HYSPLIT Training Seminar

Model Overview

- Model history and Features
- Computational method
- Trajectories versus Concentrations
- Code Installation Notes
- Model Operation

Meteorological Data

- Meteorological Data Requirements
- GRIB Data Conversions
- ECMWF ERA-40 Archive
- Forecast Data FTP Access
- Analysis Data FTP Access
- Display Meteorological Grid Domain
- Vertical Meteorological Profile
- Contour Meteorological Data

Particle Trajectory Methods

- Trajectory Computation Method
- Trajectory Example Calculation
- Trajectory Model Configuration
- Trajectory Error
- Multiple Trajectories
- Terrain Height
- Meteorological Data on a Trajectory
- Vertical Motion Options

Pollutant Plume Simulations

- Modeling Particles or Puffs
- Concentration Prediction Equations
- Turbulence Equations
- Dispersion Model Configuration
- Example Dispersion Calculation
- Defining Multiple Sources
- Simulations using Emission Grids
- Concentration and Particle Display
- Converting Concentration Data to Text

Special Topics

- Concentration Ensembles
- Chemistry Conversion Modules
- Modeling Emissions from Dust Storms
- Using the Particle File for Restarts
- Pollutant Deposition
- Source Attribution via Trajectories
- Source Attribution via Matrices
- Source Attribution Functions
- Creating Files for ArcExplorer
- Customizing Map Labels
- Scripting for Automated Operations

HYSPLIT PC TRAINING SEMINAR

HYbrid Single-Particle Lagrangian Integrated Trajectory Model

Roland R. Draxler

National Oceanic and Atmospheric Administration
10 June 2004

History

1 – 1979	rawinsonde data with day/night (on/off) mixing NOAA Tech Memo ERL ARL-112 (1982)
2 - 1983	rawinsonde data with continuous vertical diffusivity NOAA Tech Memo ERL ARL-166 (1988)
3 - 1987	model gridded fields with surface layer interpolation NOAA Technical Memo ERL ARL-195 (1992)
4 - 1996	multiple meteorological fields and combined particle-puff NOAA Technical Memo ERL ARL-224 (1997)

4.0 – 8/1998	switch from NCAR to postscript graphics for PC
4.1 – 7/1999	isotropic turbulence for short-range simulations
4.2 – 12/1999	terrain compression of sigma & use of polynomial
4.3 – 3/2000	revised vertical auto-correlation for dispersion
4.4 – 4/2001	dynamic array allocation and support lat-lon grids
4.5 – 9/2002	ensemble, matrix, and source attribution options
4.6 – 6/2003	non-homogeneous turbulence correction, dust storm
4.7 – 1/2004	velocity variance, TKE, new short-range equations

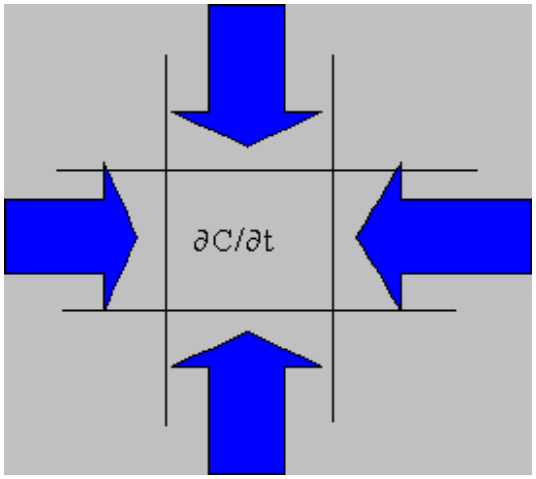
Features

- Predictor-corrector advection scheme
- Linear spatial & temporal interpolation of meteorology from external sources
- Vertical mixing based upon SL similarity, BL Ri, or TKE
- Horizontal mixing based upon velocity deformation, SL similarity, or TKE
- Puff and Particle dispersion computed from velocity variances
- Concentrations from particles-in-cell or Top-Hat/Gaussian distributions
- Multiple simultaneous meteorology and/or concentration grid

Next Page

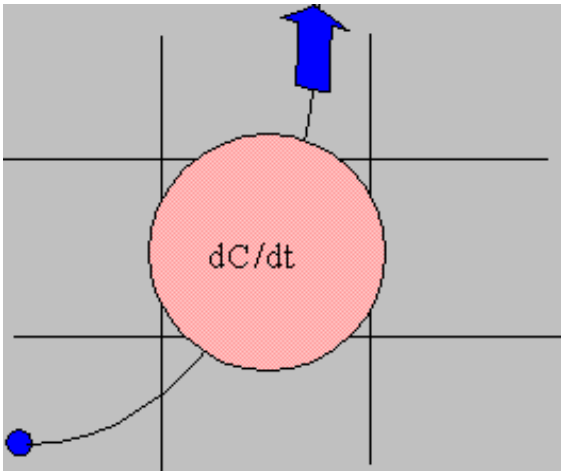
Computational Method

Eulerian Approach



Local derivative
Solve over the entire domain
Ideal for multiple sources
Easily handles complex chemistry

Lagrangian Approach



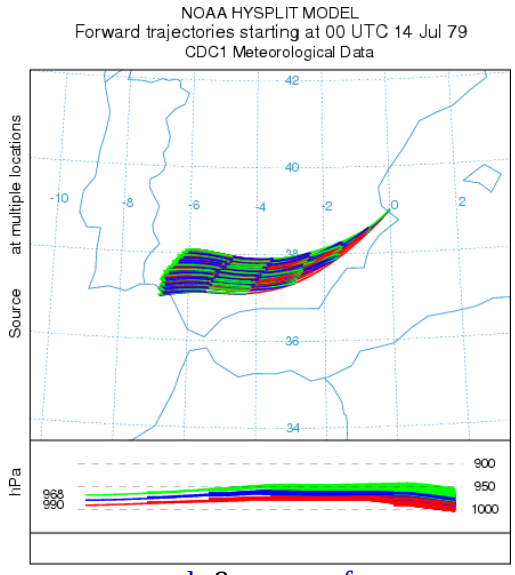
Total derivative
Solve only along the trajectory
Ideal for single point sources
Implicit linearity for chemistry

In the Eulerian modeling approach, air concentrations are computed for every grid cell by integrating the pollutant fluxes at each grid cell interface due to diffusion and advection. In the Lagrangian modeling approach, air concentrations are computed by summing the contribution of each pollutant puff that is advected through the grid cell as represented by its trajectory. In a Lagrangian model, modeling the growth of the pollutant puff's 2nd moments or explicitly modeling the growth of a cluster of particles can simulate dispersion. Contrary to its acronym, HYSPLIT can simulate a pollutant distribution starting with a single particle or puff, or by following the dispersive motion of a large number of particles.

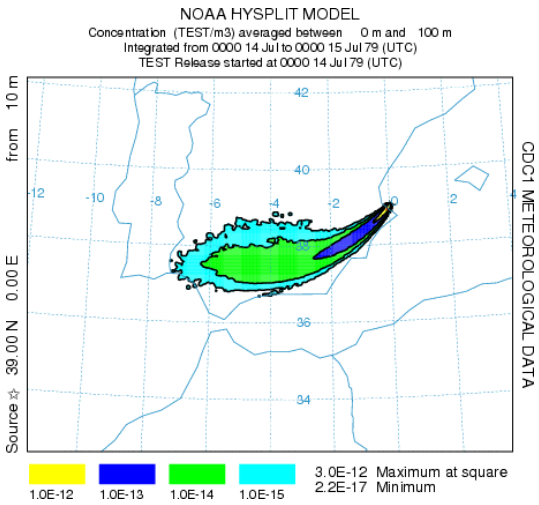
Previous Page | Next Page

Trajectories versus Concentration Plumes

A “puff” following a single trajectory cannot properly represent the growth of a pollutant cloud when the wind field varies in space and height. In these situations, the single-puff must either split into multiple-puffs or the simulation must be conducted using many pollutant particles. On the left, new trajectories are started every 4-h at 10, 100, and 200 m AGL to represent the boundary layer transport, while on the right, 2500 particles were used to simulate the air concentration plume.



control & setup.cfg



control & setup.cfg

Particle: The element (particle) is a point mass of contaminant. A fixed number of particles are released. They are moved by a wind having mean and random components. They never grow or split.

Puff: The element is a fully 3-D cylindrical puff, having a defined concentration distribution in the vertical and horizontal. Puffs grow horizontally and vertically according to the dispersion rules for puffs, and split if they become too large.

Hybrid: The element is a circular 2-D object (planar mass, having zero vertical depth), in which the horizontal contaminant has a “puff” distribution. There are a fixed number of these in the vertical because they function as particles in that dimension. In the horizontal dimension, they grow according to the dispersion rules for puffs, and split if they get too large.

Previous Page	Next Page

Code Installation Notes

Pre-installation Options

Tcl/Tk (8.3.2) is required to use the Graphical User Interface (GUI)
http://www.scriptics.com

Ghostscript (6.5) & Ghostview (3.6) to display and print the Postscript output
Installation should be to the \gs and \ghostgum directories.
http://www.cs.wisc.edu/~ghost

ImageMagick (5.3) to convert Postscript files to other graphical formats
Installation should be to the *c:\ImageMagick* directory.
http://www.imagemagick.org

Installation to different directories may require editing: *\guicode\hysplit4.tcl*.

HYSPLIT self-installing executable

setup47U.exe (25 Mb) – trial version, does not support forecast data
setup47R.exe (15 Mb) – complete version requires web site registration

Installed Directories

Arcview – information on ESRI shapefiles
Bdyfiles – surface height, land-use, and roughness length files
Browser – custom tcl scripts to support the GUI help browser interface
Concmdl – scripts and files to automate and customize concentration simulations
Csource – dll files required for the particle viewer & editor
data2arl – programs to convert meteorological data to the HYSPLIT format
document – most recent version of the technical documents and User's Guide
exec – all executables can be found in this directory
grads – source code to convert HYSPLIT output and meteorological data to grads
graphics – map backgrounds and map customization files
guicode – tcl scripts required to run the GUI
html – help files
metdata – sample meteorological data file and program to read the data
source – subroutines to compile the meteorological data conversion programs
trajmdl – sample scripts and files to customize trajectory simulations
vis5d – source code for program to convert concentration output to vis5d

Model Operation

Requirements

A trajectory or concentration simulation only requires one file called CONTROL, which defines various model parameters and other input and output files. An optional file called SETUP.CFG may be present to define more advanced simulation features. The only function of the GUI is to create these files and set any other command line options that some of the post-processing graphics programs may require.

Starting from the Tcl/Tk GUI

The desktop should contain the HYSPLIT shortcut with the following properties:

Target: \hysplit4\guicode\hysplit4.tcl
Start in: \hysplit4

The HYSPLIT "Start in" 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 "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

Starting from the command line

For instance to run the sample trajectory:

cd \hysplit4	- change to the "start in" directory
copy sample_traj control	- create the control file
..\exec\hymodelt	- runs the trajectory model
..\exec\trajplot\tdump	- creates the Postscript graphic
trajplot.ps	- should open Ghostview

For command line help on the trajectory program:

..\exec\trajplot	- without any command line options

To run the trajectory model from a customized control file:

..\exec\hymodelt traj	- opens file CONTROL.traj

Meteorological Data Requirements

The \exec directory contains several command line programs (chk_data, chk_rec) that can be used to analyze a HYSPLIT compatible meteorological data file. For example, a sample of the output from chk_file (the source code can be found in \metdata) is shown below for the NCEP global reanalysis file for July of 1979.

File characteristics and projection

Hysplit works only with meteorological data fields that have been projected on a conformal map projection (Polar Stereographic, Lambert, or Mercator) or a regular latitude-longitude grid. The data are organized as one record per variable per level. All records have the same record length. Records are written in a forward time sequence.

```
File start time:      79   7    1    0    0
File ending time:     79   7   31   18    0
Meteo data model   :  CDC1
Grid size x,y,z    :   144   73   18
Records per time   :  94
Minutes between    :  360
```

Variables

A unique 4-character string identifies meteorological variables. The minimum requirements to run the model are the U and V wind components (UWND, VWND), ambient temperature (TEMP), height (HGTS) of the data level (if on pressure coordinates), and the surface pressure (PRSS).

```
Index   Level   #      Variable listing and checksum value
18      10.     4      HGTS 53   TEMP 194  UWND 68   VWND 45
13      100.    5      HGTS 224  TEMP 111  UWND 43   VWND 59   WWND 12
9       300.    6      HGTS 252  TEMP 164  UWND 67   VWND 53   WWND 128  RELH 42
2       1000.   6      HGTS 121  TEMP 191  UWND 67   VWND 29   WWND 114  RELH 250
1       0.0     5      PRSS 235  T02M 245  U10M 185  V10M 93   TPP6 99
```

Record listing

Each data record is composed of a 50 byte header portion, describing the data packing, followed by the packed data of length (I*J bytes). A one-byte per element "Difference packing" is used for all data fields. The first data record (INDX) of each time period contains information on the variables, levels, grid, and checksums.

