking west (bottom panel). All particles in the computational domain are shown.



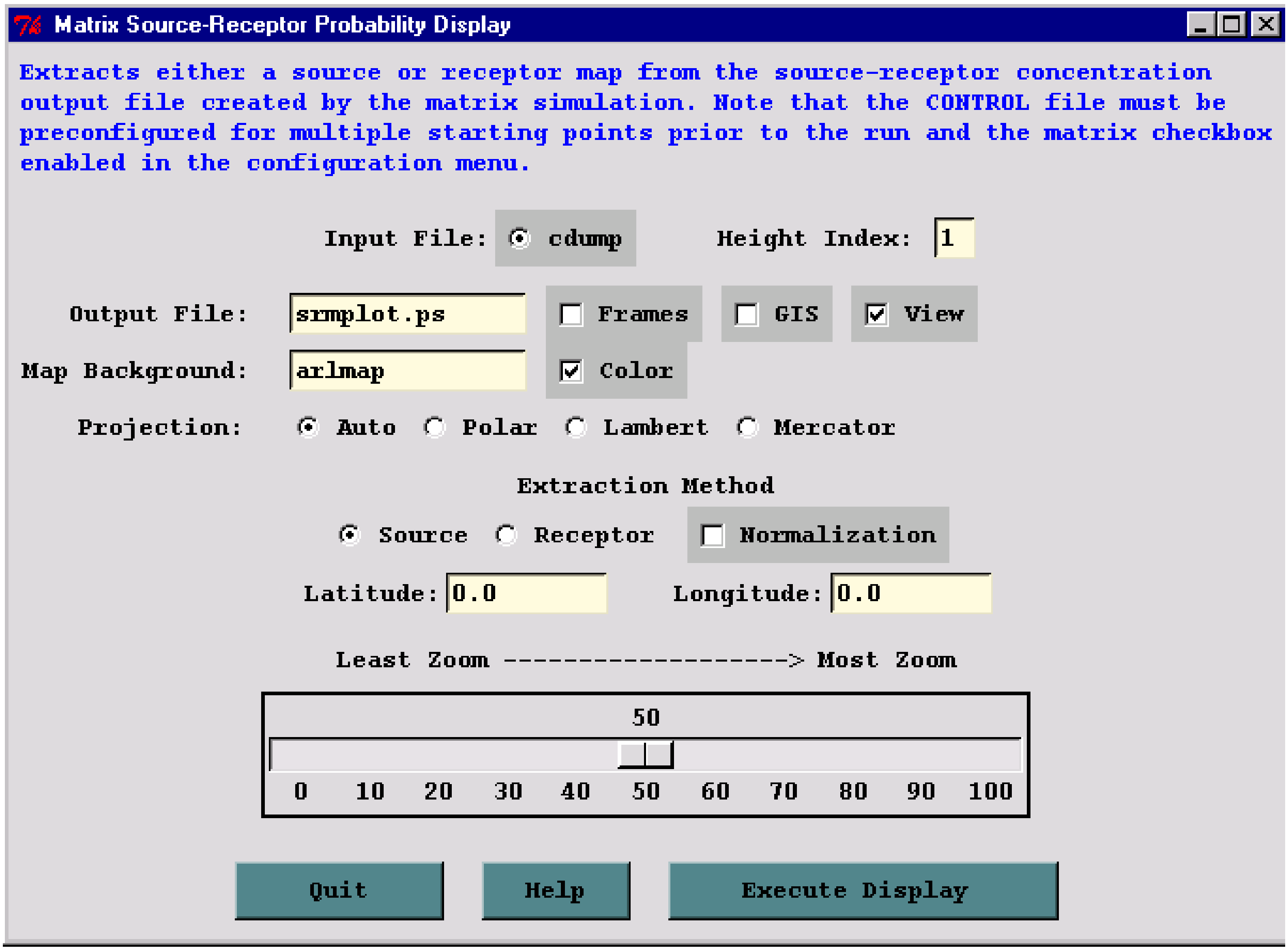
The "cross-section" view is a combination of the horizontal and vertical views. The top panel shows the horizontal particle distribution, while the bottom panel shows the vertical distribution along the red regression line through the plume. Again all particles are shown regardless of their distance from the regression line. The bottom panel view is from left to right in increasing longitude, regardless of the orientation of the regression line.



Valid Time (UTC): 95 10 17 00
Latitude
38
37
36
35
34
33
32
88
87
86
85
84
83
82
Longitude
Height AGL (m)
2500
2000
1500
1000
500
0
Particle Cross-Sections

CONCENTRATION - Display Options - Matrix

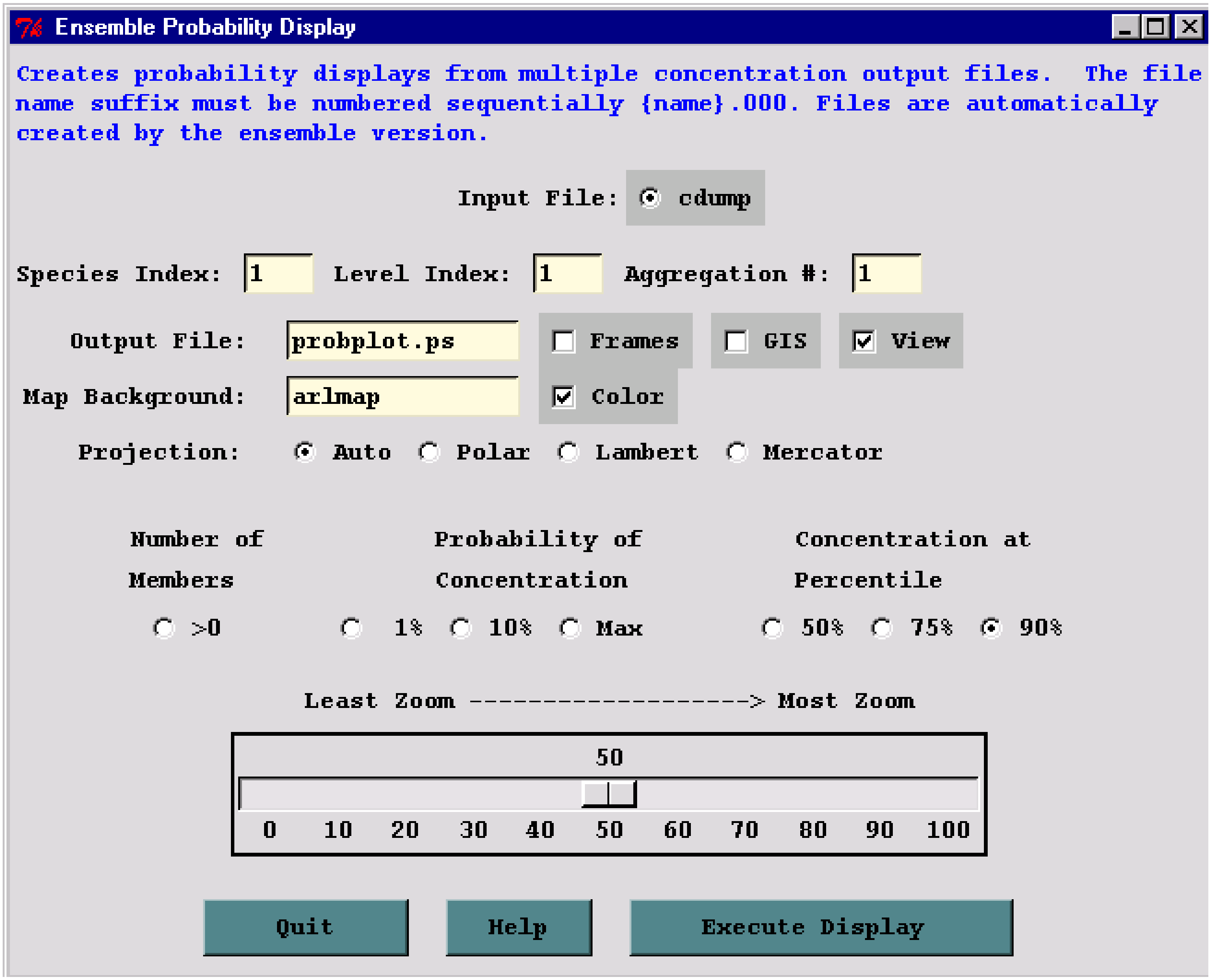
This menu option permits the extraction of information for a specific source or receptor if the original model simulation was configured to produce a source-receptor matrix formatted output file. The model should have been run with the matrix option set in the configuration menu. This results in a special concentration output file that may be called a source-receptor matrix, such that each column may be considered a receptor location and each row a pollutant source location. The display program under this menu tab permits the contouring of any row or column. If the matrix option was not set in the Advanced Concentration configuration tab, then the output file produced is a simple concentration output file, where the concentrations from all the sources of the defined matrix are summed on a single concentration output grid. Source-receptor information cannot be extracted from such a file. When a location is selected in the menu, a special program is called to extract that location from the concentration output file and then write a standard concentration file for that location. The standard concentration display program cannot extract the source-receptor information from the concentration matrix output file but it can be used to display the information in the extracted file. The source-receptor matrix extraction file name will consist of *SRM_{original file name}*. The *Display Matrix* menu, shown in the illustration below, automatically creates the extraction file, calls the standard display program *concplot*, and provides for additional custom label information to identify the source-receptor concentration matrix.



The menu consists of the standard display options such as the map background file, output file name, and zoom factor, plus the selection of the extraction method: source or receptor. A latitude and longitude point needs to be entered for all extractions. Selection of the *"source"* extraction method means that the location entered is a receptor and the output map is a contour map of how much air concentration each source contributes to the selected receptor. The *"receptor"* extraction method means that the location entered is a source location and the output map is a contour map of air concentrations from that source. The *"receptor"* option is just a conventional air concentration simulation. Note that turning on the "normalization" flag divides all concentrations by the sum of all concentrations at each grid point, resulting in a map that represents a fractional contribution.

CONCENTRATION - Display Options - Ensemble

Multiple concentration output files from an ensemble dispersion simulation can be processed to produce probability displays. The *Ensemble Display* menu calls a special program *probval* that reads the concentration files with the ensemble member three-digit suffix (001 to 027) and generates various probability files. The appropriate files are then plotted using *concplot* according to the menu selection options. Note that the ensemble display menu can be used to convert multiple concentration output files to probability form, regardless of how they were generated. It is not necessary to use the pre-configured ensemble version of the model to create air concentration probability maps. Any number of multiple simulations can be used as long as the file suffixes end with a numeric three-digit sequence. Menu settings for the output file name should reflect the base name without the numeric suffix. An illustration of the *Ensemble Display* menu is shown below.



The menu includes some of the standard display program (*concplot*) mapping choices, such as background file, output file name, and zoom factor. One critical difference is that there is an option called the *Number of aggregation periods*. By default this value is set to one, which means that only the ensemble members for one time period are aggregated together to produce the probability display. Hence there would be an independent probability map for each time period. However, multiple time periods may be combined to produce a single probability plot. The entry represents the number of time periods, not the actual time span. For instance, if the model output is set to produce a one-hour average and the model is run for 24 hours, then each ensemble member will have 24 time periods of output. By default the probability output will represent the ensemble member variation for each hour, that is 24 frames of output will be produced. However if the aggregation period is set to 24, then all output times will be combined into one output frame and the ensemble result will represent the hourly variations as well as the member variations.

The menu permits a choice of three different ensemble display options:

1. **The number of ensemble members at each grid point with concentrations greater than zero.**
2. Shows the spatial distribution of the *Number of Ensemble* members.
3. **The probability of concentration**
4. The *Probability of Concentration* produces contours that give the probability of exceeding a fixed concentration value at one of three levels: 1% of the maximum concentration, 10% of the maximum, and the maximum concentration. The maximum is determined to be the first concentration to a power of 10 that is less than the actual maximum value. The concentration level for the probability display is shown on the graphic with the pollutant identification field set to something like C14, where 14 represents the Concentration to the power of 10^{-14}.
5. **The concentration at different preset percentiles.**

The *concentration at a percentile* shows contours of areas that concentrations will be exceeded only at the given probability level. The probability level choices are 50, 75, and 90 percent.

The *probval* extraction program will always produce all the probability variations, one to each output file, for a given aggregation period. These files may be plotted individually from the command line using *concplot*. The *probval* output files contain the following information:

Cnum00 - Number of members

Cmax01 - Probability of 1% of the concentration maximum

Cmax10 - Probability at 10% of the concentration maximum

Cmax00 - Probability at the maximum concentration level

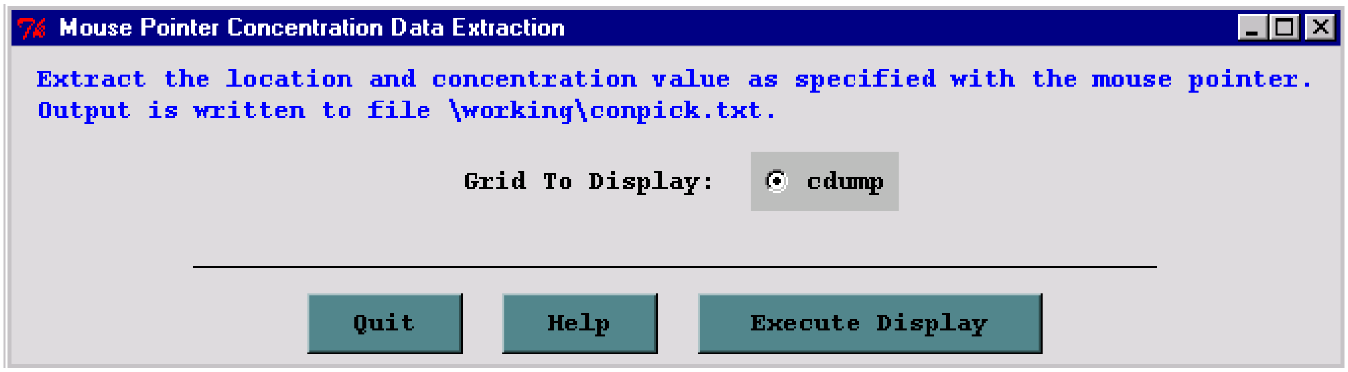
Prob50 - Concentrations at the 50^{th} percentile

Prob75 - Concentrations at the 75^{th} percentile

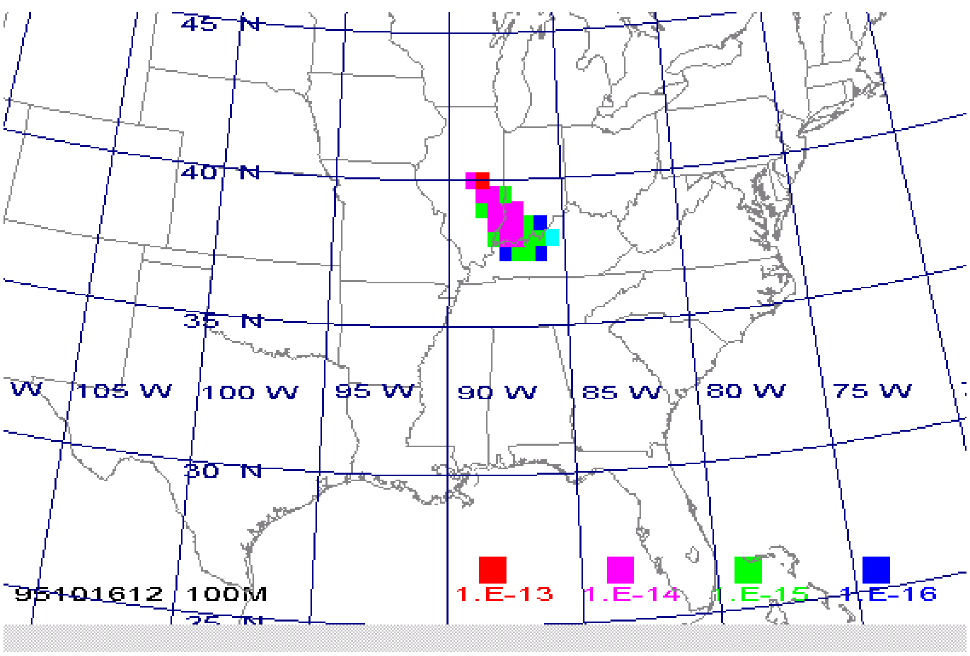
Prob90 - Concentrations at the 90^{th} percentile

CONCENTRATION - Display Options - Pointer Select

The Windows-only concentration data point registration program is designed to read the binary concentration data file, display the data on the screen, and then use the mouse as a pointer device to select locations at which the position and concentration data are read and written to the text file: *wincpick.txt*. The concentration data are displayed over the entire grid domain. There are no zoom options. If you want to zoom in on a specific area, then it is necessary to rerun the model with a smaller concentration grid domain. The program is available from the GUI or the command line with the following syntax: wincpick [input file name]. The GUI menu, which is shown in the illustration below, only permits a selection of the concentration input file name, which must have been previously defined in the Concentration Setup menu.

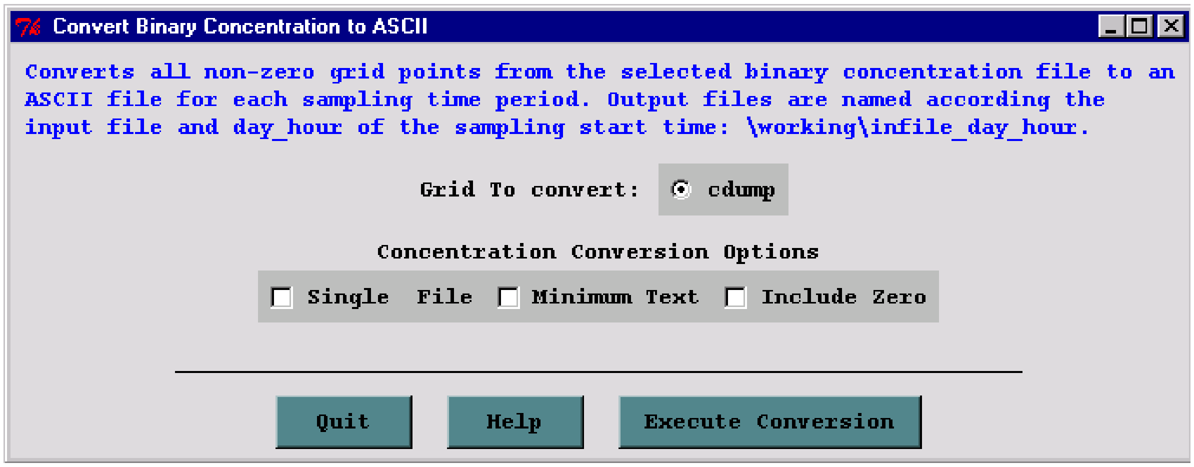


Upon startup, *Wincpick* will display the domain background map with a summary of the mouse-based instructions: **left-click** registers the latitude-longitude position of that point to the output file *wincpick.txt*. A **right-click** of the mouse redraws the map for the next time period and at the end of the input file saves and closes all files; and a **CNTL-right-click** closes all files and exits the program before the end of the input file. **Right-click** the mouse to go to the first concentration map. Maps are drawn in sequence of height, pollutant species, and time. **Left-clicks** register the position of the mouse to the output file. If you are interested in only one time period, skip past those times by a **right-click**, register the points of interest, then **CNTL-right-click** to exit. An illustration of the first output period from the sample calculation is shown below.



CONCENTRATION - Utility Programs - Convert to ASCII

The Convert to ASCII menu option uses the con2asc program to convert the binary concentration file to a simple ASCII file composed of one record per grid point for all grid points where concentrations at any level are non-zero. Concentrations for multiple levels and pollutant species are all listed on the same record for each grid point. The primary purpose of the conversion is to create a file that can be imported into other applications. An illustration of the GUI menu is shown below for the sample concentration simulation.



The Concentration Setup menu determines the file name selection option on the GUI. There are some additional checkboxes that correspond to various command line conversion options: con2asc -[options (default)].

-c[Convert entire file flag]

This option converts the entire binary input file, including all index records, to an ascii output file with the name {input file}.txt. This option corresponds to the *Single File* checkbox of the GUI menu. The output file format follows the binary file format record-by-record using the following format conventions.

Meteorological model and starting time - A4, 6I4

Starting time and locations - 4I4, 2F8.3, F10.1

Concentration grid and spacing - 2I4, 2F8.4, 2F8.2

Vertical grid index record - I4, 20I8

Pollutant identification record - I4, 20A4

Sample start time - 6I4

Sample end time - 6I4

Concentration record - A4, I6, 255(255E10.2)

-i[Input file name (cdump)]

-m[Minimum output format flag]

Setting this flag turns off the writing of the first output record, which is the column label field: DAY HOUR LAT LON SPECIES-LEVEL.

-o[Output file name base (Input File Name)]

The default base name for the output file is the input file name. A new output file is created for each sampling period, where the name of the file is composed of the *{base name}_{Julian day}_{hour}* of the sample ending time.

-z[Zero points output flag]

Setting the zero flag causes the program to output the concentration values all all grid points, including the ones that are zero.

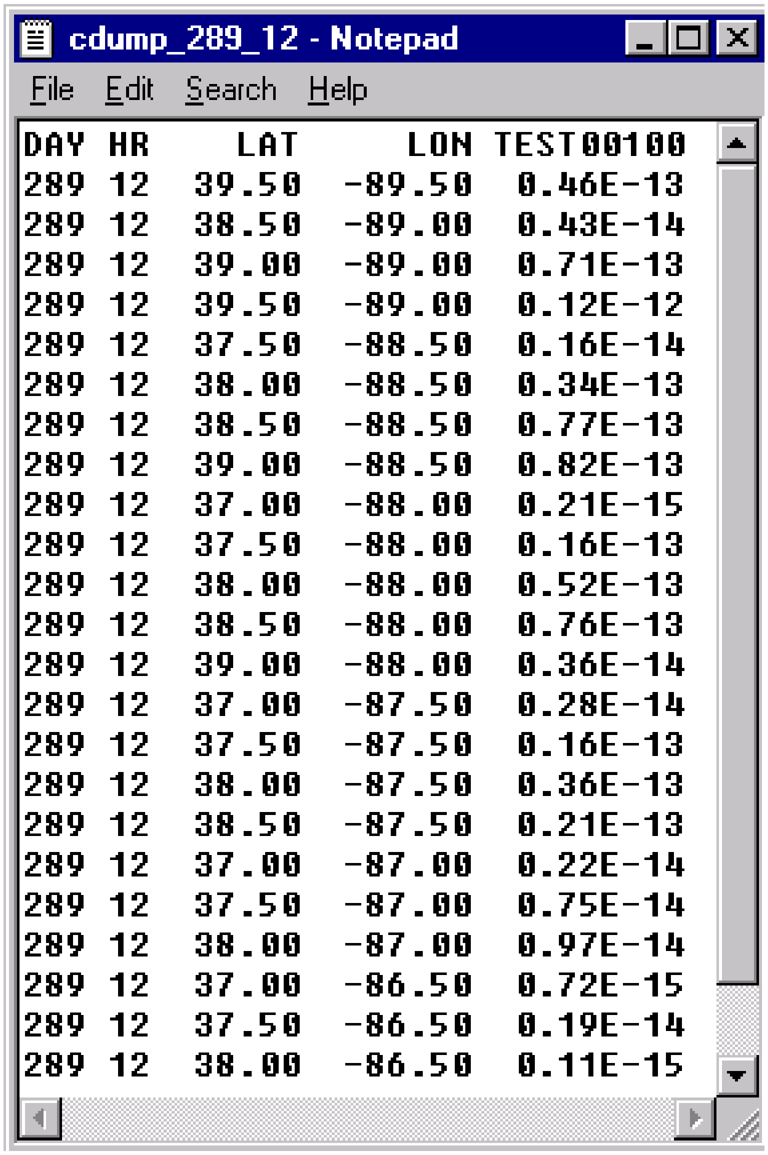
The format of each record in the output file is given by:

2I3 - End of Sample: Julian Day and Hour

F7.2, F8.2 - Latitude and Longitude of grid cell

45E9.2 - Concentration data (by level and pollutant)

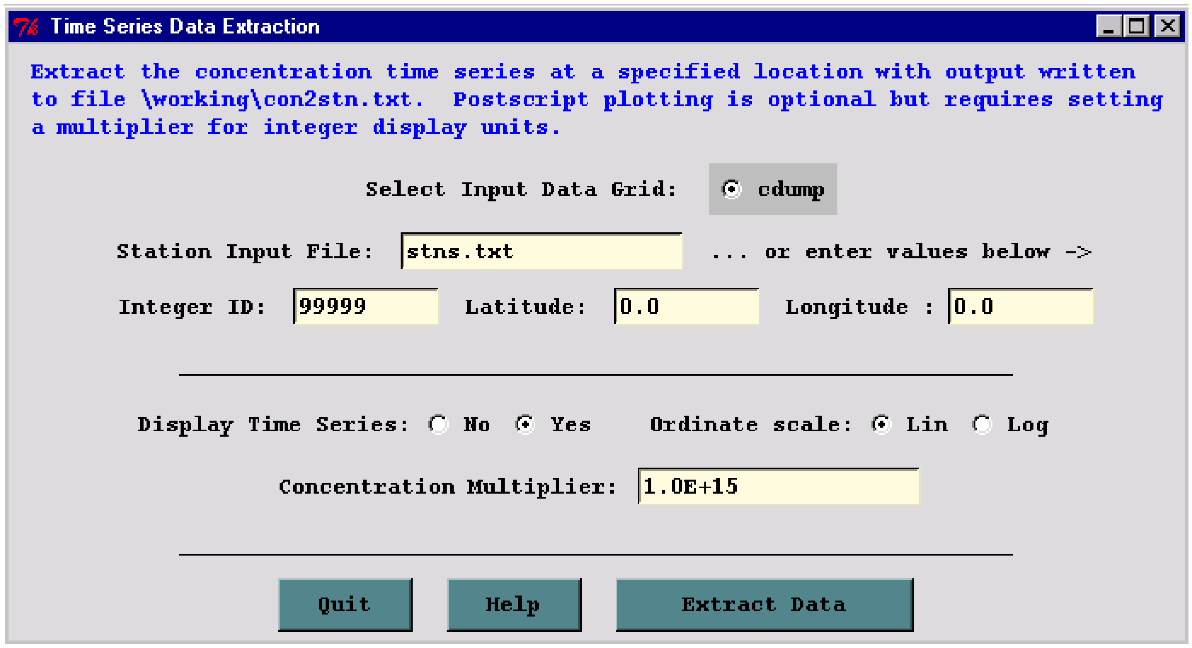
Each output record is identified by the day (Julian: 1 to 365) and hour (UTC) of the ending time of each sample. The ASCII conversion of the first file generated by the sample calculation is shown below in the illustration.



DAY	HR	LAT	LON	TEST00100
289	12	39.50	-89.50	0.46E-13
289	12	38.50	-89.00	0.43E-14
289	12	39.00	-89.00	0.71E-13
289	12	39.50	-89.00	0.12E-12
289	12	37.50	-88.50	0.16E-14
289	12	38.00	-88.50	0.34E-13
289	12	38.50	-88.50	0.77E-13
289	12	39.00	-88.50	0.82E-13
289	12	37.00	-88.00	0.21E-15
289	12	37.50	-88.00	0.16E-13
289	12	38.00	-88.00	0.52E-13
289	12	38.50	-88.00	0.76E-13
289	12	39.00	-88.00	0.36E-14
289	12	37.00	-87.50	0.28E-14
289	12	37.50	-87.50	0.16E-13
289	12	38.00	-87.50	0.36E-13
289	12	38.50	-87.50	0.21E-13
289	12	37.00	-87.00	0.22E-14
289	12	37.50	-87.00	0.75E-14
289	12	38.00	-87.00	0.97E-14
289	12	37.00	-86.50	0.72E-15
289	12	37.50	-86.50	0.19E-14
289	12	38.00	-86.50	0.11E-15

CONCENTRATION - Utility Programs - Convert to Station

The purpose of the *Convert to Station* utility menu is to list concentrations at specific latitude-longitude locations by extracting that information to a text file. The menu also has an option to produce a time series plot at one or more stations. An illustration of the menu is shown below.



Similar to the other menus in the utility section, the input concentration file must be defined by the Concentration Setup menu. The menu options correspond to the command line options of the *con2stn* extraction program. There are two options that can be used to define the extraction location. A station location can be defined directly as an entry in the menu, or a list of stations can be defined in an input file. In this example illustration, a file has been defined with three locations that are within the plume of the example simulation. The file consists of three records, one for each station:

```
001 39.0 -89.0
002 38.0 -88.0
003 37.0 -87.0
```

The extraction program is called *con2stn* and for these three stations produces the output file shown below, called by default *con2stn.txt*. In contrast to the simulation shown in all the previous examples, in this case the output averaging time was decreased from 12 hours to one hour, to generate a smoother looking graphic.

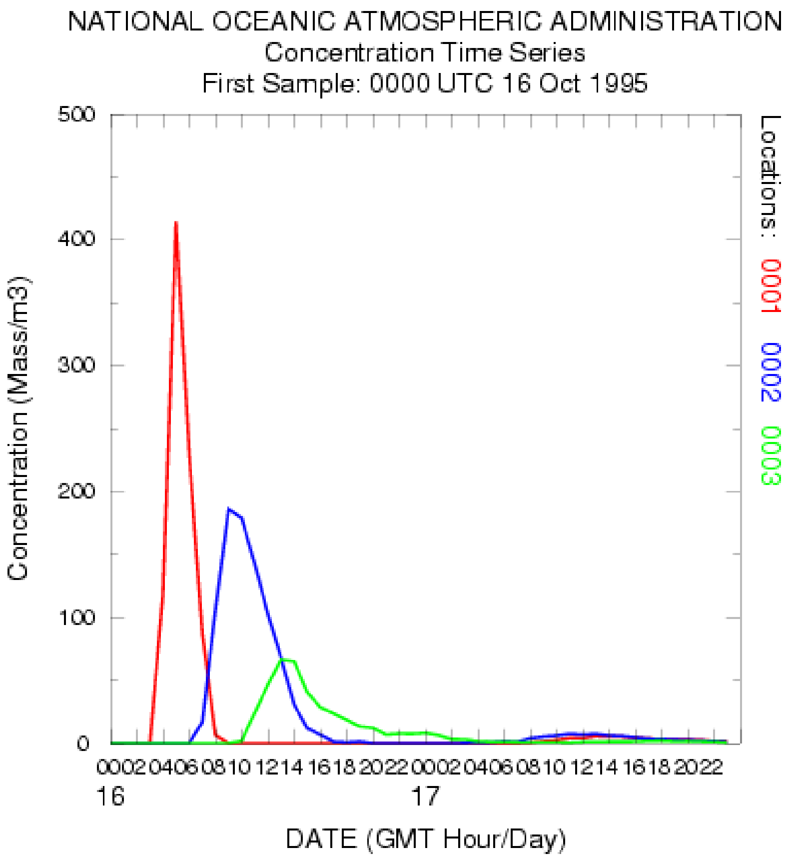
JDAY	YR	MO	DA1	HR1	DA2	HR2	1	2	3
289.000	95	10	16	0	16	1	0.	0.	0.
289.042	95	10	16	1	16	2	0.	0.	0.
289.083	95	10	16	2	16	3	0.	0.	0.
289.125	95	10	16	3	16	4	0.	0.	0.
289.167	95	10	16	4	16	5	117.701	0.	0.
289.208	95	10	16	5	16	6	414.454	0.	0.
289.250	95	10	16	6	16	7	230.186	0.	0.
289.292	95	10	16	7	16	8	87.431	16.723	0.
289.333	95	10	16	8	16	9	6.572	105.681	0.
289.375	95	10	16	9	16	10	0.	186.133	0.
289.417	95	10	16	10	16	11	0.	178.893	2.082
289.458	95	10	16	11	16	12	0.	142.384	24.254
289.500	95	10	16	12	16	13	0.	102.709	46.464
289.542	95	10	16	13	16	14	0.	68.016	66.541
289.583	95	10	16	14	16	15	0.	30.602	65.018
289.625	95	10	16	15	16	16	0.	11.883	40.429
289.667	95	10	16	16	16	17	0.	6.797	28.568
289.708	95	10	16	17	16	18	0.	1.368	23.695
289.750	95	10	16	18	16	19	0.	0.778	18.564
289.792	95	10	16	19	16	20	0.	1.482	13.358
289.833	95	10	16	20	16	21	0.	0.	12.230

The output file shows the Julian day, month, day, and hour of the sample start; day and hour of sample ending time, and the concentrations for each station (location selected by latitude-longitude). The format of each output record is as follows:

F8.3, 6I4 - Starting: Julian day, year, month, day, hour; Ending: day, hour

XF10 - Concentration value at X stations

The lower section of the GUI is used to create a simple time series concentration plot of the concentration time series. One or more stations may be plotted and the option is also available through the command line. The option is selected from the *Display Postscript Time Series* checkbox. The program, *timeplot*, reads the data file produced by the *con2stn* conversion program and plots the concentration values to the *timeplot.ps* output file. The illustration for the previous text file is shown below.



There are only two plot options supported through the GUI: linear or logarithmic ordinate scaling. In the linear scaling case, as shown above, the ordinate units must be integer whole numbers, and the minimum value will always be zero. Therefore it may be necessary to specify a units conversion factor, in this case 10^{15}, to create data in the text file that can be plotted. With the log scaling option, the conversion factor can be set to 1.0, and the ordinate scale will cover the appropriate order-of-magnitude range to plot all the data.

Command Line Options - con2stn

The program can be run from the command line, through interactive prompts from the keyboard. The command line argument syntax is that there should be no space between the switch and options. The command line arguments can appear in any order.

con2stn -[option {value}]

-i [input concentration file name: (std input)]

-o [output concentration file name: (std output)]

-c [input to output concentration multiplier: (1.0)]

-s [station list file name: (std input)]

-x [n(neighbor) or i(interpolation): n]

-z [level index: 1]

-p [pollutant index: 1]

-m [maximum number of stations: 200]

Unspecified file names will result in a standard input prompt. The default interpolation method (-xn) is to use the value at nearest grid point to each latitude-longitude position. The station positions can be read from a file (space or comma delimited) with the first field being an integer that represents the location identification, followed by the location latitude and longitude. Level and pollutant index values can be selected for files with multiple levels and species.

Examples:

1) con2stn ... Results in prompts -->

Enter input concentration file name...

[name of hysplit output file]

Enter sampler ID#, latitude, longitude ...

[integer sample ID, real latitude, real longitude]

0 0 0 (to terminate input)

2) Read the model output file 'cdump' and write text output to file 'clist.txt' for station #517 at 53N 85W.

con2stn -icdump -oclist.txt

517 53.0 -85.0

0 0 0

3) As in 1) but multiply all concentrations by 1000.0

con2stn -icdump -oclist.txt -c1000.0

517 53.0 -85.0

0 0 0

4) As in 1) but linear interpolate concentration to station rather than using the nearest grid point

con2stn -icdump -oclist.txt -xi

517 53.0 -85.0

0 0 0

5) As in 1) but read the station lat-lon from a file "stns.txt".

con2stn -icdump -oclist.txt -sstns.txt

Command Line Options - timeplot

timeplot -[option {value}]

-i[input concentration file name that contains data in the same format as output from *con2stn*]

-n[sequential station number - for files with multiple stations select the station to plot; default for multiple stations is to plot all stations; several stations can be selected to plot by appending station numbers with the plus symbol: hence -n3+5+6 will plot stations 3, 5, and 6.

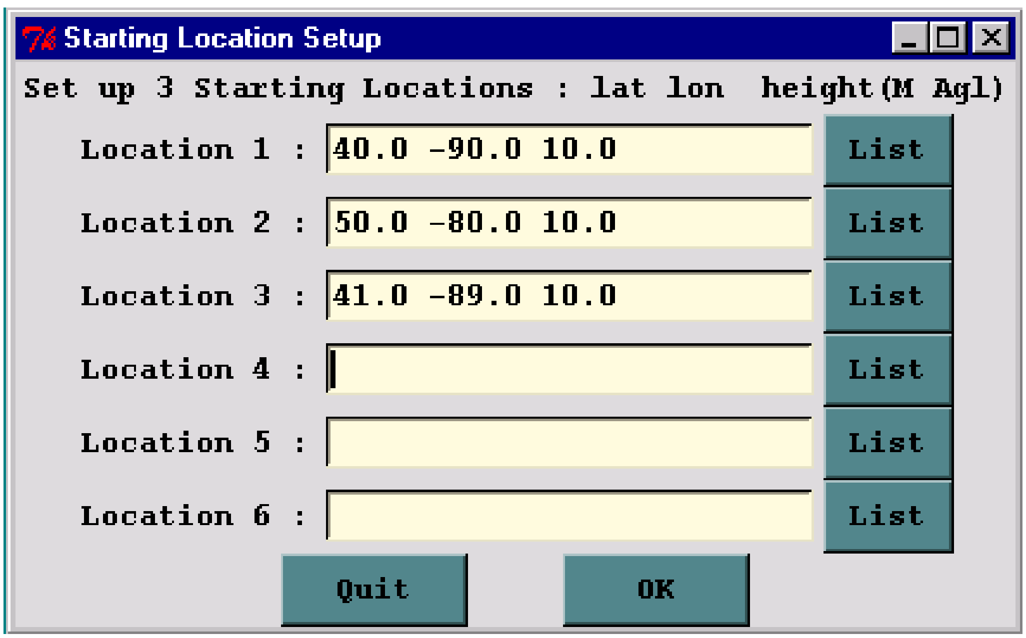
-y[The default is linear ordinate scaling. The flag sets y-axis log scaling]

CONCENTRATION - Special Simulations

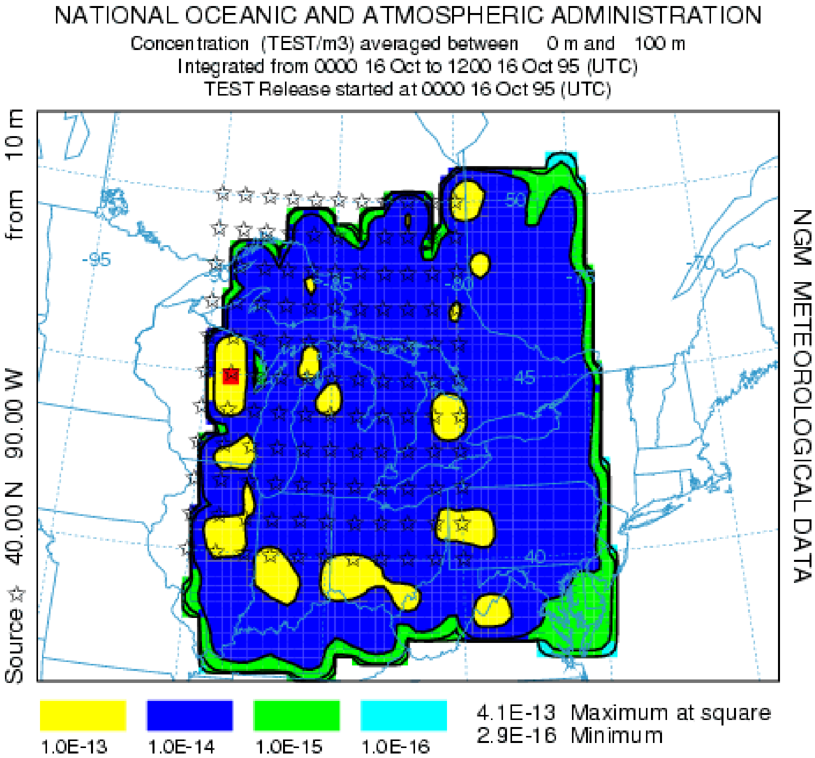
Special simulations may require a different executable file, modifications to the Control file that are not supported by the GUI, or interactions with other items under the Advanced Menu tab. More information is provided below for each special simulation. Special model configurations may not be available for all operating systems. Note that most of the special simulations may be run using a single processor system or multiprocessor system (supporting MPI).