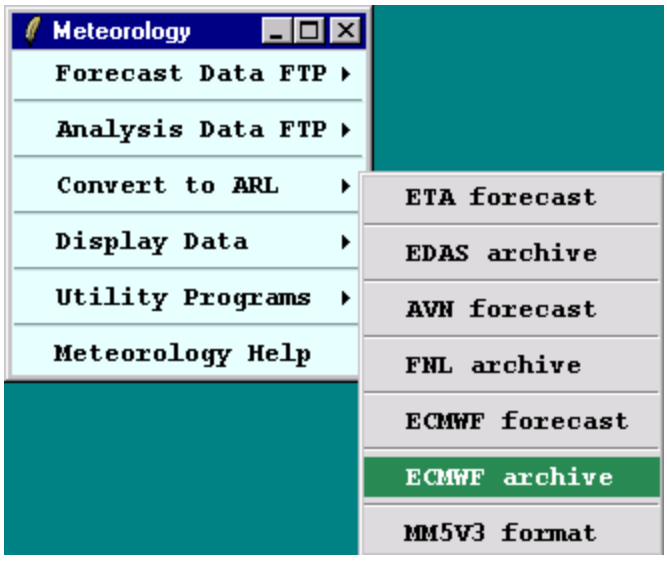
```
 1  79 7 1 0 0 099INDX   0  .0000000E+00  .0000000E+00
 7  79 7 1 0 0 199HGTS   7  .5039370E+00  .2240000E+03
 8  79 7 1 0 0 199TEMP   4  .6299213E-01  .2236000E+03
 9  79 7 1 0 0 199UWND   4  .6299213E-01 -.2799992E+01
10  79 7 1 0 0 199VWND   3  .3149606E-01 -.1089999E+02
11  79 7 1 0 0 199WWND  -8  .1537894E-04  .1749980E-02
12  79 7 1 0 0 199RELH   7  .5039370E+00  .1000000E+03
```

GRIB Data Conversions

Many meteorological centers archive their model output fields in the **GRI**dded **B**inary format (GRIB). The GRIB standard defines the contents of a data record. A GRIB record consists of six sections: 0) An indicator section which always starts with the four ASCII characters "GRIB", 1) a product definition section, 2) a grid description section, 3) the bit map section, 4) the binary data section, 5) and the record terminator which always consists of the ASCII characters "7777". More detailed information about GRIB and the new GRIB2 standard can be found at the WMO. GRIB does not define the record contents of file. Each center may package their data differently. Therefore rather than designing HYSPLIT to work directly with GRIB files, an intermediate program is required to convert GRIB data to the HYSPLIT compatible format.

Converting from GRIB to HYSPLIT

The HYSPLIT GUI contains FTP links to HYSPLIT compatible archives. Some GRIB conversion programs (for NOAA and ECMWF data) are integrated into the GUI. Programs for other meteorological formats (e.g. RAMS) are available from the HYSPLIT data web page. The "Convert to ARL" menu tab is designed to convert data already on the local computer. Conversions may are also performed in conjunction with the analysis and forecast data FTP menus. These menus will be discussed in more detail in subsequent sections. In the example on the next page, reanalysis data from the ECMWF public archive will be converted to the HYSPLIT compatible format.



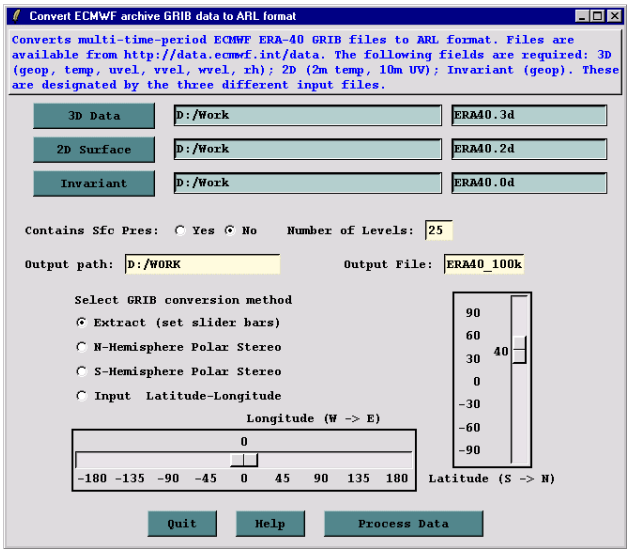
Previous Page | Next Page

ECMWF ERA-40 Archive

A direct link to the ECMWF data server is not included in the GUI. Access to the data archive requires acceptance of their copyright agreement. Once through this step, three data files should be obtained for the data period of interest. All fields should be four times-per-day (0,6,12,18). At a minimum, select geo-potential, temperature, u- and v-velocity, vertical velocity, and relative humidity, at all pressure levels within the lower troposphere. The surface variables should include the 2-m temperature, 10-m u- and v-velocity. Because surface pressure is not available (only mean sea-level), the geo-potential invariant field (terrain height) must also be retrieved.

Converting from GRIB to HYSPLIT

Open the ECMWF archive conversion menu from the GUI and set the file names for the three GRIB data sets. Define the name of the output data file if something other than the default name is required. In this example, the data will be interpolated to a 100x100 100-km resolution conformal map projection centered about the point indicated by the slider bars. The last option under the GRIB conversion does no interpolation and outputs the data on the same grid as input, but in a HYSPLIT compatible format.



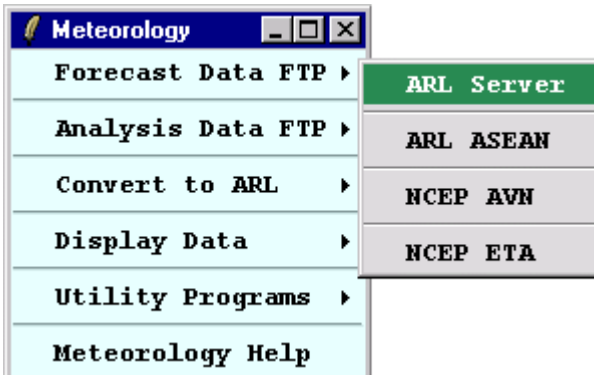
The ECMWF menu calls the generic grib2arl converter program. This program has several additional options that can be set through the command line that are not available through the GUI. This program can handle all latitude-longitude NOAA as well as ECMWF GRIB files containing single or multiple time periods. For example the –n option can be used to change the size of the interpolated grid to something other than the 100x100 default.

```
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]
```


Previous Page	Next Page

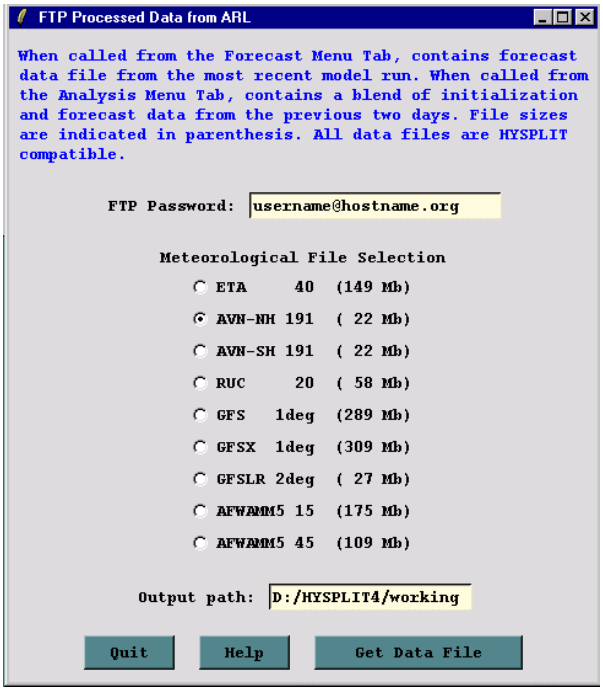
Forecast Data FTP Access

There are four options under the "Forecast Data FTP" menu tab, all of which access NOAA computer systems. The "ARL" options access the Air Resources Laboratories data server and the "NCEP" options access the National Center's for Environmental Prediction' data server. All data available from the ARL server have already been converted to a HYSPLIT compatible format. The NCEP menu's access GRIB data and these data are converted during the download process. Note that ARL forecast data consist of one file – the most recent forecast available, while the NCEP data consist of one GRIB file per forecast time. Each one is processed in sequence. The last several forecast cycles are usually available for download.



The ARL Server

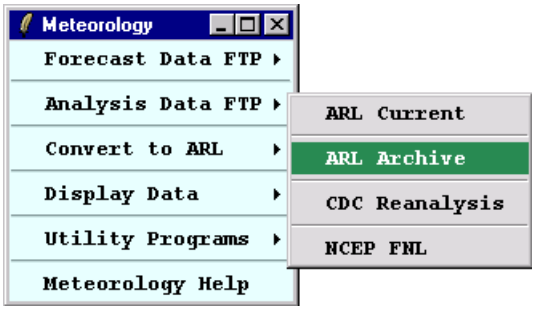
The data on the ARL server covers for 3 geographic domains. The ETA, RUC (Rapid Update Cycle), and AFWA (Air Force Weather) cover North America. The AVN (from the GFS - global forecast system) covers the northern and southern hemispheres. The GFS global output GFS is available on a latitude-longitude grid at three different temporal and spatial resolutions: (GFS to +4 d, GFSx to +8 d, and GFSLR to +12 d). Prior to selecting "Get Data File", a valid Email address should be entered into the password field. The menu system will "lock-up" until the FTP has completed. The only way to break out would be to kill the FTP process through the windows task manager.



Previous Page | Next Page

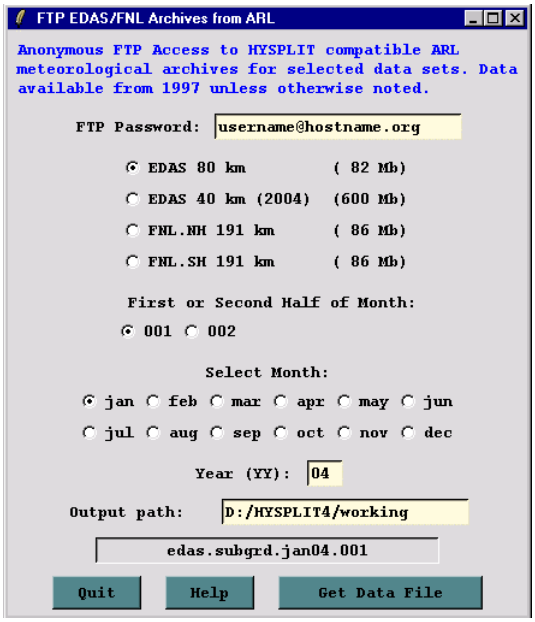
Analysis Data FTP Access

There are four options under the "Analysis Data FTP" menu tab, all of which access NOAA computer systems. The "ARL" and "CDC" options access the Air Resources Laboratories data server and the "NCEP" options access the National Center's for Environmental Prediction' data server. All data available from the ARL server have already been converted to a HYSPLIT compatible format. The NCEP menu accesses the GDAS final analysis GRIB data. These files are converted during the download process. Note that ARL analysis data always consist of one file with multiple time periods, while the NCEP data consist of one GRIB file per time period. Each NCEP file is processed in sequence to create one multi-time period file. NCEP files are only available for the last 24 hours. The "current" analysis consists of a time series of initialization files from the ETA or AVN forecasts for the last 48 hours. The "archive" menu is discussed in more detail below. The "reanalysis" menu provides access to monthly 2.5 degree NCAR/NCEP files from 1948 to present.