Run Matrix

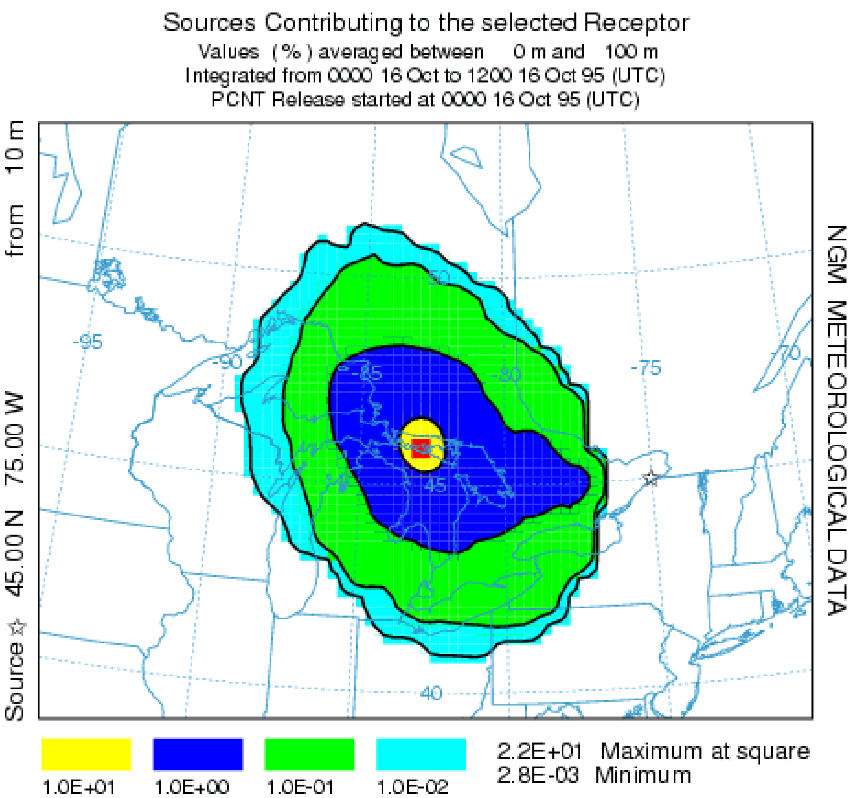
Although the setup of the concentration matrix calculation is similar to that of the trajectory matrix calculation, there is an additional option that can be set in the Advanced Menu configuration tab that changes the nature of the concentration output file to produce a source-receptor matrix. This will be discussed further below. The matrix calculation is a way to set up the *CONTROL* file for multiple starting locations that may exceed the GUI limit of 6 under the Concentration Setup menu tab. Hundreds or thousands of starting points may be specified. The *Run Matrix* menu tab first runs a program that reads the *CONTROL* file with three starting locations and then rewrites the same *CONTROL* file with multiple locations. The multiple locations are computed from the number of starting points that fall in the domain between starting point 1 and starting point 2, where each new location is offset is the same as that between starting locations 1 and 3. For instance, if the original control file has three starting locations: #1 at 40N, 90W; #2 at 50N, 80W, and #3 at 41N, 89W; then the matrix processing results in a Control file with 121 starting locations, all one degree apart, all between 40N 90W and 50N 80W.



In the normal model execution mode, the concentration contributions from multiple sources are summed on the concentration grid, hence it is not possible to determine the fraction of the material comes from each source location. This can be seen in the illustration below using the above configuration for the first 12 hours of the sample case.

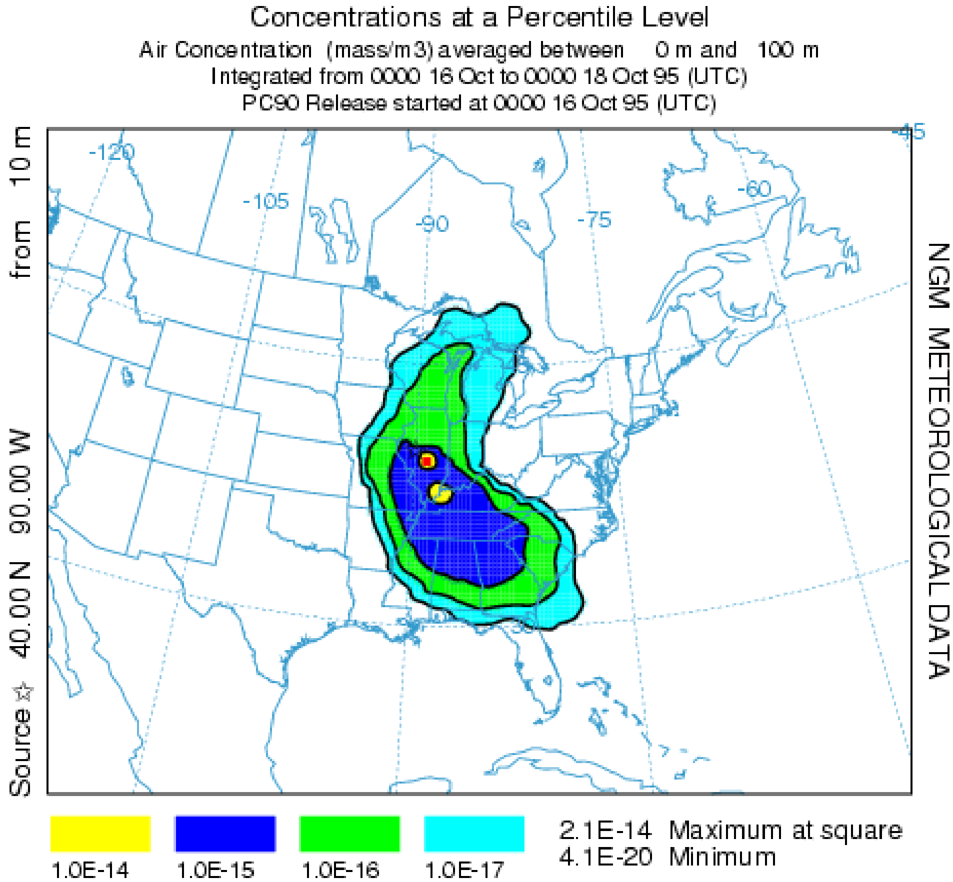


However, if the "Matrix" conversion module is checked in the [Advanced Concentration](#) Configuration menu tab, then the multiple source simulation maintains the identity of each source in the concentration output file. The [Display Matrix](#) menu tab permits extraction of information for individual sources or receptors from this special concentration output file. The results of the same simulation are shown in the illustration below. In this case the receptor check-box was set and the receptor location was identified as 45.0, -75.0 with normalization. Therefore the graphic illustrates the fractional contribution of each region to the first 12 hour average concentration at the designated receptor.



Run Ensemble

The ensemble form of the model, an independent executable, is similar to the trajectory version of the ensemble. The meteorological grid is offset in either X, Y, and Z for each member of the ensemble. The model automatically starts each member on a single processor in a multi-processor environment or cycles through the simulations on one processor. The calculation offset for each member of the ensemble is determined by the grid factor as defined in the Advanced Concentration Configuration Tab. The default offset is one meteorological grid point in the horizontal and 0.01 sigma units in the vertical. The result is 27 ensemble members for all offsets. The normal Setup Menu tab is used to configure the *CONTROL* file. Note that if fewer than 27 processors are available, the ensemble configuration menu permits starting the calculation at any ensemble member number within the valid range. Because the ensemble calculation offsets the starting point, it is suggested that for ground-level sources, the starting point height should be at least 0.01 sigma (about 250 m) above ground. The model simulation will result in 27 concentration output files named according the file name setting in the control file "{cdump}.{001 to 027}" with a suffix equivalent to the ensemble member number. On a single processor system, the calculation may take some. The menu will be locked until the simulation has completed. A message file window will open after termination. Computational progress may be monitored by noting the generation of new concentration output and message files with the ensemble number suffix in the /working directory. The concentration output from each member can be displayed through the concentration display menu tab. However, to display the probabilities associated with the multiple simulations, it is necessary to pre-process the data through the Display Ensemble menu tab. Using the default configurations for the sample simulation, the illustration below represents the 90th percentile concentrations aggregating all four output time periods.

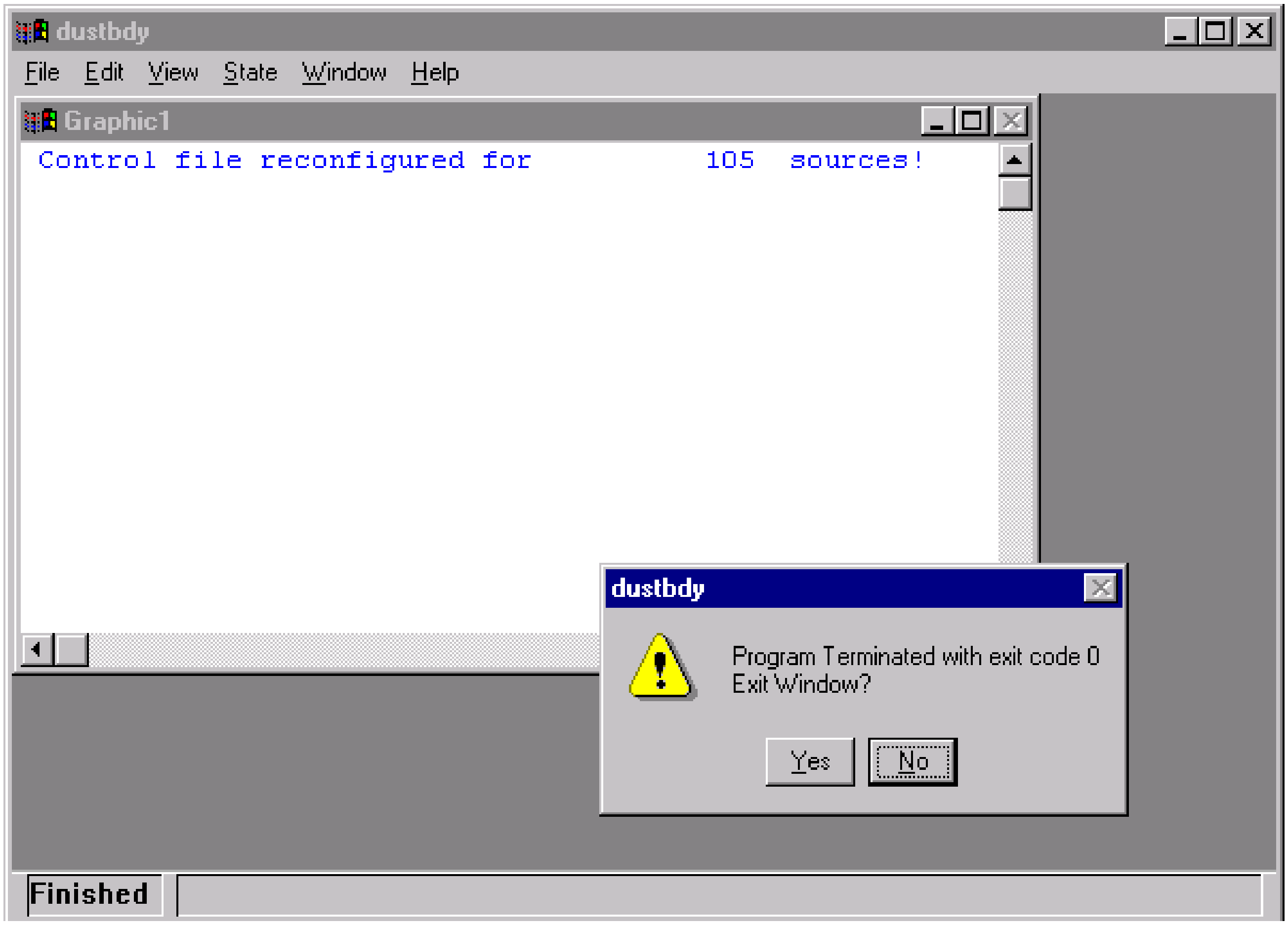


For instance the blue contour in this 90th percentile plot represents the region in which only 10% of the ensemble members have air concentrations greater than 10^{-15}.

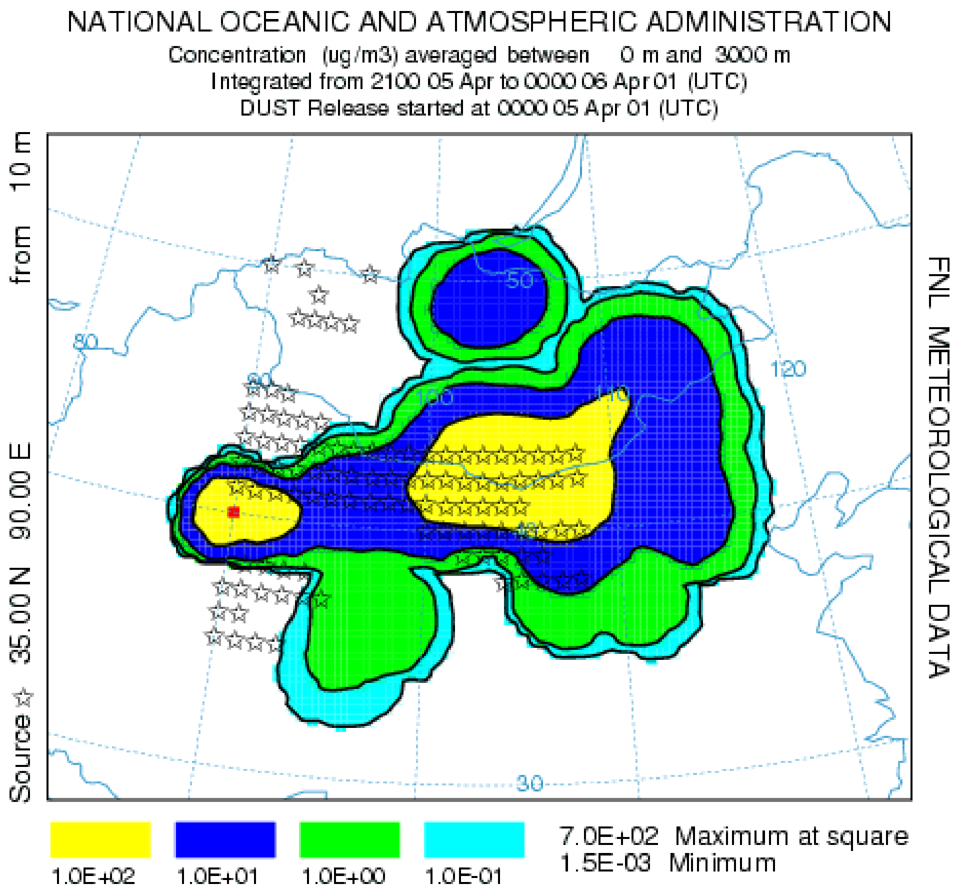
Run Dust Storm

A model for the emission of PM10 dust has been constructed (Draxler, R.R, Gillette, D.A., Kirkpatrick, J.S., Heller, J., 2001, Estimating PM10 Air Concentrations from Dust Storms in Iraq, Kuwait, and Saudi Arabia, Atmospheric Environment, Vol. 35, 4315-4330) using the concept of a threshold friction velocity which is dependent on surface roughness. Surface roughness was correlated with geomorphology or soil properties and a dust emission rate is computed where the local wind velocity exceeds the threshold velocity for the soil characteristics of that emission cell. A pre-processing program was developed that accesses the HYSPLIT land-use file over any selected domain and modify the input CONTROL file such that each emission point entry corresponds with a "desert" (active sand sheet) land-use grid cell. The original PM10 flux equation was replaced by a more generic relationship (Westphal, D.L., Toon, O.B., Carlson, T.N., 1987. A two-dimensional numerical investigation of the dynamics and microphysics of Saharan dust storms. J. Geophys. Res., 92, 3027-3029).

The dust storm simulation is configured in the same way as the matrix calculation in that it is necessary to define three source locations, the first two representing the limits of the domain, and the third defining the emission grid resolution. The pre-processor then finds all emission points within that domain that have a desert category and modify the *CONTROL* file accordingly. The dust box must be checked in the advanced configuration menu to compute the PM10 emission rate. As an example, we can configure the model to run the large Mongolian dust storm of April 2001. An animation of the calculation results can be downloaded. To run the same simulation it will be necessary to obtain the first two weeks of northern-hemisphere meteorological analysis data (FNL.NH.APR01.001). A pre-configured *CONTROL* file (*dust_conc*) should be retrieved from the working directory. The *CONTROL* file defines the emission domain by the three starting locations: 35N-90W to 50N-120W with the grid increment to 36N-91W. There is no point in defining an emission grid of less than one-degree resolution because the resolution of the land-use data file is one-degree and the meteorological data is closer to two-degrees. Once the model is setup for the simulation, including the dust check-box in the configuration menu, execute the model from the *Special Simulations / Run Dust Storm* menu tab. The window shown below will open to indicate the revision of the *CONTROL* file.



The message indicates that the initial 3-location *CONTROL* file was reconfigured by the *dustbdy* program for 105 source locations. That means in the domain specified, 105 one-degree latititude-longitude grid cells were found to have a desert land-use category. If none are found, then the *CONTROL* file is deleted to prevent model execution. Click on Yes or No to continue - yes just deletes the window. The model execution will then start. PM10 pollutant dust particles are only emitted from those 105 cells where the wind speed exceeds the emission threshold. Therefore it is possible to have simulations with no emissions. An example of the output after 24 hours simulation time is shown in the illustration below.



The concentrations represent a 3-hour average from 21 to 24 hours after the start of the simulations. It is not possible to say exactly from when or where particles are emitted except to note that the 105 potential source locations are shown. The emission algorithm emits in units of grams, but the in configuring the concentration display, the units were changed to ug by defining a conversion factor of 10^6. Maximum concentrations are on the-order-of 100, but the layer was defined with a depth of 3 km to facilitate comparison with satellite observations. The simulation could be rerun with a smaller layer to obtain concentrations more representative of near-ground-level exposures.

CONCENTRATION - Multiprocessor Simulations

Special simulations may require a different executable file, modifications to the Control file that are not supported by the GUI, or interactions with other items under the Advanced Menu tab. More information is provided below for each special simulation that can be run under a multi-processor environment that supports MPI. These special simulations may not be available for all operating systems. All the MPI simulations execute the special script *run_mpi.scr* which can be found in the *./concmdl* directory. This script executes the appropriate MPI executable variation of the concentration model and almost certainly will require some customization to match the local operating system environment.

Run MPI Model

The standard concentration model can be run on multi-processor systems. As pollutant particles are released during the simulation, they are parsed out in sequence to the available processors. The calculation proceeds independently until the end of a concentration averaging period. At this point, the concentrations from each processor are summed, and only one concentration output file is updated. The MPI version can be quite effective in speeding up simulations requiring the release of many particles. No special configuration or control file is required and output can be viewed using the standard concentration display menu.

Run Matrix Model

The multiprocessor version of the matrix calculation is configured the same way as for a single processor system. The MPI calculation proceeds as described above for the standard MPI model simulation.

Run Ensemble Model

The ensemble model automatically starts each member on a single processor in a multi-processor environment. The multiprocessor version of the ensemble calculation is configured the same way as for a single processor system. Note that if fewer than 27 processors are available, the ensemble configuration menu permits starting the calculation at any ensemble member number within the valid range of 001 to 027.

CONCENTRATION - File Format

Concentration packing has been implemented with HYSPLIT version 4.5. The updated format is downward compatible in that all display programs can read files produced from versions prior to 4.5, but older versions of the display programs cannot read the new packed output format. Note that HYSPLIT V4.5 can be configured to produced the older style unpacked concentration files. Concentration file packing does not write the same information in fewer bytes, but rather writes the same information using twice as many bytes. The packed files are generally smaller because only concentration values at the non-zero grid points are written to the output file by the model. However this requires the grid point location to be written with the concentration, hence the additional bytes. If most of the grid is expected to have non-zero concentrations, then the old style format will save space. The output format of the unformatted binary (big-endian) concentration file written by dispersion model (*hymodelc)* and read by all concentration display programs is as follows:

Record #1

CHAR*4 Meteorological *MODEL* Identification

INT*4 Meteorological file starting time

(*YEAR, MONTH, DAY, HOUR, FORECAST*)

INT*4 *NUMBER* of starting locations

INT*4 Concentration packing flag (0=no 1=yes)

Record #2 Loop to record: Number of starting locations

INT*4 Release starting time (*YEAR, MONTH, DAY, HOUR*)

REAL*4 Starting location and height (*LATITUDE, LONGITUDE, METERS*)

Record #3

INT*4 Number of (*LATITUDE-POINTS, LONGITUDE-POINTS*)

REAL*4 Grid spacing (*DELTA-LATITUDE, DELTA-LONGITUDE*)

REAL*4 Grid lower left corner (*LATITUDE, LONGITUDE*)

Record #4

INT*4 *NUMBER* of vertical levels in concentration grid

INT*4 *HEIGHT* of each level (meters above ground)

Record #5

INT*4 *NUMBER* of different pollutants in grid

CHAR*4 Identification *STRING* for each pollutant

Record #6 Loop to record: Number of output times

INT*4 Sample start (*YEAR MONTH DAY HOUR MINUTE FORECAST*)

Record #7 Loop to record: Number of output times

INT*4 Sample stop (*YEAR MONTH DAY HOUR MINUTE FORECAST*)

Record #8 Loop to record: Number of levels, Number of pollutant types

CHAR*4 Pollutant type identification *STRING*

INT*4 Output *LEVEL* (meters) of this record

No Packing (all elements) Packing (only non-zero elements)

REAL*4 - Concentration output *ARRAY* INT*4 - Loop non-zero elements

INT*2 - First (I) index value

INT*2 - Second (J) index value

REAL*4 - Concentration at (I,J)

ADVANCED - Applications Help

This section provides some guidance in configuring the model input to perform certain specialized calculations. These include deciding between particle or puff releases, dealing with continuous emissions, area source emissions, multiple pollutants, pollutant transformations, deposition and decay, compilation limits, scripting for automation, backward dispersion considerations, and how to configure the time or spatial variation of the emission rate. The Advanced menu is composed of four sections.

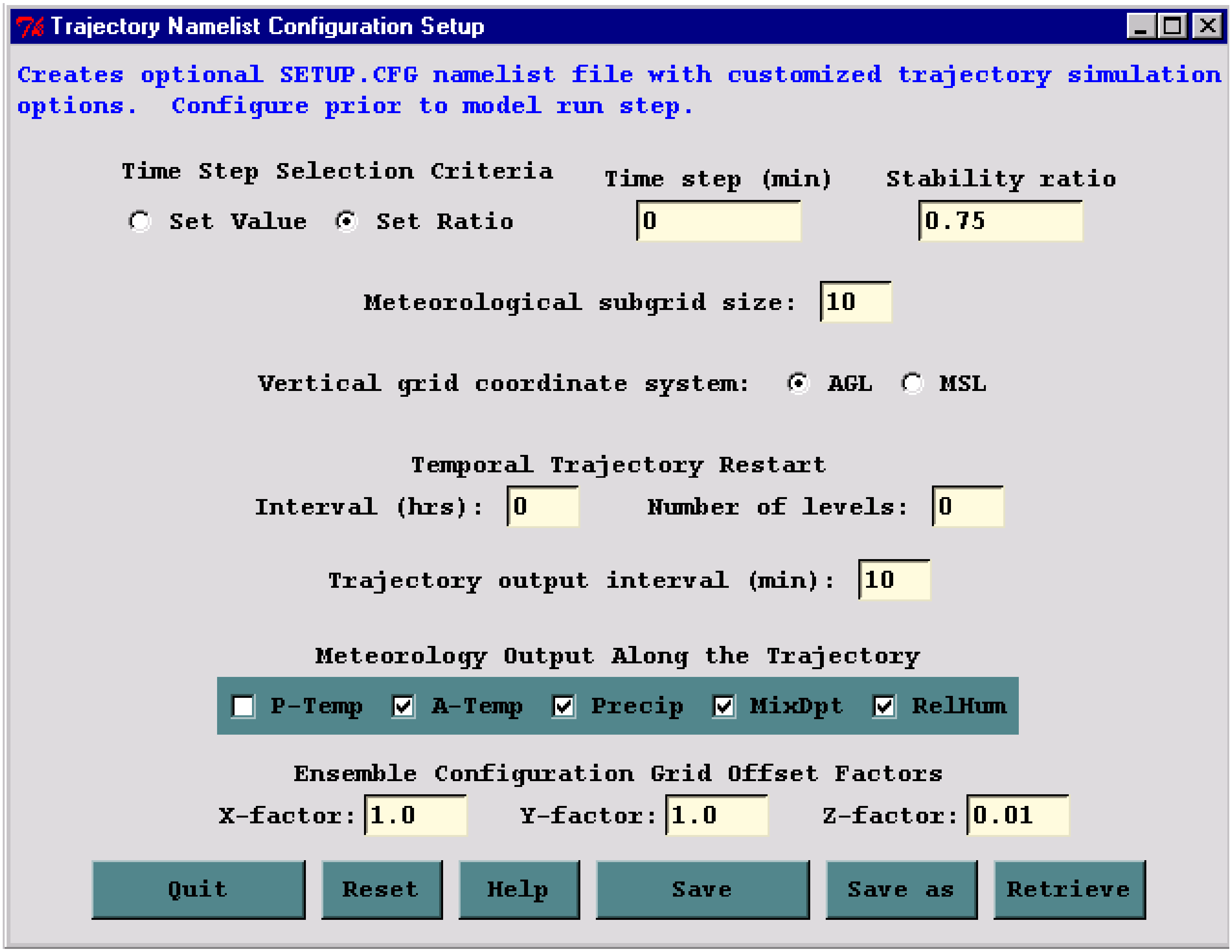
1. The Configuration Setup menu permits the creation and modification of the *SETUP.CFG* namelist file for either Trajectory or Air Concentration calculations. The namelist file is a variable length file that is used to set additional parameters that can be used to modify the nature of the simulation. The namelist file is not required because all the namelist variables take on default values when the *SETUP.CFG* file is not found. The same namelist file can be used for either trajectory or air concentration simulations but only certain variables are applicable to each simulation. Supplemental plot labeling options are also available.
2. The Particle Editor menu opens a special viewing program that will display the pollutant particle positions and satellite data. The editor can be used to adjust the particle positions to obtain a better correspondence with the satellite observations. The model calculation can then be restarted with the adjusted pollutant particle positions. The particle position file, called PARDUMP by default, is a snapshot of all the pollutant particles at a given time. Its creation is controlled through the Configuration menu.
3. The FTP satellite data menu opens up an anonymous FTP window to access the current satellite archives for TOMS aerosol index and today's AVHRR optical depth. These data can then be displayed through the Particle Editor menu.
4. The View *MESSAGE* menu is used to display the diagnostic message file. For all trajectory or concentration simulations, diagnostic information is written to standard output until the initialization process has completed. At that point the *MESSAGE* file is opened and subsequent diagnostic and certain error messages are written to this file. If the model does not complete properly, some information may be obtained from this file.

ADVANCED - Configuration Setup

This section provides some guidance in configuring the model input to perform certain specialized calculations. The default configuration supplied with the test meteorological data is confined to a simple trajectory and inert transport and dispersion calculation. More complex scenarios can be configured through the *Advanced* menu "*Configuration Setup*" for trajectories or concentrations. These situations require modification of the "*SETUP.CFG*" namelist file. This can be accomplished through the GUI or by directly editing the namelist file. The file is required to be in the model startup directory, usually *./working* when running the model through the GUI. The *trajectory* and *concentration* menus are confined to the variables relevant to the respective model calculation. More information is given in each menu's help section. An simple edit menu is provided to create a file of extra label information that is added to the bottom of trajectory or concentration plots.

ADVANCED - Configuration Setup - Trajectory

This section provides some guidance in configuring the model input to perform certain specialized calculations. The default configuration supplied with the test meteorological data is confined to a simple trajectory calculation. More complex scenarios can be configured through the *Advanced*, *Configuration Setup*, *Trajectory* menu tab. The menu is used to modify the *SETUP.CFG namelist* file. This file is not required, and if not present in the root startup directory, default values are used. These parameters can all be changed without recompilation by modification of the contents of *SETUP.CFG* and in some cases their modification will substantially change the nature of the simulation. The configuration file should be present in the root directory. An illustration of the menu is shown below.



When using the GUI, the namelist file will be deleted and all variables are returned to their default value by selecting the "*Reset*" button. The following summarizes the namelist variables and their corresponding section in the GUI menu. Not all variables can be set through the menu.

Time Step Selection Criteria

TRATIO (0.75) - defines the fraction of a grid cell that a particle or trajectory is permitted to transit in one advection time step. Reducing this value will reduce the time step and increase computational times. Smaller time steps result in less integration error. Integration errors can be estimated by computing a backward trajectory from the forward trajectory end position and computing the ratio of the distance between that endpoint and the original starting point divided by the total forward and backward trajectory distance.

DELT (0.0) - is used to set the integration time step to a fixed value in minutes from 1 to 60. It should be evenly divisible into 60. The default value of zero causes the program to compute the time step each hour according to the maximum wind speed, meteorological and concentration grid spacing, and the *TRATIO* parameter. The fixed time step option should only be used when strong winds in regions not relevant to the dispersion simulation (perhaps the upper troposphere) are causing the model to run with small time steps. Improper specification of the time step could cause aliasing errors in advection and underestimation of air concentrations.

Meteorological Sub-grid Size

MGMIN (10) - is the minimum size in grid units of the meteorological sub-grid. The sub-grid is set dynamically during the calculation and depends upon the horizontal distribution of end-points and the wind speed. Larger sub-grids than necessary will slow down the calculation by forcing the processing of meteorological data in regions where no transport or dispersion calculations are being performed. In some situations, such as when the computation is between meteorological data files that have no temporal overlap, the model may try to reload meteorological data with a new sub-grid. This will result in a fatal error. One solution to this error would be to increase the minimum grid size larger than the meteorological grid to force a full-grid data load.

Vertical Grid Coordinate System

KMSL (0) - sets the default for input heights to be relative to the terrain height of the meteorological model. Hence input heights are specified as AGL. Setting this parameter to "1" forces the model to subtract the local terrain height from source input heights before further processing. Hence input heights should be specified as relative to Mean Sea Level (MSL). In concentration simulations, the MSL option also forces the vertical concentration grid heights to be considered relative to mean sea level.

Temporal Trajectory Restart

NSTR (0) - Hours between trajectory starts for multiple trajectory-in-time simulations. When greater than zero, new trajectories will be started from the original starting location every NSTR hours from the initial trajectory starting time. See the special trajectory simulation section for more information.

NVER (0) - Number of vertical levels that trajectories are restarted when trajectories for multiple trajectory-in-time-and-space simulations. When greater than zero, new trajectories will be started at this number of levels from the endpoint position at the NSTR interval. The level heights are set in the CONTROL file and must match the number of starting locations. See the special trajectory simulation section for more information.

Trajectory Output Interval

TOUT(60) - sets the time interval in minutes at which trajectory end-point positions will be written to the output file. Output intervals of less than 60 minutes can be selected. This will force the internal time step to be an even multiple of the output interval.

Meteorology Output Along the Trajectory

Sets the option to write the value of certain meteorological variables along the trajectory to the trajectory output file. The marker variables are set to (1) to turn on the option. Multiple variables may be selected for simultaneous output but only one variable may be plotted. If multiple variables are selected in conjunction with the trajectory display option, then only the last variable output will be shown in the graphic. The variable output order is fixed in the program and cannot be changed.

TM_PRES (0) - diagnostic marker variables to output atmospheric pressure (1) along the trajectory

TM_TPOT (0) - potential temperature (1) in degrees Kelvin;

TM_TAMB (0) - ambient temperature (1) in degrees Kelvin;

TM_RAIN (0) - rainfall (1) in mm per hour;

TM_MIXD (0) - mixed layer depth (1) in meters.

TM_RELH (0) - relative humidity (1) in percent.

Ensemble Grid Configuration Offset Factors

Sets the dimensions at which the meteorological grid will be offset for the ensemble calculation. Only one offset (+ or -) in either X, Y, or Z is applied per member.

DXF (1.0) - west to east grid factor for offsetting the meteorological grid in the ensemble calculation.

DYF (1.0) - south to north grid factor for offsetting the meteorological grid in the ensemble calculation

DZF (0.01) - vertical grid factor for ensemble (0.01 ~ 250m)

Variables Not Available through the GUI

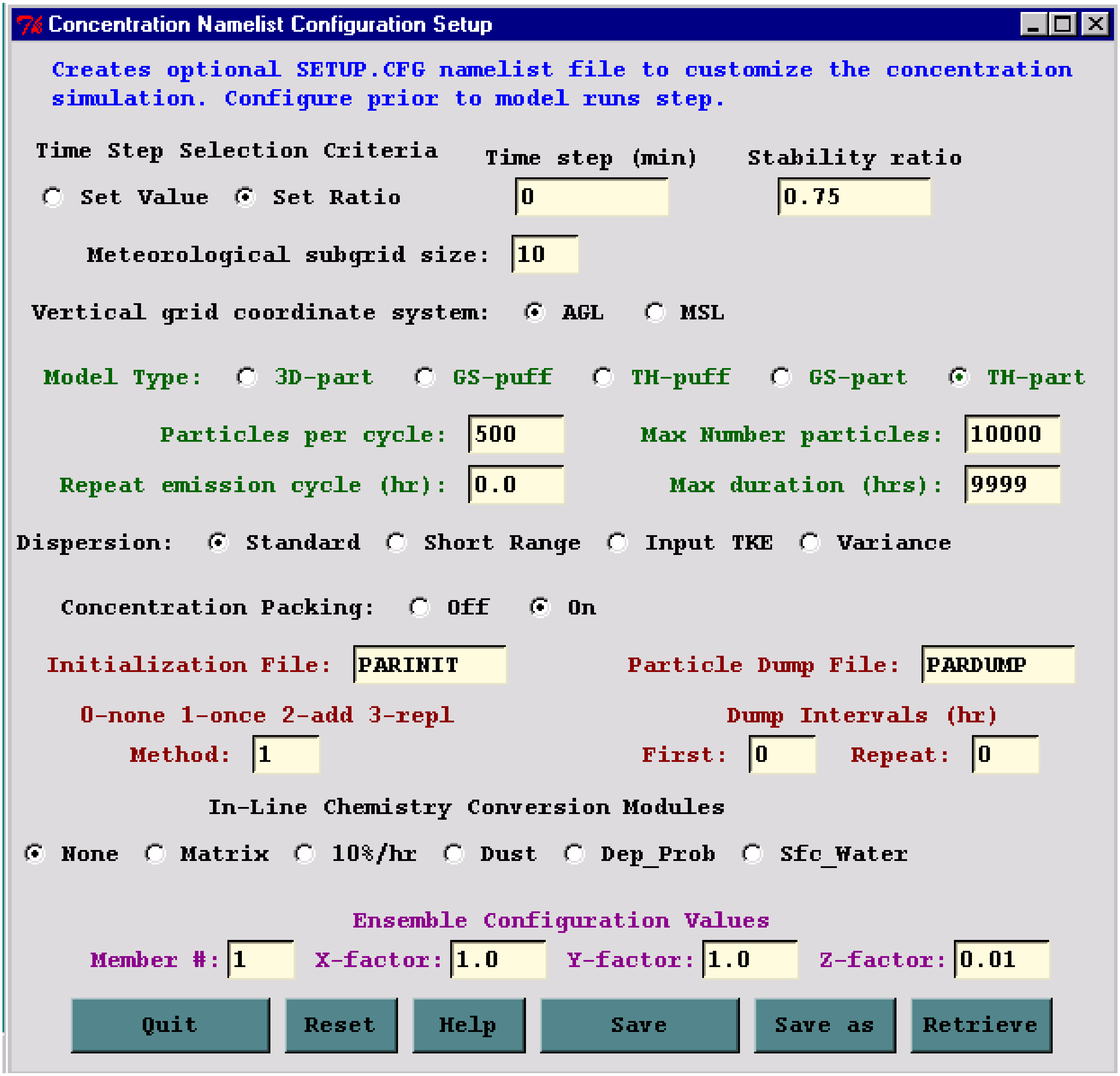
AA (30), BB (-25), CC (5) - are the polynomial parameters that control the resolution of the internal HYSPLIT terrain following grid system. Input meteorological data are interpolated to this grid. The polynomial relates height (AGL) to the internal vertical index number where k=1 would be the first level above ground: Z = *AA**k**2 + *BB**k + *CC*.

KSFC (2) - defines the vertical index of the internal meteorological data level that is considered to be the top of the surface layer. This value is used for stability scaling computations. The default index value of 2 is consistent with the default polynomial parameters that results in the height of the surface layer top to default to 75 m AGL.

MAXLVL(35) - are the maximum number of meteorological data levels that can be input to the model. This value exceeds all currently available meteorological data sets.