The ARL Archive

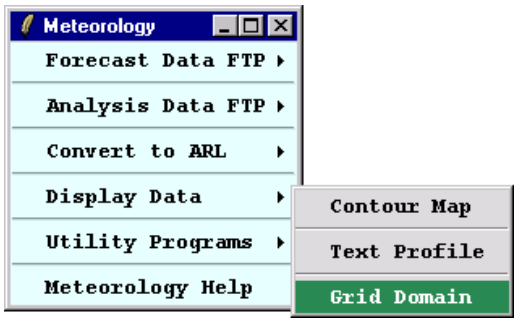
The data on the ARL server covers 3 geographic domains: the EDAS for North America and the GFS final analysis (FNL) for the northern and southern hemispheres. Starting in 2004, the EDAS is available only at the enhanced spatial resolution. Files are selected according the to year and half-month (001 for days 1 to 15 and 002 for days 16 to the end). Prior to selecting "Get Data File", a valid Email address should be entered into the password field. The menu system will "lock-up" until the FTP has completed. The only way to break out would be to kill the FTP process through the windows task manager.

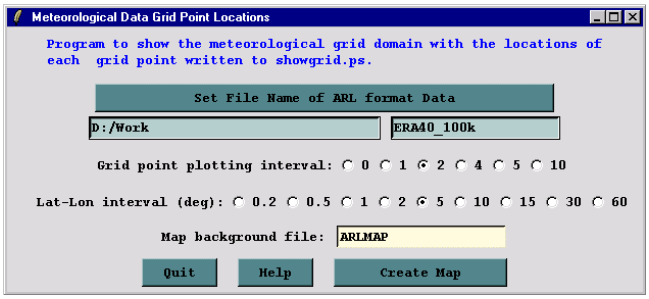


Previous Page | Next Page

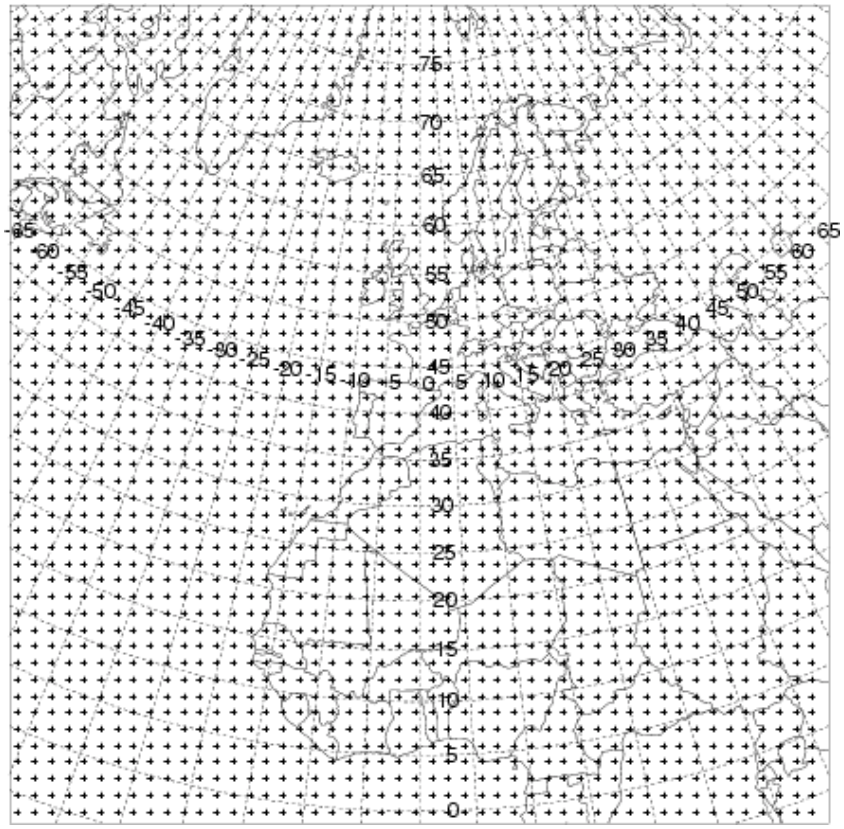
Display Meteorological Grid Domain

There are three options under the “Display Data” menu tab: 1) contouring the data fields, 2) a text listing of the profile of all meteorological variables at a selected point, and 3) the spatial domain of the grid.





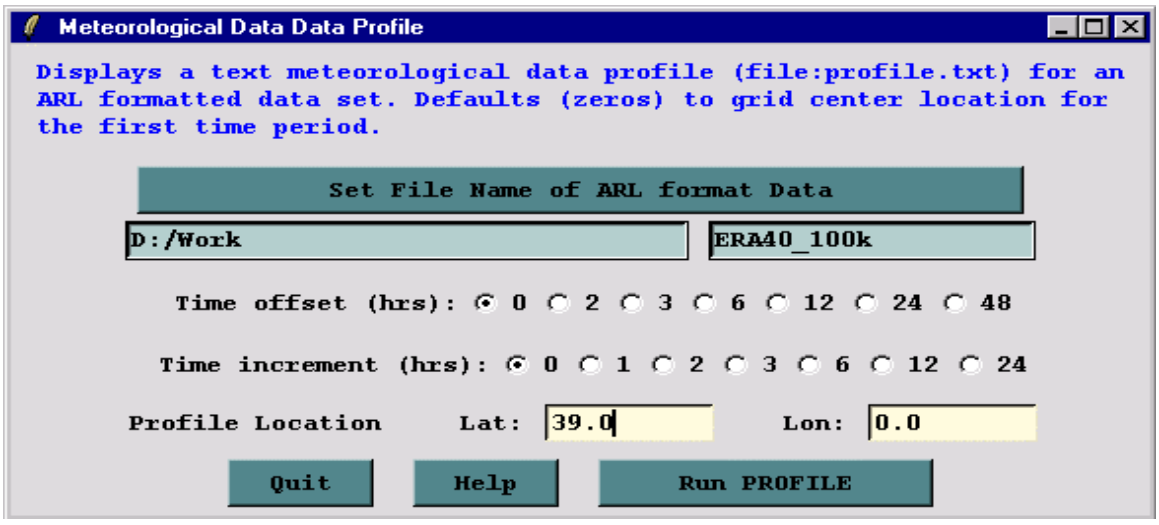
The meteorological domain can be displayed by selecting a file, in this case the 100-km conformal projection that was created from the ECMWF ERA-40 archive in a previous step. Every other grid point (select = 2) will be displayed and latitude-longitude lines will be drawn every 5 degrees.



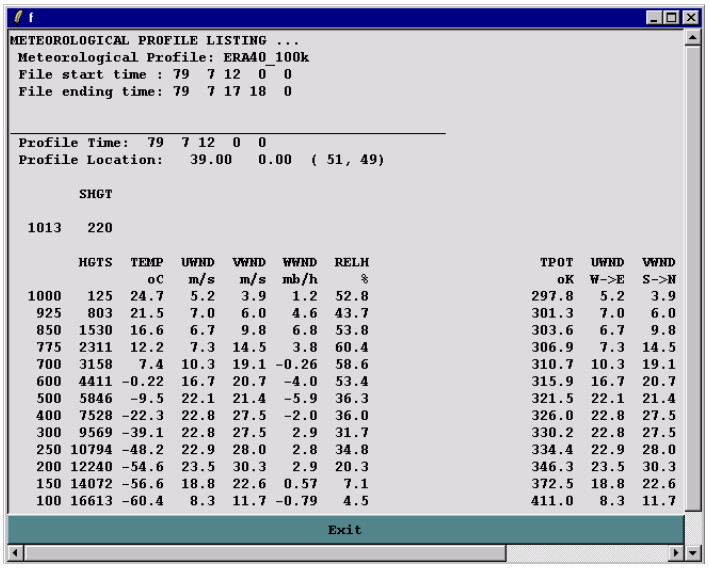
Previous Page	Next Page

Vertical Meteorological Profile

The profile utility program creates a simple text based listing of the meteorological data profile at a selected latitude-longitude point. In the example below, the ECMWF ERA-40 data set was selected with default values for offset and increment. Zeros indicated that only the first time period is displayed. The profile location was chosen at 39N on the Greenwich meridian.



The data are shown for the nearest grid location, with no temporal or spatial interpolation. The location grid index is indicated in parenthesis next to the position. The first row shows the surface variables. Subsequent rows show the upper-level data, in this case by pressure level. The leftmost columns show the data directly from the file, while on the right side ambient temperatures has been converted to potential temperature and wind components have been rotated from the native grid to true compass direction. In this case they are almost identical because the location chosen was near the center of the grid. The leftmost coordinate is pressure as taken from the index record. In other coordinate systems the program computes a height.

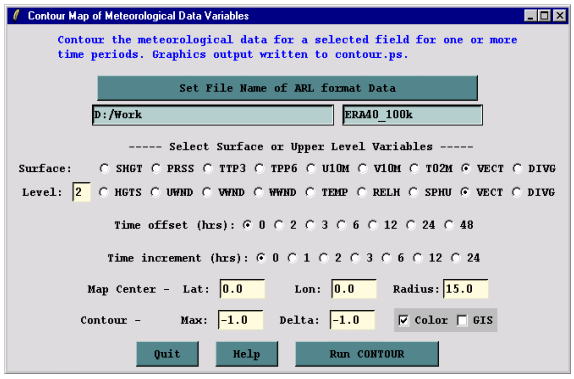


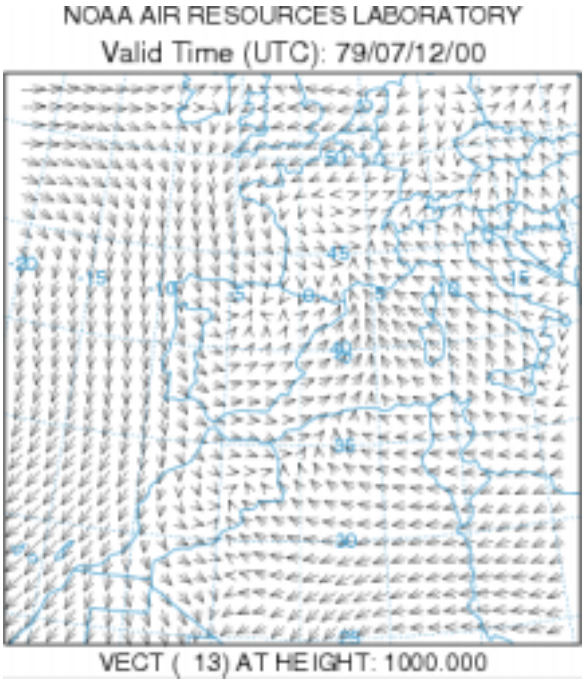
```
METEOROLOGICAL PROFILE LISTING ...
 Meteorological Profile: ERA40_100k
 File start time : 79  7 12  0  0
 File ending time: 79  7 17 18  0

 Profile Time:  79  7 12  0  0
 Profile Location:   39.00   0.00  ( 51, 49)

       SHGT

 1013   220

       HGTS  TEMP  UWND  VWND  WWND  RELH                    TPOT  UWND  VWND
              oC   m/s   m/s  mb/h     %                      oK  W->E  S->N
 1000   125  24.7   5.2   3.9   1.2  52.8                  297.8   5.2   3.9
  925   803  21.5   7.0   6.0   4.6  43.7                  301.3   7.0   6.0
  850  1530  16.6   6.7   9.8   6.8  53.8                  303.6   6.7   9.8
  775  2311  12.2   7.3  14.5   3.8  60.4                  306.9   7.3  14.5
  700  3158   7.4  10.3  19.1 -0.26  58.6                  310.7  10.3  19.1
  600  4411 -0.22  16.7  20.7  -4.0  53.4                  315.9  16.7  20.7
  500  5846  -9.5  22.1  21.4  -5.9  36.3                  321.5  22.1  21.4
  400  7528 -22.3  22.8  27.5  -2.0  36.0                  326.0  22.8  27.5
  300  9569 -39.1  22.8  27.5   2.9  31.7                  330.2  22.8  27.5
  250 10794 -48.2  22.9  28.0   2.8  34.8                  334.4  22.9  28.0
  200 12240 -54.6  23.5  30.3   2.9  20.3                  346.3  23.5  30.3
  150 14072 -56.6  18.8  22.6  0.57   7.1                  372.5  18.8  22.6
  100 16613 -60.4   8.3  11.7 -0.79   4.5                  411.0   8.3  11.7
                                Exit
```


Previous Page | Next Page

Contour Meteorological Data

The contour utility program creates a Postscript graphic of a variable. In the example below, the ECMWF ERA-40 data set was chosen with default values for offset and increment. Zeros indicated that only the first time period is displayed. A map center location of zeros sets the map to default to the center of the data grid. Negative values for the contour maximum and delta forces automatic contour scaling. All possible meteorological variables are not shown, nor may a data file contain all the variables in the selection list. More options are available from the command line version of the program called “display.exe”. Wind velocity vectors will be plotted at data level 2, which from the previous example (profile program) we know corresponds to the 1000 hPA surface. The velocity vectors are shown at every grid point over the domain selected for display. In this case a map with a radius of 15 degrees latitude. The “13” in parenthesis after the variable symbol “VECT” indicates that the maximum wind speed vector on the map was 13 m/s.