ADVANCED - Configuration Setup - Concentration

This section provides some guidance in configuring the model input to perform certain specialized calculations. The default configuration supplied with the test meteorological data is confined to a simple dispersion calculation. More complex scenarios can be configured through the *Advanced*, *Configuration Setup*, *Concentration* menu tab. The menu is used to modify the *SETUP.CFG namelist* file. This file is not required, and if not present in the root startup directory, default values are used. These parameters can all be changed without recompilation by modification of the contents of *SETUP.CFG* and in some cases their modification will substantially change the nature of the simulation. The configuration file should be present in the root directory. An illustration of the menu is shown below.



When using the GUI, the namelist file will be deleted and all variables are returned to their default value by selecting the "*Reset*" button. The following summarizes the namelist variables and their corresponding section in the GUI menu. Not all variables can be set through the menu.

Time Step Selection Criteria

TRATIO (0.75) - defines the fraction of a grid cell that a particle or trajectory is permitted to transit in one advection time step. Reducing this value will reduce the time step and increase computational times. Smaller time steps result in less integration error. Integration errors can be estimated by computing a backward trajectory from the forward trajectory end position and computing the ratio of the distance between that endpoint and the original starting point divided by the total forward and backward trajectory distance.

DELT (0.0) - is used to set the integration time step to a fixed value in minutes from 1 to 60. It should be evenly divisible into 60. The default value of zero causes the program to compute the time step each hour according to the maximum wind speed, meteorological and concentration grid spacing, and the *TRATIO* parameter. The fixed time step option should only be used when strong winds in regions not relevant to the dispersion simulation (perhaps the upper troposphere) are causing the model to run with small time steps. Improper specification of the time step could cause aliasing errors in advection and underestimation of air concentrations.

Meteorological Sub-grid Size

MGMIN (10) - is the minimum size in grid units of the meteorological sub-grid. The sub-grid is set dynamically during the calculation and depends upon the horizontal distribution of end-points and the wind speed. Larger sub-grids than necessary will slow down the calculation by forcing the processing of meteorological data in regions where no transport or dispersion calculations are being performed. In some situations, such as when the computation is between meteorological data files that have no temporal overlap, the model may try to reload meteorological data with a new sub-grid. This will result in a fatal error. One solution to this error would be to increase the minimum grid size larger than the meteorological grid to force a full-grid data load.

Vertical Grid Coordinate System

KMSL (0) - sets the default for input heights to be relative to the terrain height of the meteorological model. Hence input heights are specified as AGL. Setting this parameter to "1" forces the model to subtract the local terrain height from source input heights before further processing. Hence input heights should be specified as relative to Mean Sea Level (MSL). In concentration simulations, the MSL option also forces the vertical concentration grid heights to be considered relative to mean sea level.

Model Type

INITD (4) - determines if the model is configured as a puff or particle model. Valid options are: 0 (Horizontal and Vertical Particle); 1 (Horizontal Gaussian Puff, Vertical Top Hat Puff); 2 (Horizontal and Vertical Top Hat Puff); 3 (Horizontal Gaussian Puff, Vertical Particle); and the default, 4 (Horizontal Top-Hat Puff, Vertical Particle).

Particle Number

NUMPAR (500) - would be the maximum number of particles or puffs released. *NUMPAR* has a different meaning for puff and particle simulations. In a full puff simulation (*INITD*=1,2), only one puff per time step is released, regardless of the value of *NUMPAR*. In a particle or mixed particle-puff simulation (*INITD*=0,3,4), *NUMPAR* represents the total number of particles that are released during one release cycle. Multiple release cycles cannot produce more than *MAXPAR* number of particles. For a mixed simulation (particle-puff), *NUMPAR* should be greater than one but does not need to be anything close to what is required for a full 3D particle simulation. In all simulation types, particle or puffs are emitted if the particle count is less than *MAXPAR*.

MAXPAR (10000) - is the maximum number of particles permitted to be carried at any time during a simulation.

Emission Cycle and Duration

QCYCLE (0.0) - are the number of hours between emission start cycles. A zero value means that the emissions are not cycled. When non-zero, the number of emission hours is repeated again at *QCYCLE* hours after the starting emission time specified in the input Control file.

HMAX (9999) - is the maximum age (in hours) that any puff or particle is permitted to attain. All pollutants beyond this age are deleted.

Internal Dispersion Parameterization

ISOT (0) - is a flag used to set the turbulence computational method. The default standard method computes the mixing using a diffusivity approach based upon vertical stability estimates and the horizontal wind field deformation. In shorter-range dispersion simulations (<100 km) the deformation parameterization used in conjunction with larger scale meteorological fields will not reflect the diurnal variations in horizontal turbulence. In this situation it is desirable to use the short-range parameterization in which the turbulent velocities are computed directly from the stability parameters. The input tke (turbulent kinetic energy) option replaces the model's computation of stability with the TKE field from the input meteorological data set. Currently only the ETA forecast model data contain the TKE field. In the variance option, the input data is assumed to contain the 3-dimensional component velocity variance fields.

0 - Standard: Vertical diffusivity and horizontal deformation

1 - Short Range: Horizontal and vertical velocity variances

2 - Not defined. 3 - Same as ISOT=0 except vertical mixing not averaged

4 - Input TKE: tubulence based upon input TKE field

5 - Input TKE: Same as ISOT=4 except Zi based upon TKE profile

6 - Variances: Velocity variances from input meteorological data file

The options available through the menu are highlighted in red. Selection of an option inconsistent with the input meteorological data will result in the calculation using the default parameters.

Concentration Packing

CPACK (1) - is the flag to turn off (set to 0) concentration output packing. The default is to write the binary concentration file at only those grid points that have a non-zero concentration value (set to 1). Setting the flag to zero results in output of the entire concentration grid. Due to the nature of the packing method, if the plume covers more than 50% of the concentration grid, the default concentration packing will result in larger output file than an unpacked concentration file.

Particle Dump Intervals

PINPF (PARINIT) - default name for the particle dump input file.

NINIT - sets the type of initialization performed. When set to "0" no particle initialization occurs even if the "PINPF" file is found in the root directory. A value of "1" reads the file only during the initialization process. No initialization will occur if the time of the particle dump does not match the time of the model initialization. A value of "2" will check the file each hour during the simulation, and if the times match, the particles will be added to those already contained in the simulation. A value of "3" is similar to the previous case, except the particles in the file replace all the particles in the simulation.

POUTF (PARDUMP) - default name for the particle dump output file.

NDUMP (0) - can be set to dump out all the particle/puff points at selected time intervals to a file called *PARDUMP*. This file can be read from the root directory at the start of a new simulation to continue the previous calculation. Valid *NDUMP* settings are [0] for no I/O or [hours] to set the number of hours from the start of the simulation at which all the endpoint positions will be written to the file. The file must exist in the root directory and *NDUMP*>0 for the model to initialize pollutant particles from the file. *NDUMP* is used in conjunction with *NCYCL* (see below).

NCYCL (0)- sets the cycle interval at which the *PARDUMP* file is to be written after the first write at hours *NDUMP*. Multiple outputs will overwrite the last output. For instance in a multi-day simulation, one application would be to set *NDUMP*=24 and *NCYCL*=24 to output all points at the end of every simulation day. If the model were to crash unexpectedly, the simulation could be restarted from the last *PARDUMP* output.

Conversion Modules Changes the model's internal configuration in how it treats the pollutants. Some conversion options require additional modules and specific requirements in setting up the *CONTROL* file. See the linked discussion under each option for more information.

ICHEM (0) - chemistry module selection index:

0=none

1=matrix

2=10% / hour

3=PM10 dust storm simulation

4=Set concentration grid identical to the meteorology grid (not in GUI)

5=Deposition Probability method

6=Puff to Particle conversion (not in GUI)

7=Surface water pollutant transport

Ensemble Configuration Values

Sets the dimensions at which the meteorological grid will be offset for the ensemble calculation. Only one offset (+ or -) in either X, Y, or Z is applied per member.

DXF (1.0) - west to east grid factor for offsetting the meteorological grid in the ensemble calculation.

DYF (1.0) - south to north grid factor for offsetting the meteorological grid in the ensemble calculation

DZF (0.01) - vertical grid factor for ensemble (0.01 ~ 250m)

Variables Not Set in GUI

KSFC (2) - defines the vertical index of the internal meteorological data level that is considered to be the top of the surface layer. This value is used for stability scaling computations. The default index value of 2 is consistent with the default polynomial parameters that results in the height of the surface layer top to default to 75 m AGL.

MAXDIM (1) - is the maximum number of pollutants that can be attached to one particle. Otherwise, if multiple pollutants are defined they are released as individual particles. This feature is only required with chemical conversion modules and is not implemented in the public distribution version.

MAXLVL (35) - are the maximum number of meteorological data levels that can be input to the model. This value exceeds all currently available meteorological data sets.

KRND (6) - at this interval in hours, enhanced puff merging takes place. Enhanced merging is less restrictive and will degrade the accuracy of the simulation. Puffs can be further apart and still be merged into the same position. Less frequent merging will improve accuracy, however too many puffs may remain and the simulation time will be substantially increased. The selection of an appropriate value depends if the pollutant release is instantaneous or continuous. Enhanced merging only occurs when the puff number exceeds 25% of *MAXPAR*.

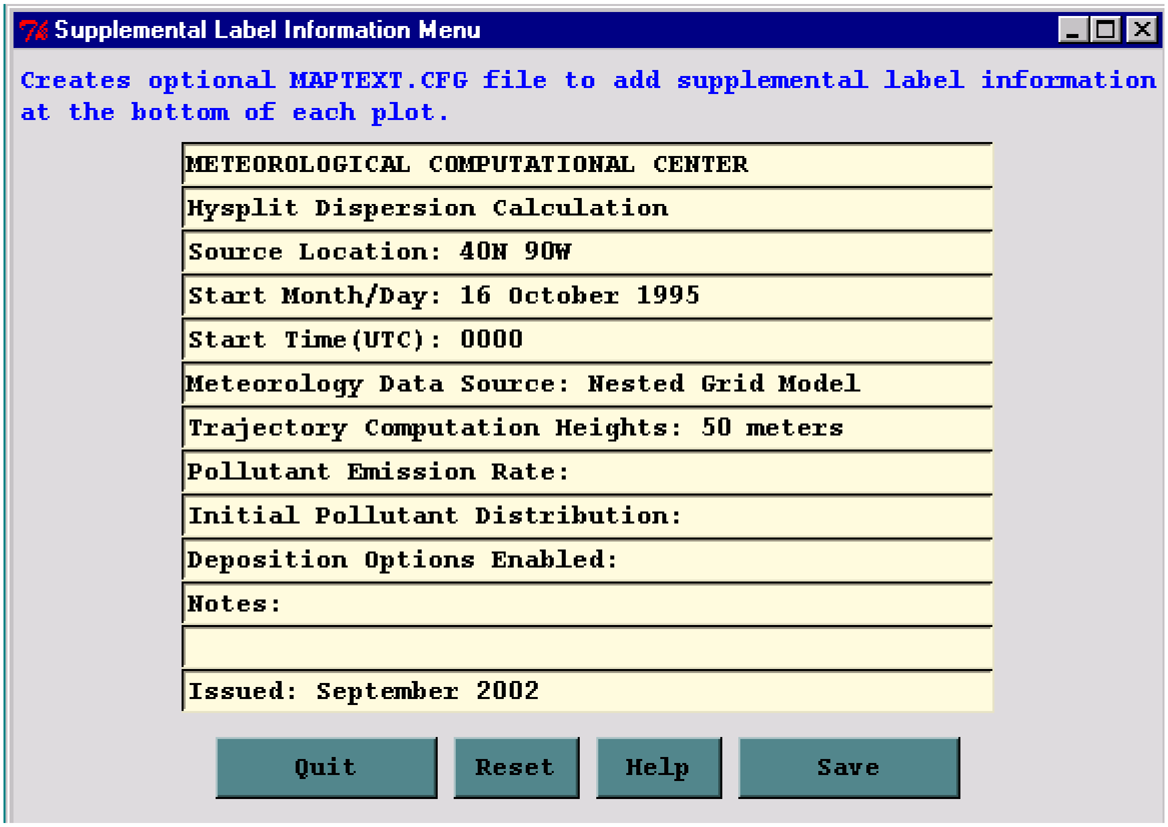
FRME (0.10) - is the fraction of the total mass that represents a puff mass at which all puffs with a mass less that puff value will only account for FRME of the total mass. These "Low Mass" puffs will be subject to enhanced merging.

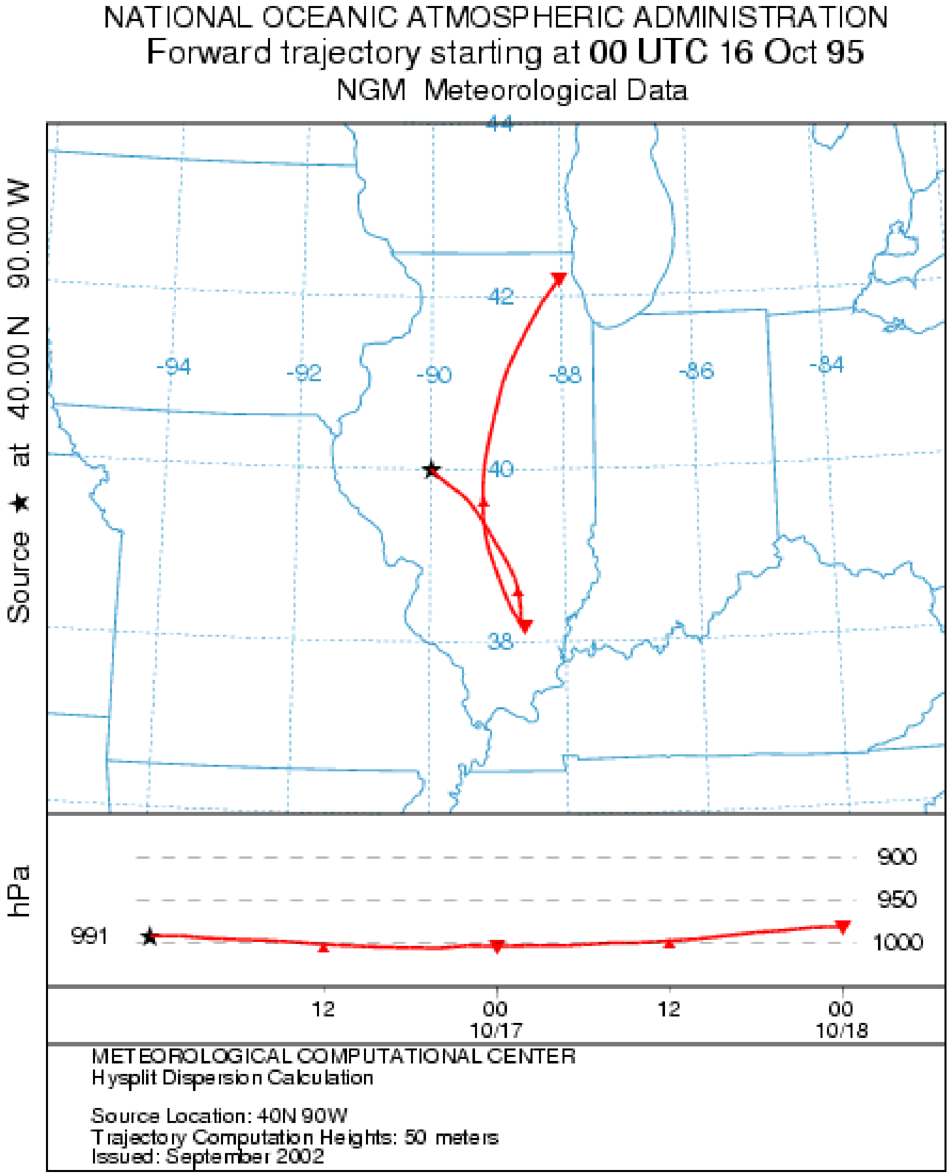
FRMR (0.0) - is the fraction of the mass that is permitted to be removed at KRND intervals. The normal situation is to permit no mass loss. However for certain simulations, such as when a pollutant has a high ambient background relative, a small removal rate will significantly reduce the number of puffs on the grid at no loss in accuracy.

EFILE (null) - file name that contains point-source temporal emission factors, where each record contains the {year month day hour duration latitude longitude emission-rate} of each emission period (including the first one defined in the *CONTROL* file, in the following format: (I4,2I3,2I5,F6.2,F8.2,F8.0). Note the hour and duration fields should be four characters long (0000), representing hours | minutes.

ADVANCED - Configuration Setup - Extra Label

If a file called *MAPTEXT.CFG* is found in the root or working directory during the execution of either the trajectory or concentration plotting program, additional label information is written at the bottom of each graphic. The *Extra Label* menu tab can be used to edit this information. An illustration of the menu is shown below, followed by the resulting trajectory graphic for the example simulation. Note that the menu entries are entirely text based and there are no restrictions regarding content. However, not all plotting programs display all lines. The example header text indicates the appropriate application.




NATIONAL OCEANIC ATMOSPHERIC ADMINISTRATION
Forward trajectory starting at 00 UTC 16 Oct 95
NGM Meteorological Data
Source ★ at 40.00 N 90.00 W
44
42
40
38
-94
-92
-90
-88
-86
-84
hPa
991
900
950
1000
12
00
10/17
12
00
10/18
METEOROLOGICAL COMPUTATIONAL CENTER
Hysplit Dispersion Calculation
Source Location: 40N 90W
Trajectory Computation Heights: 50 meters
Issued: September 2002

ADVANCED - Particle Edit / View

The *FTP Satellite Data* tab from the *Advanced* menu provides access to selected NASA and NOAA satellite archives. The *Particle Editor / Viewer* also displays the corresponding satellite image. An example of how the *Advanced* menu features can be used to view the TOMS aerosol index satellite image, and then adjust the model plume position before continuing on with the model calculation is explained in the following discussion. This is a pre-configured example only for demonstration purposes.

1) From the Concentration Setup menu, retrieve the standard example configuration *sample_conc*. Then modify the run time from 48 hours to 16 hours. The data file starts at 16 Oct 1995 on 0000 UTC. The first part of the simulation will end at 1600 UTC. This is close to local solar noon and approximately corresponds with the time of the TOMS image at the longitude of the US East Coast.

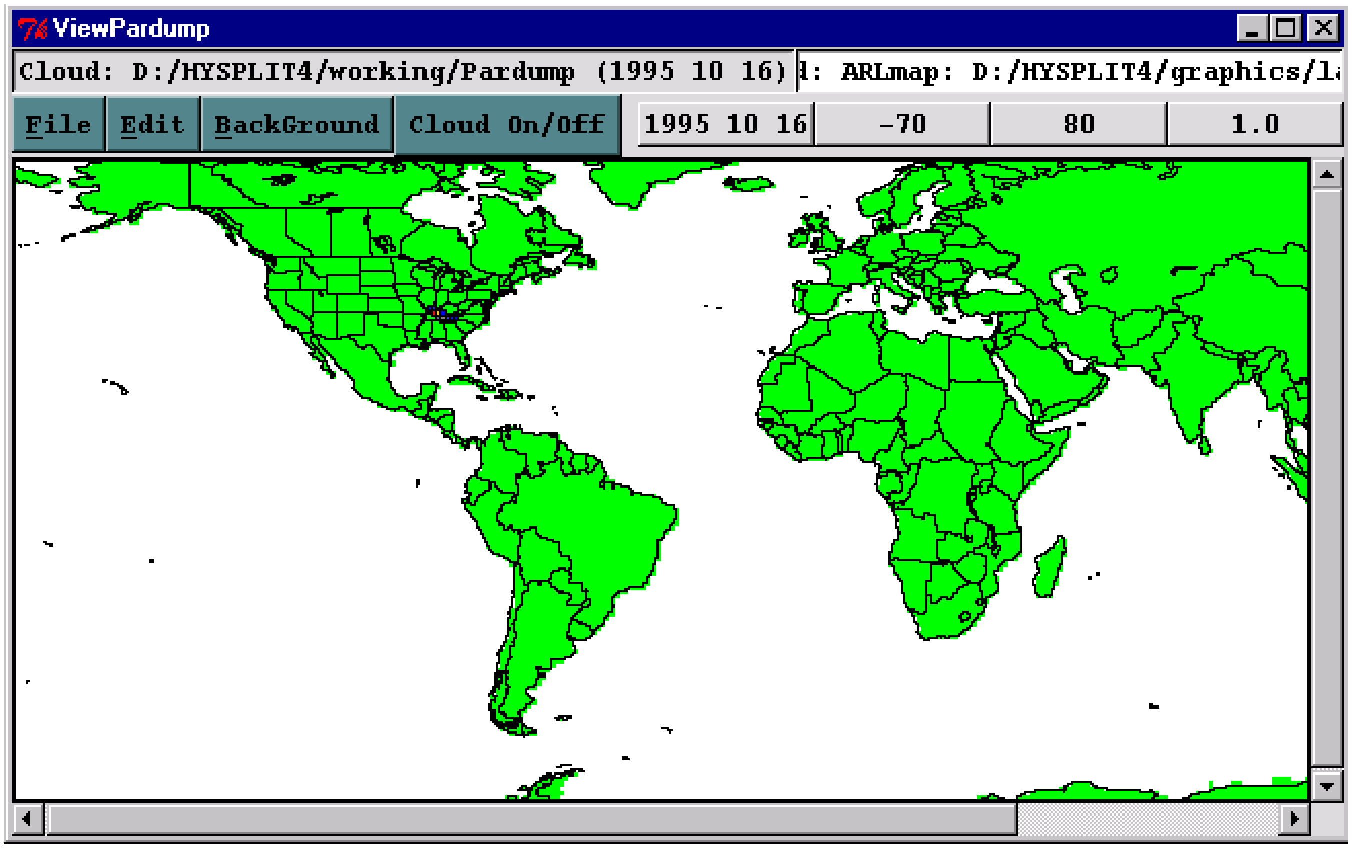
2) Go to the Configuration Setup of the *Advanced* menu and press the bottom *Rese*t button to bring all values back to their default. Then change the value *of* 1st Particle Dump from 0 to 16 hours. This configures the model to write the particle position file *PARDUMP* after 16 hours runtime. *Save* to exit the menu.

3) Now press Run Standard Model from the *Concentration* menu.

4) At the end of the simulation, go back to the *Advanced* menu and from *View MESSAGES*, review the *MESSAGE* file. Note that at the bottom of the listing the note that the *PARDUMP* file was created for 498 particles at 95 10 16 16, then *Exit*.

5) TOMS data for the 16th of October 1995 are not available, however a dummy file (for another date and renamed to reflect the simulation time) has been placed in the *./working* directory. Therefore no FTP is required. For other simulation times, use the NASA TOMS tab of the *FTP Satellite data* menu to ftp the daily file from the proper date. More information about TOMS data is available.

6) Open the *Particle Editor* from the *Edit/View* tab to display the particle positions given in the *PARDUMP* file found in the *./working* directory. In addition, the program can overlay the TOMS image for the sample data. Various images can be displayed and the particle positions manipulated as shown in the illustration below. More detailed instructions about this process are available.



7) Once the *PARDUMP* file has been adjusted to improve the comparison with the satellite data, go to the Configuration Setup menu and write particle initialization time from 16 hours to 24 hours and then *Save*. The model will be run another 24 hours and a new *PARDUMP* file will be written. Satellite passes are about every 24 hours over the same longitude.

8) From the Concentration Setup menu change the run time from 16 to 24 hours, and explicitly set the model starting time from all zeros to "95 10 16 16". Emissions need to be turned off, unless the source is continuous. In this example turn off emissions from the Pollutant selection of the Pollutant, Deposition, Grids Setup menu bar and set the emission duration from one hour to 0.0 hours. Then run the model.

9) At the completion of the simulation, review the *MESSAGE* to confirm that the model initialized with 498 particles (total mass still 1.0) and that a new *PARDUMP* file was created for the 17^{th}. The point of this exercise is to demonstrate how the model fields can be adjusted to correspond with measured data prior to running a simulation with forecast data, times at which no measurement data are available.

ADVANCED - Particle Editor - View / Edit

The Particle button initiates a display of the particles in the PARDUMP files on a geographic background. You may shift the positions of one or more groups of particles and save the results in the same or a new file. The particles may be viewed against the background of observed concentrations from TOMS Files or AVHRR Files.

File Status Bar

The Status bar is split into halves. The left half identifies the *PARDUMP* file currently being displayed and/or edited, while the right half identifies the background file. When a *PARDUMP* file is being displayed/edited, the file name is displayed, along with the valid date of the forecast particle positions. The background file may be the default *ARLMAP* geographic background, or background observed data from either of two sources, the TOMS file or the AVHRR data. If either of the observation data file types is being used, the file name is displayed as well as the date for which the observation is valid.

Menu and Status Bar

On the right side of the Menu and Status Bar are the current status labels of the mouse pointer. The center two labels give the instantaneous latitude and longitude, in degrees, of the mouse pointer. The right label gives the value of the background observation at that point. The meaning of the "observation value" depends on the type of background displayed. The left-hand label gives the valid date of the current Particle dump file, if open. The left side of the Menu and Status Bar are menu buttons, with the following effects:

File Menu

Open PARDUMP / Save As / Quit

This menu enables the user to specify the *PARDUMP* file to be loaded and displayed on the current background, and the file in which the edited results will be saved.

Edit

Zoom / Unzoom

The zoom function is provided by windowing. Click on the zoom button, move the cursor to one corner of the area of interest, then drag to the opposite corner. A rubber-band box displays the area to be magnified. A series of Zoom operations may be stacked; the unzoom button reverses them in sequence.

Particle Shift / Select Area

The particle shift sub-menu provides the editing of particle positions in an area-by-area fashion. When a *PARDUMP* File is loaded and displayed, an area may be selected for editing, first by clicking on the *select area* menu button, then moving the mouse to one corner of the area and dragging the mouse to the opposite corner. If the desired area is not selected, the *Undo Area* button will allow another selection.

New Pos / Undo New Pos

The *New Pos* button allows specification of how far the particles in the selected area will be shifted. After clicking on the menu button, place the mouse anywhere on the geographic display and drag the mouse. The arrow indicates the amount and direction by which the particles in the selected area will be shifted. Particles outside the area will be shifted by an amount that tapers rapidly with distance from the selected area. Repeated *New Pos* action will refine the desired shift.

Complete Area Shift / Undo Area

When the desired particle shifts have been made, *Complete Area Shift* should be selected. As many separate areas as desired may be acted on in this way. When all such areas have been completed, the edited positions can be saved to a file.

Background

ArlMap / TomsFile / AvhrrMap

Permits the selection of background information. The ArlMap display is always available, but the background displays for TOMS data and AVHRR data must be downloaded separately.

Cloud On/Off button

This button will cause the particle cloud display to appear and disappear as needed to judge how particle shift is needed against the background.

Geographic display

The display holds one of three types of background. The base display (ArlMap) is a geographic background of the world. The status bar observation value simply reads "0" when the mouse is over water area and "1" when the mouse is traversing land area.

The TOMS File display provides a color coded representation of analysis of aerosols from the "Total Ozone Mapping Spectrometer". A brown color represents missing data, white color represents negative values (corresponding to reflective aerosols) and a range of warmer colors represents various positive values (corresponding to absorbing aerosols. The status bar observation value reads the value itself, if available, or "m" for missing if not available.

The AVHRR Map display provides a representation of analyzed data from the Advanced Very High Resolution Radiometer. Since the analysis is valid only over water, the land areas are represented in green and the status bar observation value is "m" for missing there. Otherwise, it gives a reading of the aerosol depth. Over water, the display represents positive values in various shades of gray, with the darkest being the highest. A red color represents negative values. The TOMS File and AVHRR Map displays represent data valid at specific times. The valid time is presented on the right side of the *File status* bar.

ADVANCED - FTP Satellite - Total Ozone Mapping Spectrometer

Data from source -

"http://toms.gsfc.nasa.gov/eptoms/ep.html"

Data may be obtained from -

ftp://toms.gsfc.nasa.gov/pub/eptoms/data/ai2001/ga010901.epa

Following information saved from -

url=ftp://toms.gsfc.nasa.gov/pub/eptoms/data/1README

TOMS Earthprobe datasets are located in the following directory paths:

data/ozYYYY/gaYYMMDD.ept (YYYY=1996 >>>>>) (EarthProbe still reporting data)

Daily gridded (g), ASCII (a) files from year YY, month MM and day DD. The data in these ASCII files are in the format used for the Nimbus-7 and Meteor-3 CD-ROM. Details of this format are explained in the EarthProbe Data Products User's Guide, EARTHPROBE_USERGUIDE.PDF, available in the parent directory. A stub program, RDGRID.FOR may be used to read these datasets.

The values are in 3 digit groups 111222333444555
Sample ozone value: 234 = 234 du 0 = fill value

---

data/reflYYYY/gaYYMMDD.epr

Daily reflectivity data. Format similar to that of ozone data.

The vales are in 3 digit groups 111222333444555
Sample reflectivity value: _83 = 83% 999=fill value

---

data/a1YYYY/gaYYMMDD.epa

Daily aerosol data. Format similar to that of ozone data.

The data are an aerosol index formed directly from the measured radiances in two TOMS channels. Positive values generally represent absorbing aerosols (dust and smoke) while negative values represent nonabsorbing aerosols. The identification is not perfect because of geophysical reasons (e.g., aerosol too low to the ground).

There will soon be a corresponding data set of optical depths and single scattering albedos.

The values are in 3 digit groups 111222333444555
The numbers have been multiplied by 10

--1=0.1 -11=1.1 111=11.1

999 = missing or bad data
-- stands for blanks

References:

- Herman, J.R., P.K. Bhartia, O.Torres, C. Hsu, C. Seftor, E. Celarier, Global Distribution of UV-Absorbing Aerosols From Nimbus-7/TOMS Data, J.Geophys. Res., 102, 16,911-16,922, 1997.
- Torres, O., P.K. Bhartia, J.R. Herman, Z. Ahmad, and J. Gleason, Derivation of aerosol properties from satellite measurements of backscattered ultraviolet radiation: Theoretical basis, J. Geophys. Res. 103, 17099-17110, 1998.

UV Erythemal Irradiance

data/uvYYYY/gaYYMMDD.epe

Daily uv data. RDGRID.FOR may be used to read these data but see 1README.UV and erynotes.pdf for information about interpreting/decoding the uv data file format.

The values are in 3 digit groups 111222333444555

123 = 2.3x10^1 J/m2 -23 = 2.3 --3 = 0.3 999 = missing or bad data

---

data/monthly_averages/gmYYMM.ept .

This directory contains gridded monthly (gm) averages of ozone data from year YY and month MM. At least 20 days of data must be available for monthly averages to be considered valid.

Monthly averages of aerosol, UV and reflectivity are also provided in directories located in a path structure similar to that for ozone averages.

The Fortran statements that wrote the data records of these monthly files are:

```
      DO 570 IL=1,180
      DO 560 jj=1,288
 560  OZONE(JJ)=ozav(JJ,IL)
      write(21,'(1X,25I3)') (OZONE(jj),jj=1,275)
      write(21,'(1X,13I3,1A17)') (OZONE(jj),jj=276,288), latlab(IL)
 570  CONTINUE
```

---

data/overpass/OVPxxx.ept

A subset (seconds UT, latitude, longitude, total ozone, surface reflectivity, solar zenith angle,aerosol index, SOI index, etc) of EarthProbe data specific to a particular latitude-longitude location. xxx designates a numerical site location as listed in file "1Sitelist" in this directory.

Fortran statements that can be used to read the 1st header record and the data records in the OVPxxx files are the following:

For the header record:

```
          CHARACTER*28 site_name
          INTEGER*4    site_id, site_alt
          REAL*4       site_lat, site_lon
          Read(lun,2) site_name, site_id, site_lat,site_lon, site_alt
        2 Format(A30,4X,I3,7X,F7.2,7X,F7.2,7X,I4)
```

and for the data records:

```
Format(F7.1,1X,I4,1X,I3,1X,I5,2X,I2,1X,F6.2,1X,F7.2,1X,
I3,1X,I3,1X,F5.2,1X,F5.1,1X,F5.1,1X,F6.2,1X,I4)
```

1README, in this directory, provides more information about the OVPxxx files.

---

data/zonal_means/zm_month.ept and zmday_ YY.ept

Zonal means files. File "zm_month.ept" contains monthly averages of zonal mean column ozone over each 5 degrees of latitude for each year of EarthProbe operation to date. Additional files, named "zmday_YY.ept" (YY=96, 97...) provide zonal means on a daily basis. At least 2/3 of possible data must be present for a zonal mean to be considered valid.

The Fortran statements that wrote the header and data records of file zm_month are the following:

```
         CHARACTER*69 MONLAB
         Data MONLAB/'Jan   Feb   Mar   Apr   May   Jun   Jul   Aug   Sep
        & Oct   Nov   Dec'/
         do 20 j=1,37
         lat(j)=-95 + 5*j
      20 continue
         write(22,'(8x,A69)') MONLAB
         DO 300 IZ=1,36
         write(22,'(I3,I4,12f6.1)') lat(IZ),lat(IZ+1),(ozm(IM,IZ),IM=1,12)
    300  CONTINUE
         write(22,'(I3,I4,12f6.1)') -65,0,(ozm(IM,37),IM=1,12)
         write(22,'(I3,I4,12f6.1)') 0,65,(ozm(IM,38),IM=1,12)
         write(22,'(I3,I4,12f6.1)') -65,65,(ozm(IM,39),IM=1,12)
```

The Fortran statement that wrote the data records of the zmday_YY files is:

```
          write(21,'(A12,F9.3,1X,36F6.1,3X,3F6.1)') date,yd,zmd
```

If these EarthProbe/TOMS data are downloaded and used in publication, please give proper credit to the

NASA/GSFC TOMS Ozone Processing Team (OPT).

For more information on the EarthProbe/TOMS data set, contact

Dr. Richard D. McPeters
Code 916
NASA Goddard Space Flight Center
Greenbelt, MD 20771
mcpeters@wrabbit.gsfc.nasa.gov

AVHRR Files

The *Advanced Very High Resolution Radiometer*, flown on a NOAA satellite, measures aerosol data which are analyzed for aerosol optical depth and posted daily at an FTP site.

Format of the data

AEROSOL WEEKLY 100KM ANALYZED FIELD FILE
CEMSCS DSN = PRD.AEROSOL.FIELD.KM100

1) DATASET ABSTRACT

An aerosol optical thickness 100km analyzed field file consists of a specific set of information pertaining to global latitude and longitude intersections. The 1° resolution file, includes the area from -180° to +179° longitude and from -70° to +70° latitude.

2) FILE STRUCTURE DESCRIPTION

The file consists of one documentation record (record number one), followed by one record for each latitude or row of the field. Record 2 or the first latitude row is the southernmost row or the 70° South row. Each row consists of seven words (28 bytes) of information for each longitude column forming a grid intersection plus one seven-word unit containing the row number identification and the date and time of the last analysis made for the field. The first grid intersection of each row is the 180° west meridian or the date line. Gridpoints proceed to the east across the record from left to right ending with the 179° East meridian . The documentation record is created from a namelist dataset and is displayed in namelist format, although it is stored as a binary record. The record size is 10,108 bytes.

3) DOCUMENTATION RECORD FORMAT

A detailed listing of the documentation record format is provided here. This is the first record of the file, and contains the valid time, minimum and maximum latitudes and longitudes of the area covered, together with the grid resolution. If also contains tne number of rows (latitudes), the number of columns (number of longitudes plus one), and information about the currency and reliability of the data.

4) DATA RECORD FORMAT

Each data record (latitudinal row) consists of a series of grid intersection points. These points are 28 bytes in length. Each longitude (column) reflects one grid intersection. At the end of the data record (i.e., immediately following the last column) is a 28 byte row identifier.

GRID INTERSECTION

WORDNO. -- LENGTH in BYTES

DESCRIPTION -- UNITS -- COMMENTS

1 -- 2

Optical Thickness -- (x1000) -- 0 to 2440

1 -- 2

Average Gradient -- units/100KM (x1000) -- 0 to 300

2 -- 2

Gradient X+ -- units/100KM (x1000) -- 0 to 300

2 -- 2

Gradient X- -- units/100KM (x1000) -- 0 to 300

3 -- 2

Gradient Y+ -- units/100KM (x1000) -- 0 to 300

3 -- 2

Gradient Y- -- units/100KM (x1000) -- 0 to 300

4 -- 1

Physiographic Descriptor -- 0 = sea 1 = land -- 0 to 15

4 -- 1

Spare -- Undefined -- 0 to 255

4 -- 1

Number Observations -- Integer -- 0 to 255

4 -- 1

Age Recent Observation -- Hours -- 0 to 255

5 -- 2

(Weight) Wxy -- Integer -- 0 to 32767

5 -- 2

Class 1 Coverage -- Bits -- 0 to 1

6 -- 1

Spatial Covariance X+ -- Grid Units -- 0 to 10

6 -- 1

Spatial Covariance X- -- Grid Units -- 0 to 10

6 -- 1

Spatial Covariance Y+ -- Grid Units -- 0 to 10

6 -- 1

Spatial Covariance Y- -- Grid Units -- 0 to 10

7 -- 2

Climatological Temp. -- oC (x10) -- -850 to +610

7 -- 2

Spare -- Undefined -- 0 to 32767

All parameters are stored as integer values.

DEFINITION OF TERMS IN GRID INTERSECTION

Optical Thickness -

The latest Aerosol Optical Thickness calculated based on the previous analysis optical thickness, weighted according to its reliability, combined with a weighted average of current observations within a surrounding area which is determined according to the grid point's gradient.

Average Gradient -

The average of the gradients in all four directions (N, S, E, W) from the grid intersection.

Gradient in X+ Direction -

Change in optical thickness between the grid point and neighbor points within the field in the positive direction along the X axis.

Gradient in X- Direction -

Change in optical thickness in the negative direction along the X axis.

Gradient in Y+ Direction -

Change in optical thickness in the positive direction along the Y axis.

Gradient in Y- Direction -

Change in optical thickness in the negative direction along the Y axis.

Physiographic Descriptor -

The land/sea tag indicating whether a grid intersection is a land or sea point.

Spare -

Unused parameter.

Number of Observations -

The total number of current observations used in the analysis of the new optical thickness for the grid intersection.

Age of Most Recent Observation -

The age, in hours from the time of last analysis, of the most recent observation used to determine the new optical thickness for a grid intersection.

Reliability -

New reliability associated with the new optical thickness, based on the previous reliability combined with the weighted reliability of all observations used in the last analysis.