Previous Page	Next Page

Trajectory Computational Method

If we assume that a particle passively follows the wind, then its trajectory is just the integration of the particle position vector in space and time. The final position is computed from the average velocity at the initial position (P) and first-guess position (P').

$$P(t+\Delta t) = P(t) + 0.5\,[\,V(P,t) + V(P',t+\Delta t)\,]\,\Delta t$$
$$P'(t+\Delta t) = P(t) + V(P,t)\,\Delta t$$

The integration time step is variable:

$$V_{max}\,\Delta t < 0.75$$

The meteorological data remain on its native horizontal coordinate system. However, the meteorological data are interpolated to an internal terrain-following (σ) vertical coordinate system:

$$\sigma = (Z_{top} - Z_{msl}) / (Z_{top} - Z_{gl})$$

Z_{top} - top of the trajectory model's coordinate system
Z_{gl} - height of the ground level
Z_{msl} - height of the internal coordinate

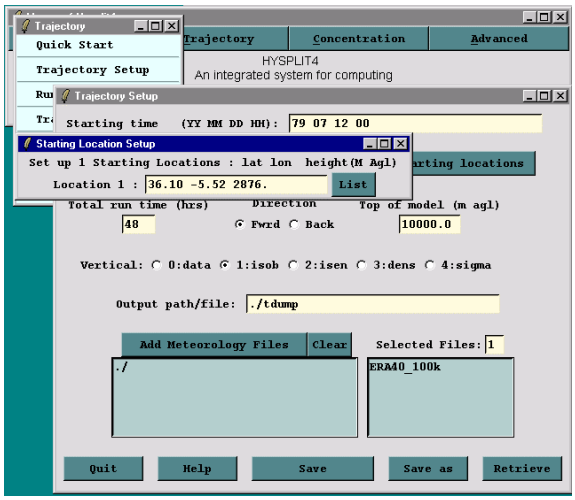
The model's internal heights can be chosen at any interval, however a quadratic relationship between height and model level is specified, such that each level's height with respect to the model's internal index, k, is defined by

$$Z_{agl} = ak^2 + bk + c$$

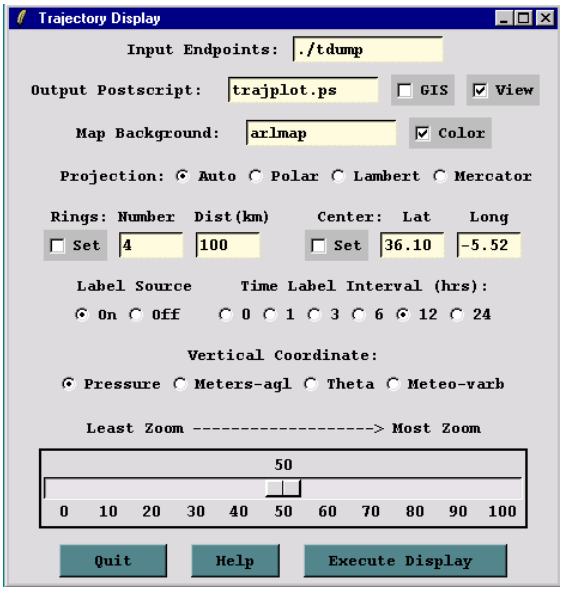
The constants are automatically defined such that the model's internal resolution has the same or better vertical resolution than the input data.

Trajectory Example Calculation

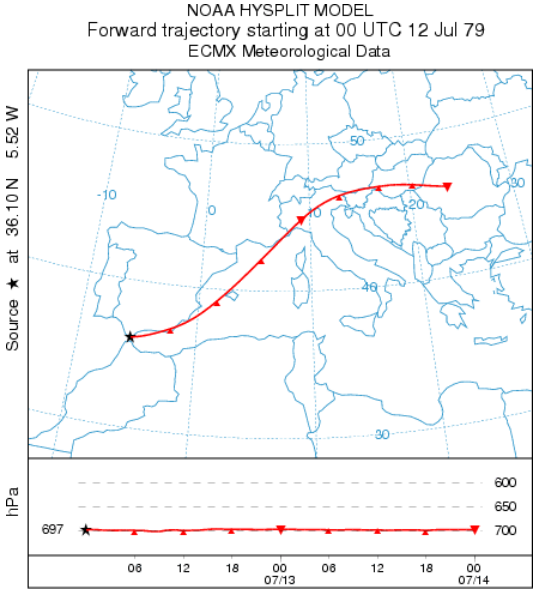
A pollutant particle's transport can be illustrated by a trajectory calculation. For this example select the southern most point in Spain, set the height for about 2876 m AGL, the duration to 48 hours, and the vertical motion method should be isobaric. In this way we can compare the trajectory result to the 700 hPa height fields. When properly configured, the GUI menu should be similar to that shown below on the left. Then press "Save" to close the menu and then "Run Standard Model". After the run has completed, press "Trajectory Display," select any special options, and then "Execute Display."



CONTROL file



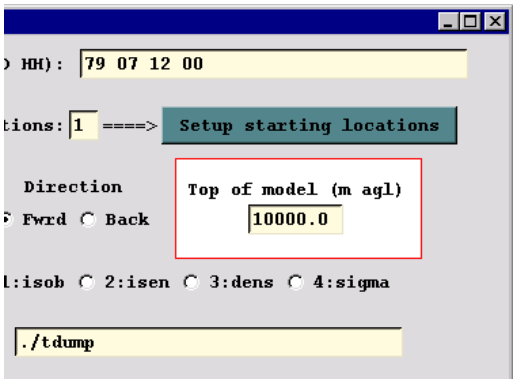
The relationship of the trajectory to the temporal and spatial variations of the 700 hPA height field is illustrated in the attached animation which was created using only the standard tools that come with HYSPLIT. These procedures will be discussed in more detail later. In addition, there are several tasks that should be completed in this section to become familiar with more of the program options: 1) run this case from the command line, 2) draw concentric distance circles on the map, and 3) force the map center to a different location.

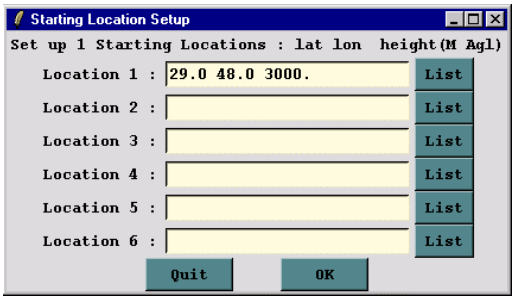


Previous Page | Next Page

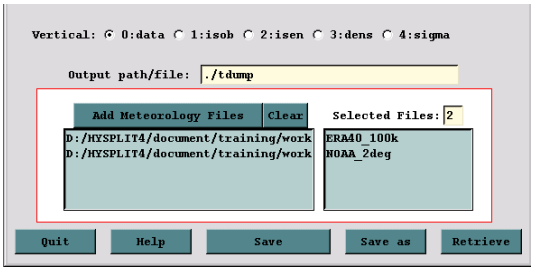
Trajectory Model Configuration

The setup menu has several options that control the computation. Usually they should be left at their default values. For instance, computations should use the vertical motion field (Data) contained within the data file. Only under special situations, such as the previous example, should other methods be selected. Some of these options will be explored in more detail later. The "top of the model" is the height above which the meteorological data are not processed. For calculations within the troposphere, 10 km is a good top. Trajectories are terminated when they reach this height. Processing fewer levels reduces computational times. A starting location can be entered directly from the "starting locations" menu or a position may be chosen from a predefined "list". This list is user editable "plants.txt" file. For this example, select a height of 3 km near Kuwait.





One key feature is for any simulation is selecting the best meteorological data files. In the current compiled version up to 12 files may be defined simultaneously. When multiple files are defined, at each integration step, the model finds the finest spatial resolution file at the location and time of the trajectory end-point. Execution of the control file for this case results in a trajectory that goes south.



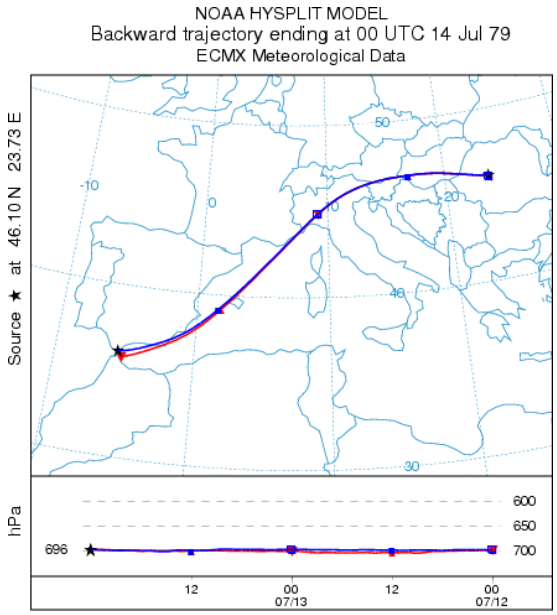
The meteorological file indicator is written with each end-point position in the second column of the ASCII trajectory output file. The diagnostic MESSAGE file also provides additional detail about the calculation. In this example the switch from ECMX to CDC1 occurs at 1500 GMT, causing the 1200 and 1800 GMT data to be reloaded.

Previous Page	Next Page

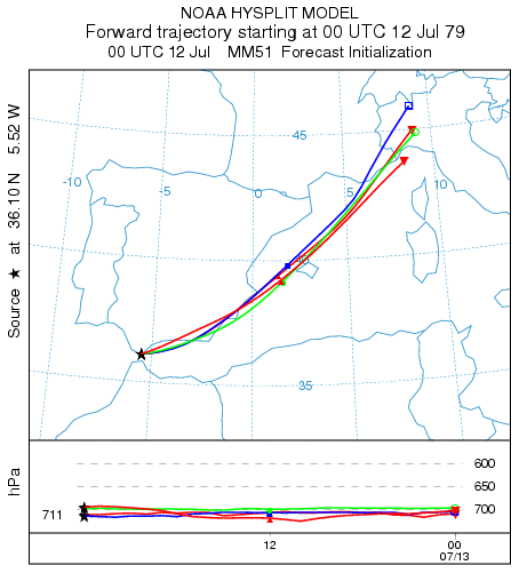
Trajectory Error

A trajectory calculation is composed of three error components: the computational error due to numerical inaccuracies, the inadequacy of the data's representation of the atmosphere, and measurement errors in creating the model's meteorological data fields.

The numerical accuracy of the computation can be estimated by running a forward and backward trajectory to the origin point. Run the previous 700 hPA isobaric example. View the endpoints file, and use the final position (46.104N, 23.727E, 2694.7 m AGL) as the starting point (on the 14th 0000 UTC) for a backward trajectory calculation. Insure that the endpoints file names are different for both the forward and backward calculations. Then display both trajectories on the same plot by enter both file names using a + symbol (e.g. tdump1+tdump2). Note how the return trajectory is very close to the initial origin point.



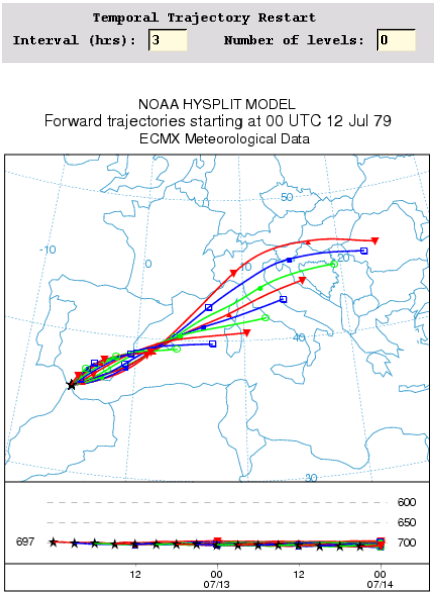
A greater source of error is due to the difficulty in representing atmospheric variables, which are continuous in space and time, by discrete data points on a grid. This error is difficult to quantify, but a sense of the error can be determined by running trajectories using meteorological data from several different sources. In the adjacent calculation, trajectories have been computed using meteorological data from ECMWF, NOAA, and MM5 (108 km and 36-km resolution). Within the first 24-h, differences between trajectories is much greater than the numerical error. These calculations are more consistent than most simulations due to the isobaric assumption.



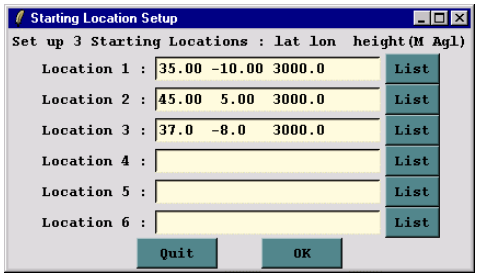
Previous Page	Next Page

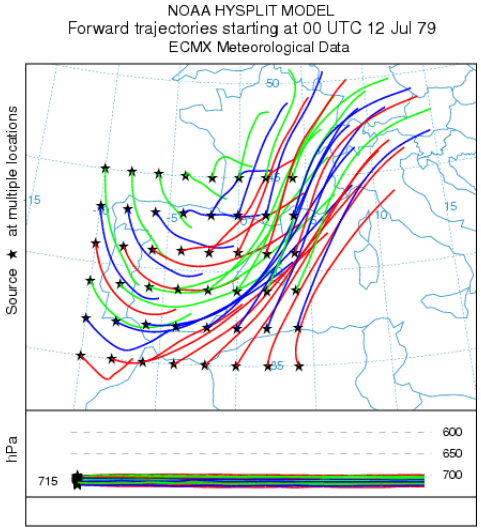
Multiple Trajectories

Normally trajectories are started only at the initial time at the locations specified. Using the "advanced, trajectory, configuration" menu tab, trajectories can also be started at regular time intervals from the same location by setting the restart interval (e.g. to 3-h). Leave the other parameters the same as in the original isobaric trajectory. The resulting trajectory graphic shows new trajectories every 3-h, terminating at the end of the 48-h computational period. Trajectories starting later will have a shorter duration. All trajectories can be set to have the same duration from the "advanced" menu tab. The multiple-level option was shown earlier.



The GUI can be used to configure only 6 starting locations. The model can support an unlimited number of locations. The GUI does have a shortcut method to configure a regular matrix of locations by defining three points, representing the lower left, upper right, and location increment. Once configured, the "run matrix" option is selected through the "special simulations" menu tab. This causes the control file to be rewritten externally from the GUI to the number of locations defined by the grid (in this case 48). The graphic shows the matrix of 24-h duration isobaric trajectories (labels off). In this case, because height varies across the matrix, the MSL option was selected from the "advanced" menu.





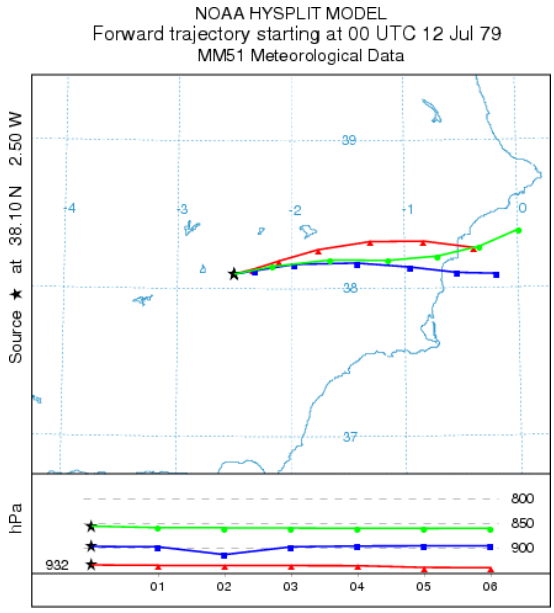
Previous Page | Next Page

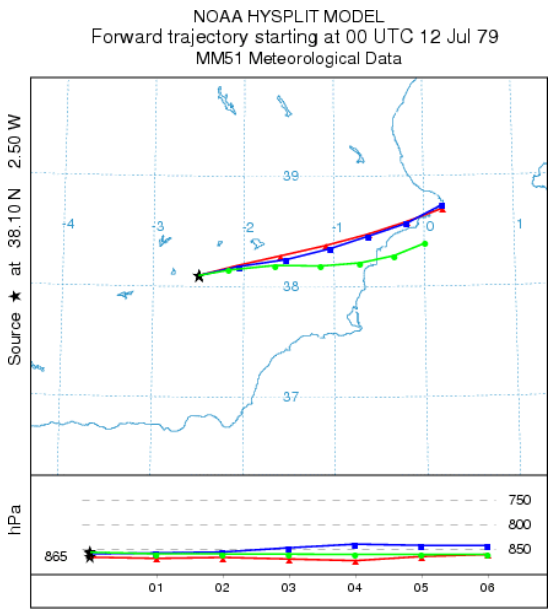
Terrain Height

The trajectory starting height is defined as AGL (above ground level). Height definitions can be changed to MSL (mean sea level) from the "advanced, configuration" menu tab. Regardless of how the input heights are defined, internally HYSPLIT treats all heights in a terrain following coordinate system. The terrain values are those of the meteorological data. These may be quite different from the actual terrain height at a point of interest. As an example of how one might define a starting trajectory, examine the location 38.1N and 2.5W. Using the profile program, the terrain heights at that location for several of the different data sets can be determined.

Model	Resolution	Terrain
MM5	12 km	1450 m
MM5	36 km	1081 m
MM5	108 km	816 m
ECMWF	2.5 deg	364 m
NOAA	2.5 deg	400 m

Assume the 12-km data to be the base case (the most accurate), then the isobaric trajectories (to minimize vertical motion effects) from 10 m AGL for each MM5 data sets is shown to below left. Although the data have the same vertical resolution, and the trajectories start out at the same height AGL, they start at different pressure levels due to differences in elevation between the data. The 36-km (blue) and 12-km (green) trajectories are the most similar. In the second example, the initial heights of the 36-km and 108-km (red) simulations were adjusted so that the trajectories start at the same pressure levels and neither one matched the 2-km trajectory. The proper trajectory depends if the interest is related to interactions with the surface or longer-range transport.

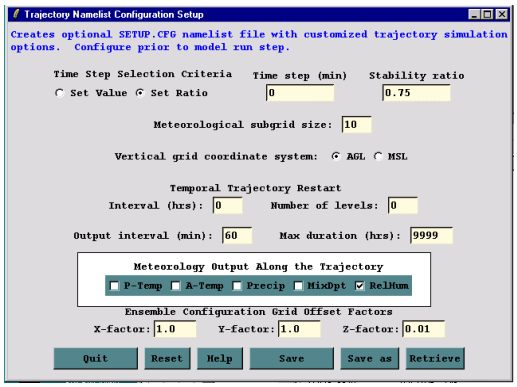




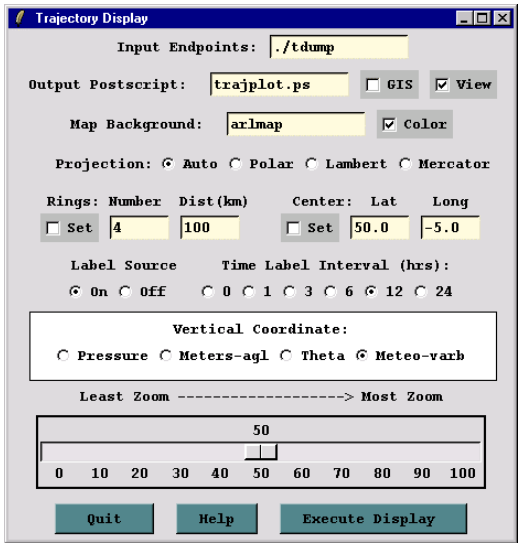
Previous Page | Next Page

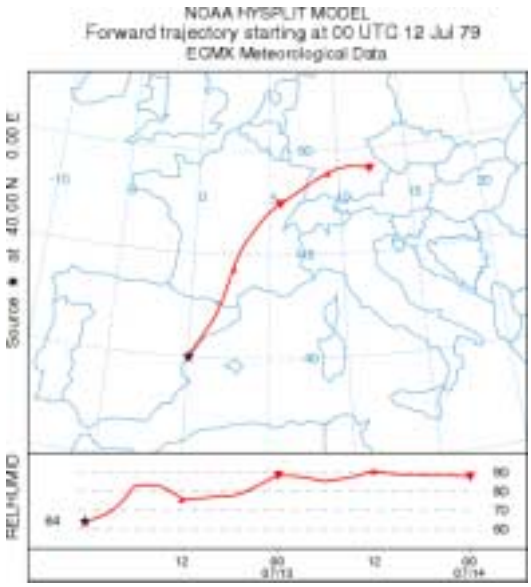
Meteorological Variations on a Trajectory

Configure the control file for a trajectory simulation from 40 N and 0 W from 1500 m using the ECMWF data and the vertical motion data field. The resulting trajectory goes to the northeast. If the value of the selected meteorological variables is desired along that trajectory, a flag in the configuration file can be set through the "advanced" menu tab. Ambient and potential temperature, precipitation, mixing depth, and relative humidity are currently the only available options. One or more fields may be selected, all will be written to the output file, but only the last value can be plotted.



Upon completion of the calculation, the vertical coordinate used to display the trajectory can be selected from the display menu. Pressure and height are available for all trajectories, potential temperature (theta) is only available for isentropic trajectories, and only the last selected meteorological variable can be plotted.



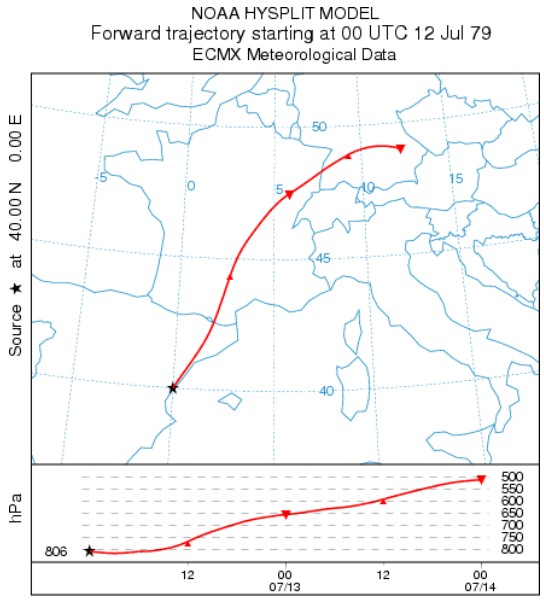


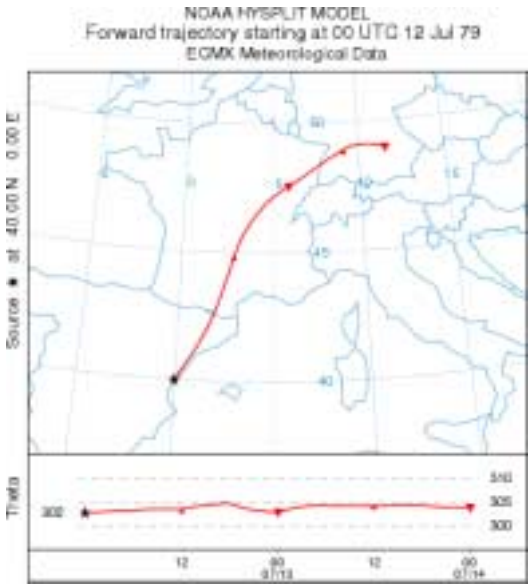
Previous Page | Next Page

Vertical Motion Options

There are five different vertical motion options. The suggested default is to use the vertical velocity field that is included with most meteorological data. Other options may be required for special situations such as following the transport of a balloon on a constant density surface, comparing isobaric flow fields between data sets, or situations when the meteorological data's vertical velocity field may be too noisy compared with the time step at which the data are available (high spatial resolution simulations).