Class 1 Temporal Coverage -

Set of bits (0-15) of which bit 1 is set to 1 for each analysis which included observations with a reliability greater than or equal to a specific minimum reliability considered as class 1. Bit 0 always remains a 0, and all bits are shifted right during each analysis leaving bit 1 a 0 when no class 1 reliability observations are used for a grid intersection.

Spatial Covariance X+ -

The distance in grid units from the grid intersection to the nearest land mass in the positive direction along the X axis.

Spatial Covariance X- -

The distance in grid units from the grid intersection to the nearest land mass in the negative direction along the X axis.

Spatial Covariance Y+ -

The distance in grid units from the grid intersection to the nearest land mass in the positive direction along the Y axis.

Spatial Covariance Y- -

The distance in grid units from the grid intersection to the nearest land mass in the negative direction along the Y axis.

Independent Grid Temperature -

The average sea surface temperature of a grid intersection for a particular month over a number of years, taken from the global climatology file.

ROW IDENTIFICATION INFORMATION

Full Word -- Length Bytes

Description -- Units -- Comments

1 -- 4

Row -- Integer -- 1-141

2 -- 4

Spare -- Undefined --

3 -- 4

Spare -- -- Undefined

4 -- 1

Marker -- Integer -- Always 255

4 -- 3

Spare -- Undefined --

5 -- 4

Hour of Day, Minutes of Hour -- (100 x Hours) +Minutes -- 0-2359

6 -- 4

Day of Year -- Days -- 1-366

7 -- 4

Year -- Years --

NOTE: All parameters are stored as integer values. Words 5 to 7 are the date and time at which analysis was performed.

AVHRR DOCUMENTATION RECORD FORMAT

Word # ---- Parameter --- Description

1 ---- LDBGN

Record number of the first row of the field (currently 2 for all fields since the documentation record requires only one record).

2 ---- SMGLAT

Minimum latitude included in field which is the bottom edge and first row of field.

3 ---- AXLAT

Maximum latitude included in field which is the top edge and last row of field.

4 ---- SMLONG

Minimum longitude included in field which is the left edge and first column of field.

5 ---- AXLONG

Maximum longitude included in field which is the right edge and last column of field (excluding the I.D. column).

6 ---- RES

Number of latitude/longitude degrees between each grid point.

7 ---- SMHOUR

Youngest time, in hours of the year, of observations used during last analysis, which becomes the oldest time allowed for the next analysis. If the difference between this time and time of next analysis is greater than the maximum time gap allowed, SMHOUR for the beginning of the next analysis is reduced to make the difference equal to the maximum time gap.

8 ---- HOURS

Oldest time, in hours of the year, of observations used during last analysis.

9 ---- TIMGAP

Number of hours between youngest and oldest times of observations used in analysis.

10 ---- MAXDAT

Maximum number of hours allowed in time period for observation times to be included in analysis.

11 ---- SMREL

Minimum reliability of observations to be used in analysis.

12 ---- AXREL

Maximum reliability of observations to be used in analysis.

13-22 ---- SORC(10)

List of source codes of observations to be used in analysis.

23-32 ---- OBTYPE(10)

List of observation types allowed to be used in analysis.

33 ---- NROWS

Number of rows (latitudinal parallels) included in field, excluding documentation record.

34 ---- NCOLS

Number of columns (longitudinal meridians) in field, including the I.D. column.

35 ---- IBLK

Number of rows or logical records per physical block.

36 ---- NWRDS

Number of fullwords (32 bits) allocated to each grid point.

37 ---- ISZ

Number of rows to be maintained in an array in core for optical thickness and analysis and calculation of gradients.

38 ---- ICENT

Center line within the array upon which calculations will be performed.

39-41 ---- LWT, LNT, LBT

Word number, length in bits, and starting bit location of optical thickness within a grid intersection information unit of an SST Field.

42-44 ---- LWG, LNG, LBG

Word number, length in bits, and starting bit location of Average Gradient.

45-47 ---- LWGXP, LNGXP, LBGXP

Word number, length in bits, and starting bit location of Gradient X+ direction.

48-50 ---- LWGXN, WNGXN, LBGXN

Word number, length in bits, and starting bit location of Gradient X+ direction.

51-53 ---- LWGYP, LNGYP, LBGYP

Word number, length in bits, and starting bit location of Gradient Y+ direction.

54-56 ---- LWGYN, LNGYN, LBGYN

Word number, length in bits, and starting bit location of Gradient Y- direction.

57-59 ---- LWPD, LNPD, LBPD

Word number, length in bits, and starting bit location of Physiographic Descriptor.

60-62 ---- LWNO, LNNO, LBNO

Word number, length in bits, and starting bit location of Number Observations.

63-65 ---- LWAGE, LNAGE, LBAGE

Word number, length in bits, and starting bit location of Age Recent Observation

66-68 ---- LWREL, LNREL, LBREL

Word number, length in bits, and starting bit location of Reliability.

69-71 ---- LWCLS, LNCLS, LBCLS

Word number, length in bits, and starting bit location of Class 1 Coverage.

72-74 ---- LWSXP, LNSXP, LBSXP

Word number, length in bits, and starting bit location of Spatial Covariance in the positive X direction.

75-77 ---- LWSXN, LNSXN, LBSXN

Word number, length in bits, and starting bit location of Spatial Covariance in the negative X direction.

78-80 ---- LWSYP, LNSYP, LBSYP

Word number, length in bits, and starting bit location of Spatial Covariance in the positive Y direction.

81-83 ---- LWSYN, LNSYN, LBSYN

Word number, length in bits, and starting bit location of Spatial Covariance in the negative Y direction.

84-86 ---- LWIND, LNIND, LBIND

Word number, length in bits, and starting bit location of Independent Temperature.

87-96 ---- GRDWTS(10)

Weight assigned to each grid unit, according to its distance from the grid intersection for which gradients are being calculated.

97 ---- NP

Number of grid points to be used in calculation of gradients.

98-117 ---- KMDST(10,2)

Look up table of gradient values and corresponding distances to be used in determining the search area for analysis.

118 ---- MKM

Number of paired entries in KMDST.

119-138 ---- H(10,2)

Look up table of gradient values and corresponding factors to be used in determining the new weight assigned to the observation temperature for analysis.

139 ---- MH

Number of paired entries in H.

140 ---- EXP

Exponent used in temperature analysis.

141 ---- FDX

Factor used in determining new weight assigned to the optical thickness observation for analysis.

142 ---- XCLASS

Factor used to place gradients into a class for Gradient Class Summary.

143 ---- DEL

Maximum number of optical thickness units that the new analysis temperature may differ from the previous optical thickness field value.

144 ---- MF

Factor applied to the previous optical thickness and reliability to determine the final optical thickness and its reliability.

145 ---- MSTAR

Factor applied to the combined observations optical thickness and weight in determining the new analysis optical thickness.

146 ---- MNSRCH

Minimum distance in kilometers to be searched for analysis observations.

147 ---- MXSRCH

Maximum distance in kilometers to be searched for analysis observations.

148 ---- BDEL

Maximum difference allowed between new analysis optical thickness and the previous one for the Class 1 Coverage Bit to be set to 1.

149 ---- FCWT

Maximum value that can be assigned as the reliability of the new analysis optical thickness.

150 ---- IYYY

Year of youngest time of observation data used (0-99).

151 ---- IYMM

Month of youngest time of observation data used (1-12).

152 ---- IYDD

Day of youngest time of observation data used (1-31).

153 ---- IYHH

Hour of youngest time of observation data used (0-23).

154 ---- IOYY

Year of oldest time of observation data used (0-99).

155 ---- IOMM

Month of oldest time of observation data used (1-12).

156 ---- IODD

Day of oldest time of observation data used (1-31).

157 ---- IOHH

Hour of oldest time of observation data used (0-23).

158 ---- ICURTM

Last time used in analysis.

Values are stored as real (IBM floating-point) or integer according to the format implied by the first letter of their label. (Parameters beginning with I,J,K,L,M,N are integer values.)

IBM360 Floating points

This floating point representation is not compatible with the IEEE floating point specification which is used by most modern computers. The IBM360 float was native to the now obsolete IBM 360/370 series of computers, and a few others. Since documentation of this representation is getting difficult to find, a summary is provided here.

The 4-byte floating point representation is divided into one byte (SEXP) for sign and exponent, followed by a 3-byte MANTISSA. The mantissa should be thought of as a six digit hexadecimal number, with a radix point preceding the first digit. The numeric value of the MANTISSA is the value of the integer with the same hex representation, divided by 0x1000000, and ranges from 0.0 to just less than 1.0. For a non-zero, normalized float, the first hex digit will range from 0x1 to 0xf.

In the SEXP byte, the high order bit is the sign of the number, 0 for plus and 1 for minus. The remaining seven bits are a biased exponent. After subtracting 0x40, the result is the number of hexidecimal digits (powers of 16) to shift the radix point right (if positive) or left (if negative). This means the range of the representation is from 16^63 to 16^(-64).

ADVANCED - Special Topics

This section provides some guidance in configuring the model input to for certain specialized calculations. The default configuration supplied with the test meteorological data is confined to a simple trajectory and inert transport and dispersion calculation. Some of these more complex scenarios are configured through the *Advanced* menu Configuration Setup tab which modifies the "*SETUP.CFG*" namelist file.

Particle or Puff Releases

The concentration model default simulation assumes a particle dispersion in the vertical direction and a top-hat puff dispersion in the horizontal direction. Other options are set with the *INITD* parameter of the *SETUP.CFG* namelist file defined in the advanced menu section. Normally changes to the dispersion distribution are straightforward. However there are some considerations with regard to the initial number of particles released. The default release is set to be 500 particles over the duration of the emission cycle (see NUMPAR). A 3-dimensional (3D) particle simulation requires many more particles to simulate the proper pollutant distribution, the number depending upon the maximum downwind distance of the simulation and the duration of the release, longer in each case require more particles. Too few particles result in noisy concentration fields. A 3D puff simulation starts with one puff as the puff-splitting process in conjunction with the vertical dispersion quickly generates a sufficient number of puffs to represent the complex dispersion process. The default configuration represents a compromise in permitting particle dispersion in the vertical for greater accuracy and puff dispersion in the horizontal to limit the particle number requirements.

Continuous Emissions

As noted above the default release is 500 particles over the duration of the emission cycle. If continuous emissions are specified (e.g. over the duration of the simulation), then those 500 particles are spread out over that time period. This may easily result in the release of too few particles each hour to provide smooth temporal changes in the concentration field. Imagine a single particle passing in and out of the vertical concentration cell due to turbulent diffusion. One solution would be to increase the *NUMPAR* parameter until smoother results are obtained. Another possibility would be to cycle the emissions by emitting 100 particles only for the first time step of each hour. Those particles would contain the total mass for a one-hour release (see how to set *QCYCLE*).

Area Source Emissions

Normally emissions are assumed to be point or vertical line sources. Virtual point sources (initial source area >0) can be defined two ways: 1) through the definition of an initial area on the source location input line of the CONTROL file or 2) by the definition of a gridded emissions file. If the model's root startup directory contains the file *Emission.txt*, then the pollutants are emitted from each grid cell according to the definitions previously set in the *Control* file. Two source points should be selected, which define the lower left (1st point) and upper right (2nd point) corner of the emissions grid that will be used in the simulation. This should be a subset of the grid defined in *emission.txt*. The release height represents the height from the ground through which pollutants will be initially distributed. Note that the structure of the "*emission.txt* " file has changed with the Hysplit 4.6 revision of October 2003.

The "emission.txt" file contains all the information that is required to interpret the data in the gridded emission inventory file. The file that contains the inventory is now independent of the emission.txt file. The file's first record contains information about the internal grid cell size that is used by the dispersion model to accumulate the file's emissions. The emission file defines the emissions at latitude-longitude points, which may represent the emissions from an area or from a point. The values at these points are accumulated in an internal grid, the size of which is defined on the first record. This value can be arbitrarily changed according to the desired resolution of the simulation. The pollutant puffs are released with an initial size comparable to the accumulation cell size. Because the emission file data are remapped to an internal grid, the file can consist of emissions data on a regular grid or just a collection of individual cells. The emission rate in the Control file is used as an additional multiplication factor for the data in the emission file. Also note that previously discussed particle number restrictions still apply. The particles are spread out over the duration of the emission and the number of grid cells that are defined in the emission domain. The format of the emission.txt file is given below:

Record #1

I4 - Number (n) of pollutant species in file

I4 - Number of emissions defined for each 24 hour period

F10.4 - Conversion factor: file units to model units/hour

2F10.4 - Accumulation cell size (latitude & longitude)

Record #2

nA4 - Four character pollutant identification string for each pollutant

Record #3

A - The /directory/filename of the emission data file

The actual emission data file will contain one record identifying the grid location and then two records for each pollutant species. The first record defines emissions from GMT hours 0 to 12 and the second record from hours 12 to 24. This pair of records is repeated for each pollutant species:

Records Loop #1 to the number of i,j grid point

2I4 - I,J grid point of emission cell (arbitrary units for identification)

2F10.4 - Southwest corner Longitude and Latitude of this emission cell

-

Record Loop #2 to the number of pollutant species

12E10.3 - emissions for pollutant#1 hours 1-12

12E10.3 - emissions for pollutant#1 hours 13-24

Multiple Pollutants

The model can easily be configured to simulate more complex pollutant episodes with multiple pollutant types on different particles or multiple pollutant species on the same particle. The former is accomplished by defining additional pollutants in the *CONTROL* file. In this configuration, multiple species are emitted, have no interaction, and may track differently. This situation may represent a volcanic ash plume, where each pollutant, a different sized particle, settles at a different rate. An example configuration "*volc_conc*" can be retrieved in the *SETUP* menu.

Pollutant Transformations

In the latter situation, when multiple pollutants are defined on the same particle, an external chemistry routine is required that converts mass from one species to another, all tracking together (advecting and dispersing). In this situation, *MAXDIM* should be raised to the required value. Increasing the *MAXDIM* value always requires an external routine to adjust the mass between species. A simple species conversion program is included with the standard model distribution. In the default configuration it is only necessary to define two different pollutants in the [concentration setup](#) menu and select the 10% /hr checkbox in the [advanced configuration](#) menu's conversion section. This option combination sets *MAXDIM=2* and calls the transformation routine every time step to convert pollutant #1 to #2 at a rate of 10% per hour. Other conversion rates or a greater number of pollutants can be defined by creating the *CHEMRATE.TXT* file in the local directory. This file would consist of one or more records, each record defining a pollutant conversion. The data are free-format and consist of four fields, the integer "from" and "to" pollutant index numbers, and the real hourly conversion "rate", and molecular weight adjustment "factor". For instance, if the file were to be defined for the default case, the one data record would have the following values: (1 1 0.10 1.0). The molecular weight adjustment factor can be used to account for other reactions not considered in the simple conversion module. For instance, if one were to define pollutants #1 and #2 as SO2 and SO4, respectively, then the molecular weight adjustment factor should be 1.5 as SO2 transforms to SO4 (the conversion picks up two additional oxygen molecules).

Complex Chemistry

Although there are other more complex chemical conversion modules available for HYSPLIT, they are not incorporated into the standard compilation. More information on these special compilations may be found at http://www.arl.noaa.gov/ready/hyspchem

One feature, required for all these modules, is that there is a more complex interaction between the individual pollutant plumes, requiring a close link between the concentration grid and the meteorological data grid. This option is available in the standard model compilation. By setting the namelist file parameter ICHEM=4, the concentration grid is redefined to be equal to the meteorological data grid in terms of spatial resolution and extent. This simplifies the computation of the grid based chemical reactions that are dependent upon the meteorological conditions within each concentration grid cell.

Deposition and Decay

A simple particle deposition configuration (*rsmc_conc*) for radioactive Cs-137 can be retrieved into the *SETUP* menu, which shows the default settings for radioactive decay and wet and dry deposition. In conjunction, a list of sites can be loaded into any *SETUP* menu, from the "Set Starting Locations" tab by pressing the LIST button. The site locations can be found in the file "*\working\plants.txt*" and could be replaced by any user generated location file listing.

The normal deposition mode is for particles to loose mass to deposition when those particles are within the deposition layer. An additional option was added to deposit the entire particle's mass at the surface, that is the particle itself, when subjected to deposition. To insure the same mass removal rates between the two methods, a probability of deposition is computed, so that only a fraction of the particles within the deposition layer are deposited in any one time step. The probability of deposition is a function of the deposition velocity, time step, and depth of the layer. One limitation of this method is that only one mass species may be assigned to a particle. The probability deposition method can be invoked from the namelist file with ICHEM=5.

Compilation Limits

With Hysplit V4.5 most compilation array limits have been eliminated through the use of dynamic array allocation. However, one restriction remains with regard to the meteorological input data: meteorological data files are limited to a maximum of 10 per simulation, with no more than 35 levels or variables in each file. This restriction does not limit any computation with data files available through the ARL web site, because all available data files meet the number of variable and levels restriction.

The use of dynamic memory allocation can result in unpredictable results if the computer's hardware memory limits are exceeded. Although there are several memory error allocation traps that will result in a message and execution termination, memory limits can be exceeded in a variety of different locations, such as when opening a file. Memory usage is a primarily a function of the meteorological sub-grid size, meteorological data grid size, concentration grid size, and the number of pollutants.

Script Automation and Configuration for Operational Applications

Most of the discussion in various sections of the User's Guide are tailored to individually configured simulations. However there are several features to the model that can be used to automate the computational environment. For instance, a sample *Auto_traj.tcl* script is provided in the *\trajmdl* directory that can be used as a guide to automate many applications.

```
# Auto_traj.tcl

# the next line restarts using wish \

# exec wish "$0" "$@"

set Start_hgt "10.0"

set Traj_path "../trajmdl"

set Start_time "00 00 00 00"

set Run_hours "24"

set Vert_coord "0"

set Top_model "10000.0"

set Meteo_path "../metdata/"

set Meteo_file "oct1618.BIN"

set Output_path "./"
```

```
set Output_base "tdump"

set Output_numb 1

foreach {Start_lat Start_lon} {35.0 -90.0 40.0 -90.0 45.0 -90.0} {

set Start_loc "$Start_lat $Start_lon $Start_hgt"

set Output_file "$Output_base$Output_numb"

file delete Control

set f [open Control w]

puts $f "$Start_time"

puts $f "1"

puts $f "$Start_loc"

puts $f "$Run_hours"

puts $f "$Vert_coord"

puts $f "$Top_model"

puts $f "1"

puts $f "$Meteo_path"

puts $f "$Meteo_file"

puts $f "$Output_path"

puts $f "$Output_file"

close $f

exec "$Traj_path/hymodelt.exe"

incr Output_numb}
```

In this particular example the test trajectory case is run for three different starting locations, each simulation writing a new endpoints file with a unique file name. The *CONTROL* file is recreated for each simulation. It would be trivial to rewrite the script to set the latitude-longitude and loop through a different number of starting days and hours. With Tcl/Tk installed, this script can be run under Windows or Unix. For instance, to compute new forecast trajectories each day, the process can be automated by including a data FTP at the beginning of the script to get the most recent meteorological forecast file, setting the starting time as "00 00 00 00" so that the trajectories will start at the beginning of the file, and finally calling the script once-a-day though the Unix crontable or the Window's scheduler commands.