In the sigma option the trajectory remains on its original sigma surface. In the isobaric, isentropic, and constant density (isopycnic) options, the vertical velocities are computed from the equation, $W = (-\partial q/\partial t - u\,\partial q/\partial x - v\,\partial q/\partial y) / (\partial q/\partial z)$, where "W" is the velocity required for the trajectory to remain on the "q" surface (pressure, potential temperature, density). Note that the equation results in only an approximation of the motion and a trajectory may drift from the desired surface. Shown below left is the same trajectory of the previous example using the ECMWF vertical velocity fields. To the right is the same trajectory computed using the isentropic flow assumption, showing that the potential temperature varied by only about 1 degree. This illustrates that under adiabatic flow conditions the two trajectory methods should give comparable results.





Previous Page | Next Page

Modeling Particle Motion or Particle Distributions (Puffs)

To compute air concentrations it is necessary to follow all the particles needed to represent the pollutant distribution in space and time. This can be done explicitly by following the trajectory of each particle, where a random component is added to the mean velocity (from the meteorological model), to define the dispersion of the pollutant cloud. In the horizontal, the computations can be represented by the following equations:

$X(t+\Delta t)=X_{mean}(t+\Delta t)+U'(t+\Delta t)\,\Delta t$,
$U'(t+\Delta t)=R(\Delta t)\,U'(t)+U''\,(\,1-R(\Delta t)^2\,)^{0.5}$,
$R(\Delta t)=\exp(-\Delta t/T_{Lx})$,
$U''=\sigma_u\,\lambda$,

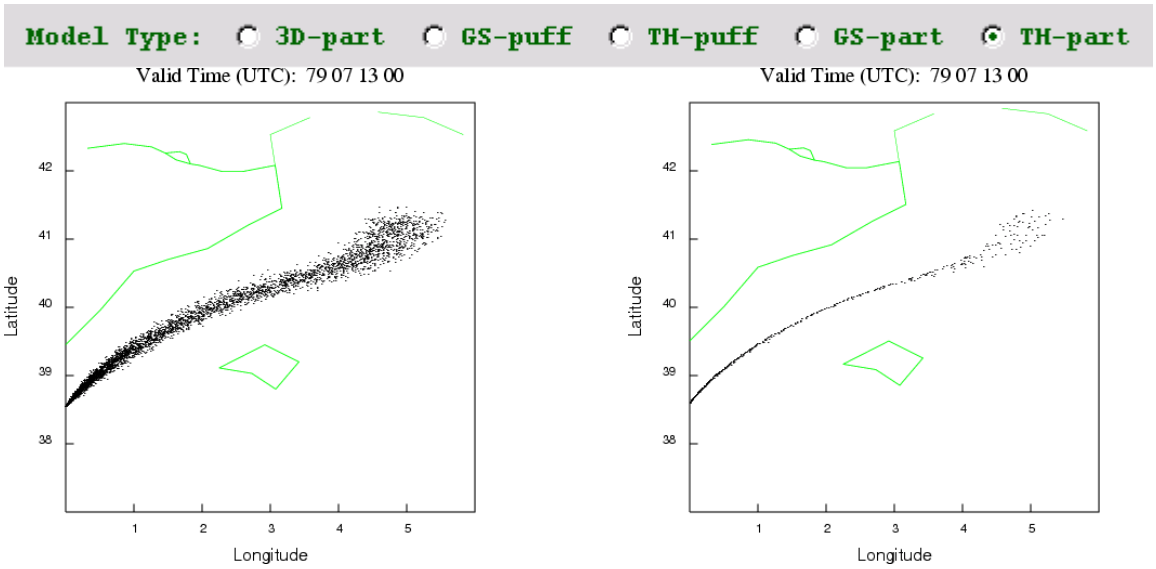
where λ is a random number with 0 mean and σ of 1. The computations can be simplified, if instead of modeling the motion of each particle, we compute the trajectory of the mean particle position and the particle distribution. The standard deviation of the particle distribution could be computed from all the particles,

$$\sigma^2 = \overline{(X_i-X_m)^2}$$

or it can be computed without following individual particles by assuming a distribution shape (puff) and relationship to the local turbulence. Many different formulations can be found in the literature.

$$d\sigma_h/dt = \sqrt{2}\,\sigma_u$$
$$\sigma_u = (K_x / T_L)^{0.5}$$

These computations are set in the "advanced configuration" menu.



3-D Particle distribution

Top-Hat Puff center positions

Previous Page | Next Page

Concentration Prediction Equations

The previous example showed a snapshot of the particle or puff center positions after 24 hours. Air concentrations are computed by summing each particle's mass as it passes over the concentration grid. In the particle model mode, the concentration grid is treated as a matrix of cells, each with a volume defined by the grid dimensions. Therefore the concentration is just the particle mass divided by the cell volume.

3D particle: $\Delta c = q\ (\Delta x\ \Delta y\ \Delta z)^{-1}$

Top-Hat: $\Delta c = q\ (\pi\ r^2\ \Delta z)^{-1}$

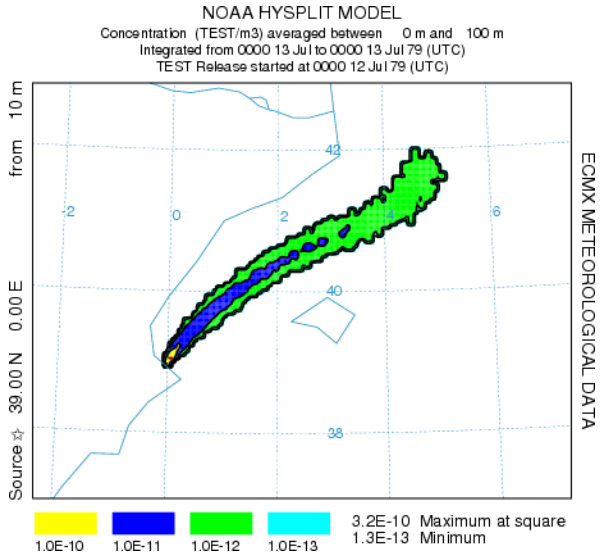
Gaussian: $\Delta c = q\ (2\ \pi\ \sigma_h^2\ \Delta z)^{-1}\ e^{-0.5\ x2/\sigma h2}$

In the puff calculation, the concentration grid is considered as a matrix of sampling points, such that the puff only contributes to the concentration as it passes over the sampling point. In the puff calculation mode it is possible for a puff to pass between points and not be shown on the display.

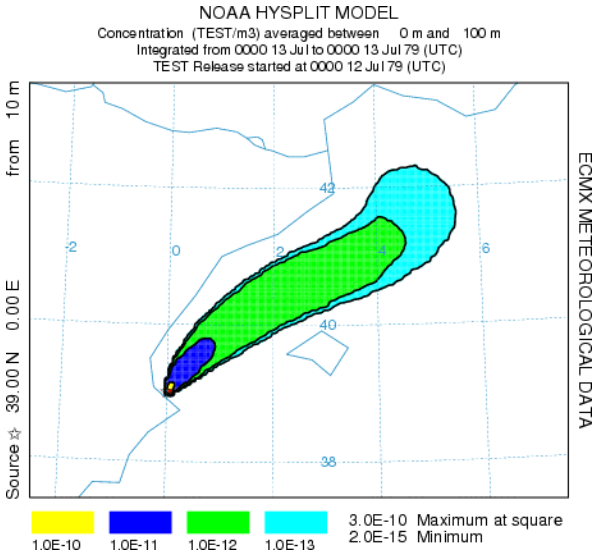
Top-Hat: $\Delta c = q\ (\pi\ r^2\ \Delta z_p)^{-1}$

Gaussian: $\Delta c = q\ (2\ \pi\ \sigma_h^2\ \Delta z_p)^{-1}\ e^{-0.5\ x2/\sigma h2}$

The concentration patterns associated with the particle and puff distributions are shown below. Note that the puff distribution is smoother but much broader. In this particular case, the horizontal puff growth equations give larger values than the particle expansion.



Particle Concentrations



Top-Hat Puff Concentrations

Previous Page	Next Page

Turbulence Equations

Dispersion: ◉ Standard ○ Short Range ○ Input TKE ○ Variance

The method by which the meteorological data are evaluated to determine the turbulent velocities, used in either the puff or particle computation is set in the advanced configuration menu. The default method is called "standard" and is defined by a similarity approach for vertical mixing and velocity deformation for horizontal mixing.

$$K_z = k\, w_h\, z\, (1 - z/Z_i)$$
$$K_h = 2^{-0.5}\, (c\, \Delta)^2\, |\, \partial u/\partial y + \partial v/\partial x\, |$$

Dispersion: ○ Standard ◉ Short Range ○ Input TKE ○ Variance

The turbulent velocity variance can also be obtained directly from the stability functions instead of through the intermediate step of computing a diffusion coefficient. This method is called "Short Range" and the boundary layer velocity variances are defined as a functions of u*, w*, and Z_i. This method does not use the diffusivity and no assumptions are required about turbulent scales. For instance, in the stable/neutral BL:

$$w'^2 = 3.0\, u_*^2\, (1 - z/z_i)^{3/2}$$
$$u'^2 = 4.0\, u_*^2\, (1 - z/z_i)^{3/2}$$
$$v'^2 = 4.5\, u_*^2\, (1 - z/z_i)^{3/2}$$

Dispersion: ○ Standard ○ Short Range ◉ Input TKE ○ Variance

If the TKE field is available, then the velocity variances can be computed from its definition and the previous velocity variance equations to yield relationships with TKE.

$$E = 0.5\, (u'^2 + v'^2 + w'^2)$$
$$w'^2 = 0.52\, E,\ u'^2 = 0.70\, E,\ v'^2 = 0.78\, E$$
$$u'^2 = v'^2 = 0.36\, w_*^2$$

Dispersion: ○ Standard ○ Short Range ○ Input TKE ◉ Variance

Some meteorological data sets may already contain the component turbulent velocity variances. This would normally be the case for data that have been generated from local measurement programs.

Dispersion Model Configuration

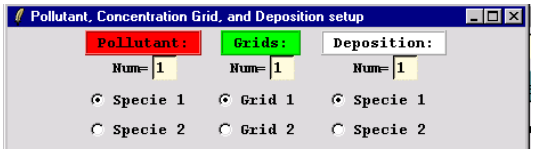
The control file for dispersion simulations is configured from the "concentration setup" menu tab. The concentration setup layout is identical to the trajectory menu with the exception of an additional button to set the emissions, deposition, and concentration grid.

This button will bring up a submenu with each of the three main options. The pollutant emission rate and deposition

A unique 4-character field identifies each pollutant. The emission rate is mass units per hour. The actual mass unit is not specified, so for instance, if the units are kg, then concentration will be kg/m^3. Any unit is acceptable, however some chemical conversion modules require specific units. The emission start time can be set to any time after the start of the simulation. As is true for all time units, zero's default to the simulation start time in the main menu. Zero for the month and non-zero

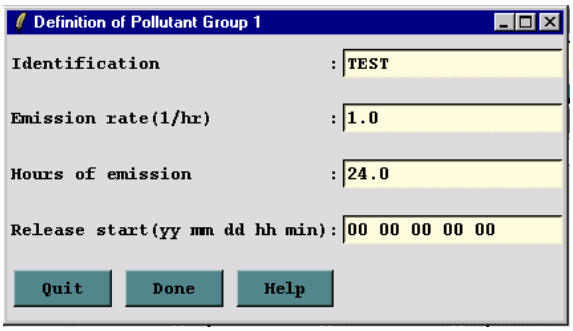
Each concentration grid must be defined. Zeros for the grid center default to the source location. The grid spacing is especially important in concentration computations in determining the cell size (particles) or sampling resolution (puffs). When multiple levels are defined, each height represents the top of the cell (particles) or actual height (puffs). The averaging time starts at the sampling start time. Snapshot concentrations are the average over one time step at the time interval specified.

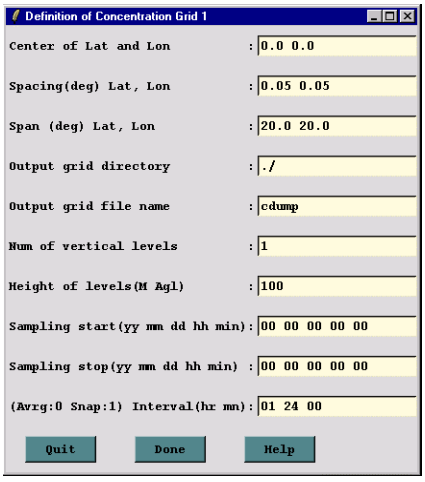
must be set for each pollutant. Normally simulations are run for one pollutant.