One problem with automated operations is that it is possible to generate simultaneous multiple jobs which may interfere with each other. The executables, Hymodelc and Hymodelt have a command line option of adding the process ID (PID): e.g. hymodelt [PID]. In this situation all standard named input and output files [those not defined in the Control file] have the PID added as a suffix to the file name: e.g. Control.[PID], Setup.[PID], Message.[PID].

An example of another type of operational configuration is the extended simulation of a pollutant emission using archive data to bring the simulation to the current time and then using forecast meteorological data to provide a daily projection. Each day the archive simulation must be updated with the new archive data and a new forecast product generated. This process can also be automated through a script, but for illustration purposes one can use the advanced features of the GUI to configure such a case. Assume a one-hour duration accidental pollutant release that occurred 48 hours prior to the current time. The following sequence applies:

1) From the "*Meteorology*" menu tab download the appropriate archive meteorological data and the most recent forecast meteorological data (assume it is available to +48h).

2) Setup the concentration simulation to run 96 hours using two meteorological files starting with the archive data and then switching to the forecast data.

3) Under the *Advanced* menu tab and *Configuration Setup* write the initialization file after 72 hours.

4) Run the model.

At the completion of the simulation you will have the plume projection from release (-48 h) through the current forecast (+48 h). The *PARDUMP* file will contain all the endpoint positions at +24 hours, corresponding to the initialization time of when the next forecast will be available (assume there is one forecast per day).

The next day, when the new forecast data are available, reconfigure the model to run only with the forecast meteorological data for a duration of 48 hours and write the initialization file after 24 hours, then run the model to obtain the new projection. In this second part we assume that the first 24 hours of the forecast are not much different than the analysis. In practice, this procedure can be run at the same frequency that the new forecast data are available and typically at 4 times per day, data at the initial forecast hour are identical to the analysis data.

Backward Dispersion Simulations

A common application of atmospheric trajectory and dispersion models is to try to determine the source of a pollution measurement. If a high value has been collected at a particular receptor, from which pollutant source region did the air originate? One common approach is the calculate the trajectory "backwards" from the receptor site. In the trajectory calculation this is accomplished by setting the integration time step to a negative value. However the trajectory only represents the upwind path of a single point, while the pollutant measurement may require of hundreds or thousands of trajectories to represent the dispersion of the pollutant in time and space.

Another approach is the run the entire dispersion-trajectory model "backwards" which is computationally attractive because in a 3D particle model the dispersion process is represented by a turbulent component added to the trajectory calculation and the advection process is fully reversible. The trajectory equation can be correctly integrated in either direction. The interpretation of the output is a bit more complex because dispersion is an irreversible process. Although the equivalent numerical calculation will yield a result because the integration of the dispersion equation is still in the normal downstream mode on top of the backward upstream integration of the advection. The meaning of the upwind dispersion result is less clear. In any event as noted in the earlier instructions it is possible to run the dispersion model "backwards" by setting the run duration and emission hours to their equivalent negative value. The stop time of the sampling should be set prior to the start time. All start and stop times should be set to their exact values - relative start-stop times are not supported in the backward mode. To simplify interpretation of the results, horizontal dispersion is "turned off" in the backward calculation, resulting in a more reversible calculation. An example configuration (*back_conc*) can be retrieved into the *SETUP* menu.

Time Variation of the Emission Rate

One way to incorporate a time varying emission rate into the existing model structure is to use the particle dump feature to restart the model each time with a new emission rate. Another option is to assign the name of a temporal emission input file to the "EFILE" variable in the setup.cfg namelist file. This ascii file must consist of at least three records, the first two of which are used for identification purposes, and the third, and all subsequent records, define the temporal sequence of emissions. Each emission record contains the start time, duration, location, and emission rate. If the EFILE is present, the first emission record's values replace the emission values set in the control file. Once the model computation time has passed the emission period defined on the first emission record, the emission data from the second record are loaded and the calculation continues with the new emission data. The format of the emission file is given below:

Record #3 -> end

I4 - Start year

I3 - Start month

I3 - Start day

I3 - Start hour

I2 - Start minute

I3 - Duration hours

I2 - Duration minutes

F6.2 - Latitude

F8.2 - Longitude

F8.0 - Emission rate in units/hour

Transport of Particles Deposited on Water Surfaces

The main code was modified (October 2003 Version 4.6) to permit particles deposited on water surfaces to continue to be transported on the water surface by the wind generated drift current. The transport output is treated as a deposition surface for display purposes. This new deposition method then creates particles that can be transported on water surfaces. Particles can be deposited on any surface. However, if the surface is defined as water, then the particle is assigned a unique identification code to distinguish it from atmospheric particles or puffs. These new particles may continue to be transported along the surface of the water contributing to deposition each time step but not air concentration. Dispersion is not computed for these particles. When they approach a land surface they are deleted. The water surface transport option is invoked from the namelist file with ICHEM=7. This option automatically forces the probability deposition computation (ICHEM=5) and should only be used only with the 3D particle mode (IN ITD=0). Surface water deposition can only be displayed if the deposition output level (0) is defined. Although particles may deposit over land, over-land deposition values are never shown.

The wind induced surface water drift current is assumed to equal the vector atmospheric friction velocity. The friction velocity represents the momentum transport to the surface and it is an approximation of the surface water movement. Currently only the GFS meteorological model output file contains the vector momentum flux components.

ADVANCED - Particle Dump File Format

The concentration configuration menu provides an option to write a model initialization file, which by default is always named "PARDUMP" (for particle dump). This file can be written at regular intervals during the simulation, a convenient way to restart a simulation in case of unexpected failure. To restart the model using the PARDUMP file it is only necessary for the file to be present in the root working directory. If the internal time stamp of the file matches the start time of the simulation, the model will initialize the particle count from the file before emitting new particles according to the emission scenario defined in the control file. The format of the PARDUMP file is given below:

Record #1

INT*4 Number of particles

INT*4 Number of pollutants

INT*4 Time of particle dump (YEAR, MONTH, DAY, HOUR)

Record #2 Loop to record: Number of particles

REAL*4 Particle pollutant mass (times the number of pollutants)

REAL*4 Particle LATITUDE, LONGITUDE, HEIGHT, SIGMA-U,

SIGMA-V, SIGMA-X

INT*4 Particle AGE, DISTRIBUTION, POLLUTANT, METEO-GRID,

SORT-INDEX

The "Particle" tab of the "Special File Display" menu brings up a Windows based viewer that shows the particle positions over a map background. The display can be zoomed and otherwise adjusted using the left and right mouse buttons in conjunction with the shift and cntl keys. Help is provided on the screen with the left and right side comments corresponding to the respective mouse button. The particle viewer can also be used to overlay satellite images on the particle positions. More information on this is provided "FTP Satellite Data" help menu.

ADVANCED - Message File Format

A MESSAGE file is created during each simulation. The file contains information from certain key subroutines that can be used for diagnostic purposes if a simulation were to fail. An example file is shown below in the left column from the test concentration simulation. A trajectory MESSAGE file would be similar but without the particle number and mass information. In the right column, in red, a more detailed description of each item is provided.

HYSPLIT46 (Oct 2003)

-- Start Namelist configuration --

Internal grid parameters (nlvl,aa,bb,cc): 19 30.0 -25.0 5.0

&SETUP INITD = 4, KHMAX = 9999, NUMPAR = 500, MAXPAR = 10000, MAXDIM = 1,

QCYCLE = 0.0000E+00,

FRME = 0.10000000,

FRMR = 0.0000E+00,

KRND = 6,

DELT = 0.0000E+00,

ISOT = 0,

TKER = 0.50000000,

NDUMP = 0,

NCYCL = 0,

TRATIO = 0.75000000,

MGMIN = 10,

KMSL = 0,

NSTR = 0,

CPACK = 1,

ICHEM = 0,

NOUT = 0, TM_PRES = 0, TM_TPOT = 0, TM_TAMB = 0, TM_RAIN = 0, TM_MIXD = 0,

DXF = 1.0000000,

DYF = 1.0000000,

DZF = 9.9998E-03,

NINIT = 1,

PINPF = PARINIT,

POUTF = PARDUMP

-- End Namelist configuration ---

NOTICE main: pollutant initialization flags

Gas pollutant - T

Current version and release date

Tabulates value of namelist variables

Number of internal model sigma levels and the polynomial parameters used to describe the vertical grid. These are configured automatically based upon the meteorological input data files defined for this simulation.

|

|

|

|

The value of all variables that can be defined by the SETUP.CFG namelist file are listed. If no SETUP.CFG file was defined for this simulation, then the default values for these variables are listed.

|

|

|

|

The settings of all internal deposition flags are shown here. In this case the simulation is for a gas.

NOTICE metpos: (mtime,ftime) - 50379840 0

NOTICE metpos: (mtime,ftime) - 50379960 0

NOTICE advpnt: (kg,nx,ny,nz) - 1 10 10 19

NOTICE sfcinp: reading ASCDATA.CFG

NOTICE metgrd: (kg, xyr,xy1) - 1 10 10 17 9

NOTICE metinp: NGM 1 1 10 10

17 9 50379840 95 10 16 0

##################################

WARNING prfsig: extrapolation from level (k,m): 16 7159.121

Input data levels: 10

Internal Sigma levels: 19###################################

NOTICE metinp: NGM 1 63 10 10 17 9

50379960 95 10 16 2

NOTICE main: Initial time step

(min) 20

NOTICE main: 1 50379860

166 0.3333332 NOTICE main: 1 50379880

332 0.6666647 NOTICE main: 1 50379900

498 0.9999961

NOTICE main: 2 50379930

498 0.9999961 NOTICE main: 2 50379960

498 0.9999961

NOTICE metpos: (mtime,ftime) - 50380080 50379840

NOTICE metinp: NGM 1 125 10 10 17 9

50380080 95 10 16 4

The subroutine that determines which meteorological data are required is called for the first time. Data for times 840 and 960 are requested. The zeros for the second field indicate no data are in memory. Times are always in relative minutes.

The first advection entry sets sub-grid #11 to 10x10 with 19 levels.

The surface boundary files are opened.

The lower left corner of sub-grid #1 is set at position 17,9 of the main meteorological data grid.

The NGM data for the first computational hour are read starting at record #1, loaded into a 10x10 sub-grid, corner 17,9, at time [x]840 for the date: 95 10 16 0

When these data are interpolated to the internal grid, it is determined that there are no input data records above 7159 m, therefore data for those levels are extrapolated.

Computations for the first hour require data at two time periods for interpolation (hours 0 and 2).

The initial time step was set to 20 minutes. Subsequent time steps may change.

The time, number of particles, and the total mass is shown for the three time steps of the first hour. After one hour the emission stops.

During hour two, no further emissions (or particles) are released. The time step is now 30 minutes.

After the hour 2, new NGM data are required. The data in memory at 0 UTC (time 840) are replaced with data at 4 UTC (time 080).

The new data are input into the same sub-grid location.

NOTICE main: 3 50379990

498 0.9999961 NOTICE main: 3 50380020

498 0.9999961 NOTICE main: 4 50380050

498 0.9999961 NOTICE main: 4 50380080

498 0.9999961

NOTICE metpos: (mtime,ftime) - 50380200 50379960NOTICE metinp: NGM 1 187 10 10 17 9

50380200 95 10 16 6

NOTICE main: 5 50380110

498 0.9999961 NOTICE main: 5 50380140
498. 0.9999961

NOTICE main: 6 503801704

498 0.9999961 NOTICE main: 6 50380200

498 0.9999961 Index Height %Mass 6 935.0 1.41 5 630.0 10.84 4 385.0 31.33 3 200.0 29.72 2 75.0 20.48 1 10.0 6.22

Computations proceed as before for computational hours 3 and 4. Particle number remains the same because no particles have moved off the computational domain. The mass remains the same because deposition is not turned on for this simulation.

At the end of hour 4, data are required for 6 UTC (time 200) to proceed with the calculation. These data replace the 2 UTC (time 960) in memory.

Computations proceed to hour 6.

Every 6 hours, the model prints out the vertical mass distribution of all the particles within the computational domain. This is the mass distribution relative to the model's internal sigma levels and there is no relation to the levels that may be specified for the concentration output file. Only non-zero levels are shown. The internal levels are defined by the polynomial parameters given at the beginning of the MESSAGE file.

At this point the computation will continue for the number of hours specified in the CONTROL file. Vertical profiles are shown every 6 hours. As the particles move across the computational domain, the sub-grid position may be moved (from position 17,9) or expanded larger than 10x10) to match the spatial extent of the particle distribution. This may occur at any time during the computation or even multiple times during the same computational hour.

VERIFICATION STATISTICS

Enhancements have been incorporated into HYSPLIT, starting with the December 2002 version, to improve the model's performance for short-range (1-20 km) simulations. The code was restructured to use the velocity variance for dispersion calculations. New equations were added to compute the turbulence directly from boundary layer stability functions (see Kantha and Clayson, 2000, *Small Scale Processes in Geophysical Fluid Flows*). This method does not use the diffusivity and hence no assumptions are required about the turbulence scales. Either equation set (diffusivity or turbulence) can be selected for a simulation. However, simulations using the new turbulence equations require considerably more CPU time than the former diffusivity calculation. An important aspect of introducing a new calculation scheme is to show that the model's performance has actually improved. The computations have been tested against tracer data collected during METREX.

HYSPLIT Verification over Washington, D.C. using METREX data

The Metropolitan Tracer Experiment (METREX) consisted of simultaneous 6-h duration perfluorocarbon releases from two locations every 36-h in the Washington D.C. suburbs for one year (1984). Sequential 8-h air concentrations were collected at three locations in the urban area. Monthly average tracer concentrations were collected at 93 locations. The approximate distance between the sequential sampling locations is 10 km. Although meteorological data were collected during METREX, these files are not compatible with HYSPLIT. The 2.5 deg 6-h NCAR/NCEP reanalysis data were interpolated to a 5 km resolution 30-min interval grid covering a 1 degree square domain centered over Washington D.C. Meteorological tower observations at five locations, collected during METREX, were blended into the gridded data using a linear interpolation procedure. The model was run for1984 for each release location. However, only the results from the monthly sampling network are discussed because they provide the greatest number of sampling locations within 10 km of each release location. The measured data and model results were converted to the DATEM compatible format so that existing statistical programs could be used for data analysis. The results, summarized in the Table shown below, have been averaged for the entire year so that the statistics represent the means and variations between the 93 monthly sampling locations. The PMCH was released from Rockville from January through May and from Lorton, VA the rest of the year, while the PDCH was released from Mt. Vernon the entire year. Absolute PDCH numbers are higher because its release rate was six times that of the PMCH release rate. Concentration units are in pico grams (pg/m^3). The diffusivity approach is the original HYSPLIT mixing scheme and the turbulent velocity method is the new approach for short-range dispersion discussed in the previous item.

Statistic	Reanalysis Data Diffusivity		Reanalysis Data Turbulent Velocity		+ Tower Data Diffusivity		+ Tower Data Turbulent Velocity	
	pmch	pdch	pmch	Pdch	pmch	pdch	pmch	pdch

Correlation	0.90	0.65	0.88	0.59	0.91	0.55	0.86	0.80
Mean Calculated	132	297	130	282	23	70	116	296
Mean Measured	72	385	72	385	72	385	72	385
Ratio C / M	1.83	0.77	1.80	0.73	0.32	0.18	1.6	0.77
NMSE (pg/m^3)	0.93	4.40	1.00	4.85	8.62	23.5	0.86	3.36
Bias (pg/m^3)	+60	-88	+58	-103	-49	-315	+44	-89
Percent x2	40	55	42	54	21	9	48	64
Percent x5	94	98	92	92	73	38	93	98

In general the use of the turbulent velocity variance method provides slightly better results than the diffusivity method only when the finer resolution tower data are introduced into the calculation. These results, shown in the last two columns, have the lowest NMSE and the greatest number of samples within a factor of two of the measurements. The introduction of tower data without corresponding improvements to the model result in a degradation of performance. The poor performance of the model when the older diffusivity method is used with the tower data is believed to be caused by too much vertical mixing. The addition of the tower temperatures to the reanalysis data temperature profiles resulted in larger changes to the vertical mixing in the diffusivity method than in the turbulent velocity method.

The new dispersion equations were also tested against some of the longer range dispersion data used in previous evaluations. These results are shown in the two tables below for the un-averaged global statistics, where all measurements and model calculations are paired in space and time without averaging, and temporally averaged statistics, which represent the spatial correlation of the model calculations. The performance of the newer dispersion methods is comparable to the previous method for these longer range data sets. See the DATEM web page on arl.noaa.gov/datem form more detailed information on each of these tracer experiments.

Global Verification Statistics for Long-Range Experimental Data

Experiment Model R FB FMS KS RANK

ACURATE Apr 2001 0.28 -0.02 18.59 79 1.46

May 2002 0.25 -0.05 18.73 79 1.43

Dec 2002 0.25 +0.48 18.01 79 1.21

ANATEX1 Dec 2002 0.34 0.29 35.09 59 1.73

Dec 2003 0.44 0.18 32.84 59 1.85

ANATEX2 Dec 2002 0.18 0.16 34.06 56 1.73

Dec 2003 0.18 0.19 33.97 56 1.72

ANATEX3 Dec 2002 0.14 0.12 21.01 50 1.67

Dec 2003 0.17 0.0 20.70 56 1.73

CAPTEX Apr 2001 0.01 -0.17 15.34 71 1.36

Dec 2002 0.00 -0.12 16.78 71 1.40

Dec 2003 0.02 -0.27 17.07 71 1.33

INEL74 Apr 2001 0.00 0.13 7.97 92 0.48

Dec 2002 0.00 1.39 7.27 92 0.46

OKC80 Apr 2001 0.16 -0.67 51.15 43 1.77

Dec 2002 0.08 -0.66 27.60 34 1.52

Dec 2003 0.43 -0.87 35.01 43 1.67

ETEX Apr 2001 0.48 0.33 57.27 68 1.96

Dec 2003 0.45 0.78 47.69 68 1.61

Temporally Averaged Verification Statistics for Long-Range Experimental Data

Experiment Model R FB FMS KS RANK

ACURATE Apr 2001 0.91 -0.03 100.0 39 3.43

May 2002 0.90 -0.06 100.0 58 3.20

Dec 2002 0.86 +0.47 100.0 78 2.73

ANATEX1 Dec 2002 0.85 0.26 100.0 18 3.41

Dec 2003 0.85 0.14 100.0 23 3.43

ANATEX2 Dec 2002 0.56 0.13 100.0 26 2.98

Dec 2003 0.48 0.16 100.0 26 2.89

ANATEX3 Dec 2002 0.25 0.05 98.67 48 2.54

Dec 2003 0.20 -.06 98.67 46 2.54

CAPTEX Apr 2001 0.78 -0.18 93.59 10 3.35

Dec 2002 0.80 -0.17 94.87 17 3.33

Dec 2003 0.76 -0.31 94.87 19 3.18

INEL74 Apr 2001 0.13 1.39 100.0 89 1.43

Dec 2002 0.35 1.41 100.0 98 1.44

OKC80 Apr 2001 0.96 -0.65 90.70 14 3.36

Dec 2002 0.96 -0.64 80.0 18 3.21

Dec 2003 0.83 -0.85 90.0 20 2.97

ETEX Apr 2001 0.51 0.23 84.89 20 2.79

Dec 2003 0.59 0.65 82.55 20 2.65