Several independent concentration grids may be defined for each simulation. However, they may be nested in space or time if desired. Grids are automatically defined for each pollutant species.

values for day and hour cause those values to be treated as relative to the simulation start time.

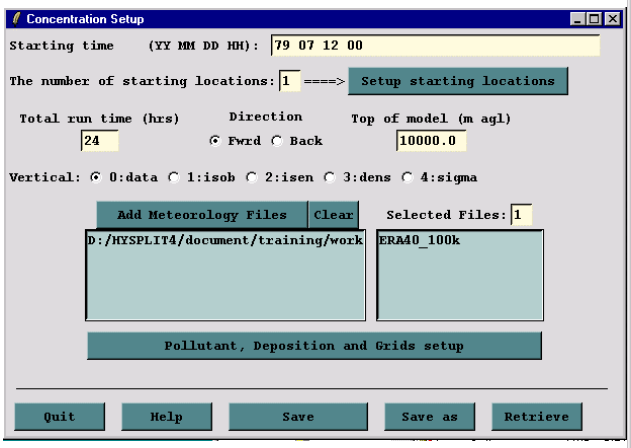




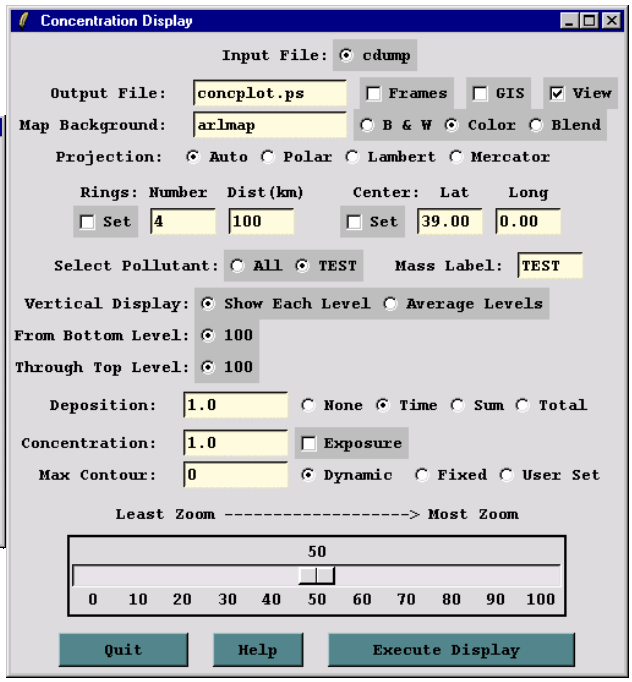
Previous Page | Next Page

Example Dispersion Calculation

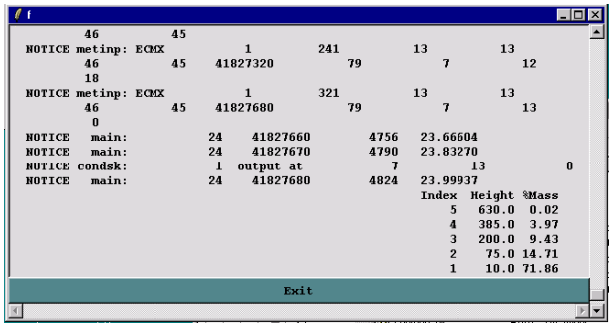
The concentration setup menu is shown for the example at the beginning of this section (ECMWF data, release 39N 0W 10m, 24 h emission and simulation, snapshot concentration after 24 h, and 5000 particles – the advanced menu).



After "run standard model", the results can be displayed through the "display concentration" menu. In the example, default values were used to generate the graphic. As an exercise, the high- resolution map background file (map_spain) could replace the default map background file.



All HYSPLIT simulations generate a text MESSAGE file, which contains diagnostic information about the calculation. Use the GUI from the advanced menu tab to view the file. In this case, at the end of the simulation, 23.99937 units of mass were still on the domain. The vertical mass distribution showed 96% of the mass to be within 200 m of the ground. The vertical mass distribution is computed independently of the vertical concentration grid.

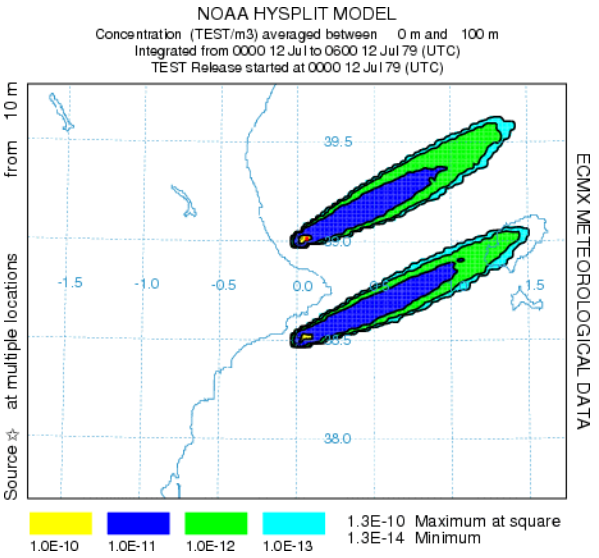


As an exercise, the model should be rerun from the command prompt using the CONTROL file and SETUP.CFG files that were created by the GUI. These could then be edited manually for different simulations. Most programs, such as the concentration-plotting program, have additional command line options available that are not supported by the GUI.

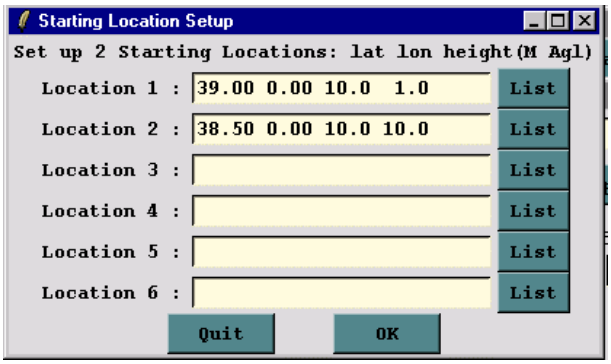
Previous Page	Next Page

Defining Multiple Sources

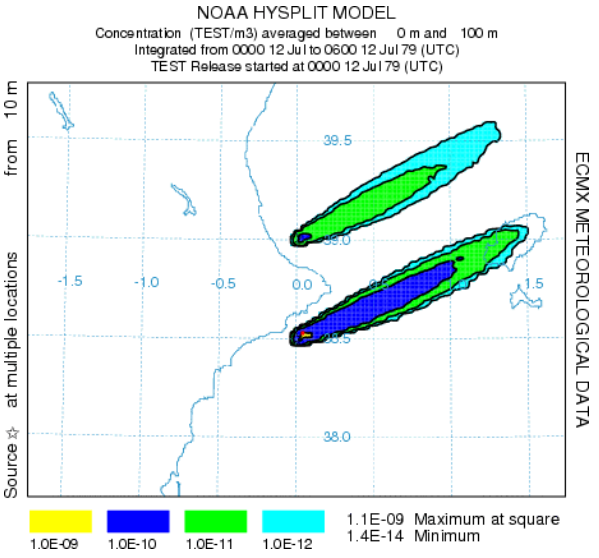
The CONTROL file for the base example in this section has been slightly modified so that the run duration and output interval is now only 6 hours, the concentrations are averaged, and the emission grid resolution has been reduced to 2 km. A second source added to location 38.5N results in two adjacent, almost identical plumes. Note that the emission rate of 1 unit per hour is applied to all sources.



The emission rate can be set for each source by including that information after the release height. Although not shown in the GUI, this is explained in more detail in the linked help files. In the example shown here, the emission rate at the second source has been increased by a factor of 10.



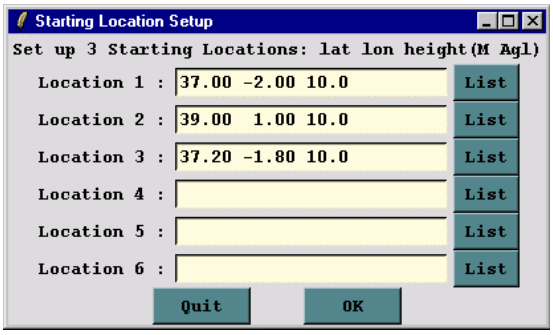
Rerunning the model with the new CONTROL file results in a plot with an obvious concentration difference between the plumes. Although the color scales are different between the two plots, the concentrations in the second plume have increased by the same amount as the emission increase. There is a fifth field that can be added to the source location line that sets an initial plume size (virtual source). The field represents the area in square-meters and is only valid for “puff” simulations.



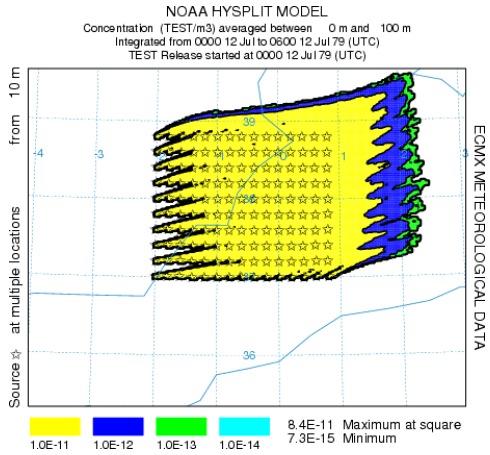
Previous Page | Next Page

Simulations using Emissions Grids

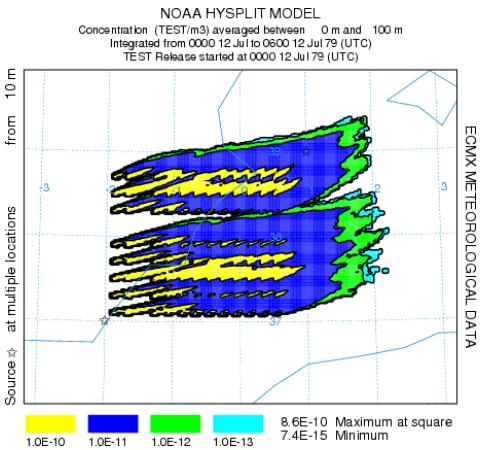
There are two approaches to modeling more complex emission scenarios. The first is an extension of the use of the control file in creating an emission matrix using three locations, the first two representing the grid corners, and the third point representing the grid spacing. The model is then run from the "Special Simulations – Run Matrix" menu.



Prior to executing the model, the initial 3-location control file is rewritten to a new control file showing 150 locations. The default graphic for the model simulation shows a 6-h average concentration with each emission point location marked. This graphic may not be desirable for some applications. Using additional command line options not available through the GUI, a considerable simplified graphic can easily be obtained. The modified control file could be edited manually to customize the emission rate.



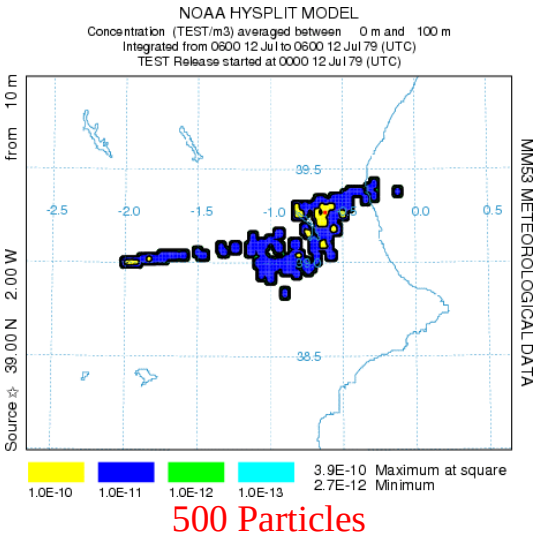
Another approach is to define an emissions file, which tabulates hourly emission rates by location. An emission text file, defines the grid to which those emissions data will be accumulated. A simulation using this approach for the same case is shown here. The control file must define the lower left and upper right corners of the desired emission domain, which may be larger or smaller than the data available. The results shown here have about the same concentration levels as the control file emission run. However, there appear to be some systematic gaps. This is because the internally defined grid resolution (0.2 degrees) is the same as the resolution of the input data, resulting in occasional round-off errors. Reducing the internal resolution to 0.19 degrees can eliminate this problem. Due to the continuous emissions at many locations, all these simulations require the maximum number of particles to be increased from its default value.



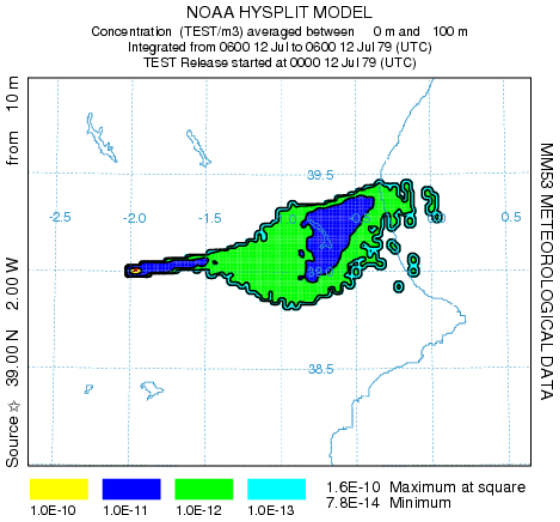
Previous Page | Next Page

Concentration and Particle Display Options

Set up a 3D-500 particle, 6-h snapshot simulation from 2W-39N using a 0.02 degree resolution grid and the 12-km MM5 meteorological data. The concentration output clearly shows a noisy pattern indicating too few particles. Run the simulation again with 5000 particles and the results will be much smoother. However if it is run with 50,000 particles, the concentration pattern now shows more structure. These simulations can take quite some time and if the model is re-run with the much quicker default scenario of 500 top-hat puffs, the results are very similar to the 50,000 3D-particle simulation.

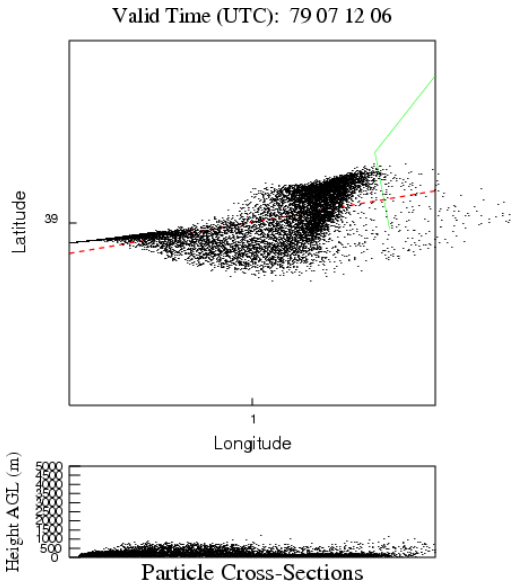


500 Particles



50,000 Particles

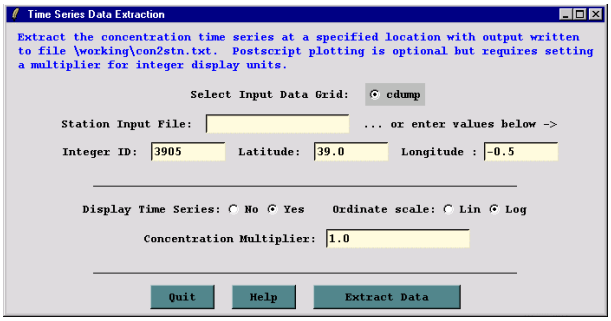
The display menu has options to show snapshot particle distributions, if the particle dump option was set in the advanced configuration menu. Horizontal, vertical, and cross-sections views are available. The cross-section view of the 50,000 particle run is drawn automatically based upon the particle distribution. Another option is to view the concentration values directly on the grid without any interpolation through the "pointer select" tab. This will show the entire concentration domain and normally the "span" should be adjusted to optimize the view region.

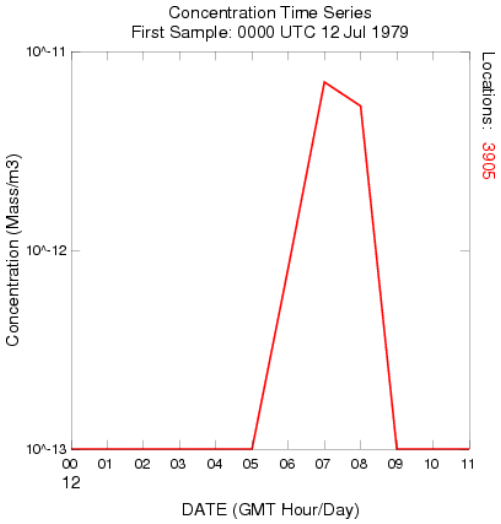


Previous Page	Next Page

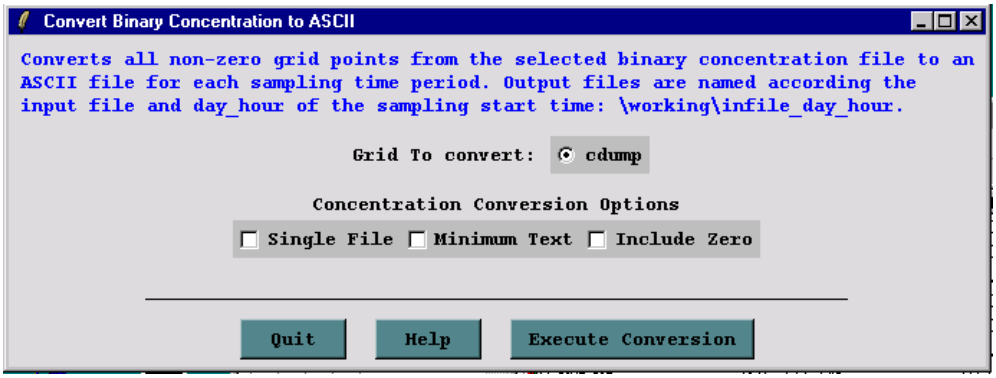
Converting Concentration Data to Text Files

The concentration output file is in a binary format. There are several options available through the "Utility Programs" menu that can be used to convert the concentration data to other formats. First prepare a multi-time period output file by setting up a simulation as in the previous example, run the model for 12-h with hourly average output fields using location 39N-2W. The continuous emission plume moves east sweeping from north to south. Upon completion select the "Grid to Station" menu and select a point downwind (39N-0.5W) and give it a unique ID. An ASCII output file will be created with the concentration values interpolated to that location. If there are multiple locations, then an input data file must be created with the station locations. Selecting the "Display Time Series" button results in the creation of a time series plot.





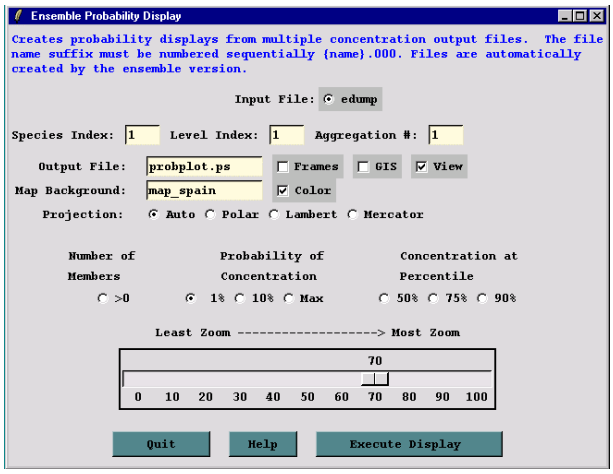
The "Convert to ASCII" menu will convert every non-zero grid point value to its ASCII equivalent, writing the output to one file per time period. Files are labeled according to the name of the binary file, Julian day, and hour of the sampling period. See the contents of this file for the output from the first time period.

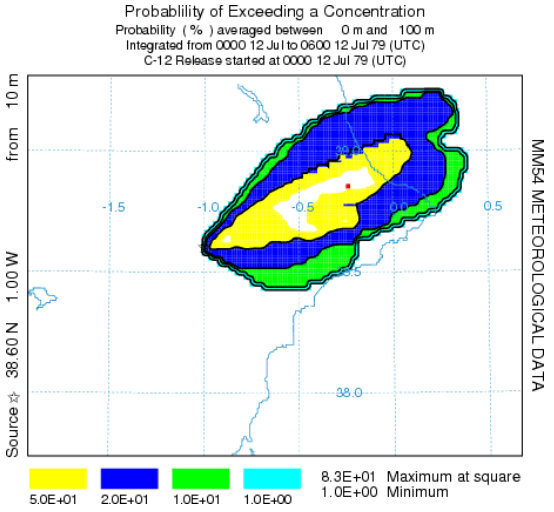


Previous Page	Next Page

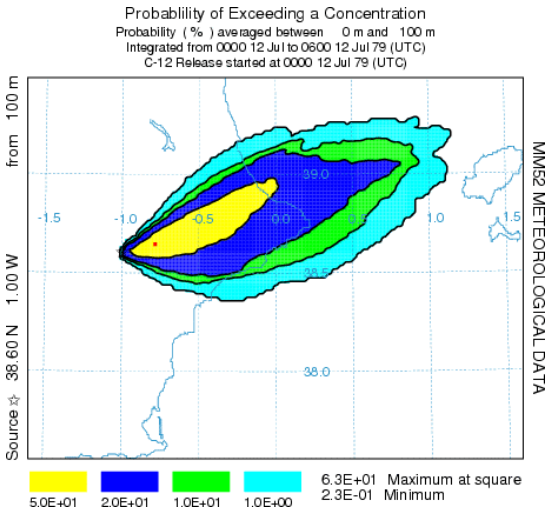
Concentration Ensembles

Instead of creating a single deterministic air concentration simulation, several programs are included that can be used combine multiple HYSPLIT simulations into a single graphic that represents some variation of a concentration probability. The simplest approach is to run the model multiple times varying some parameter. In the first case the model is run with multiple meteorological data. Each simulation was from 38.6N-1W from 10 m. The first 6-h average concentration result for each meteorological data set is given for data from NCEP, ECMWF, MM5-108km, MM5-36km, MM5-12km, and MM5-4km. To convert these data to a probability format, the binary output files must be named with the suffix .001, .002, etc. Then select "Ensemble" from the "Display Options" menu. Three display options are available: the number of members, the probability of exceeding a concentration, and the concentration at different probability levels. Selecting the 1% of the maximum, results in a probability plot at the 10^{-12} level.





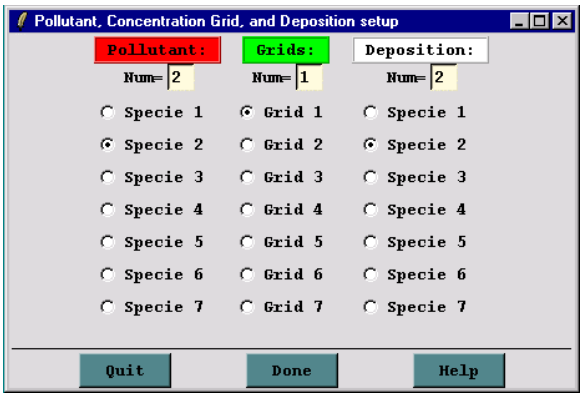
Another possibility is to generate an internal ensemble from a single meteorological data set. This computation is part of HYSPLIT and can be selected as the "Run Ensemble" option from the "Special Simulations" menu tab. In these simulations the meteorological data are perturbed to test the sensitivity of the simulation to the flow field. Twenty-seven members are produced. The ensemble for the 36-km MM5 case is shown here to the right. Ensemble sensitivity can be set in the "advanced configuration menu



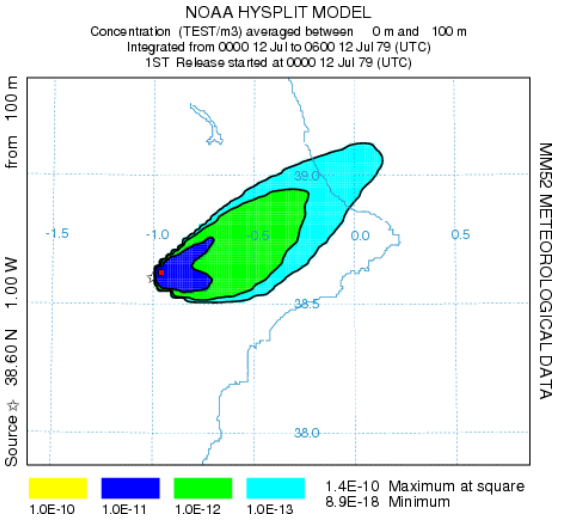
Previous Page	Next Page

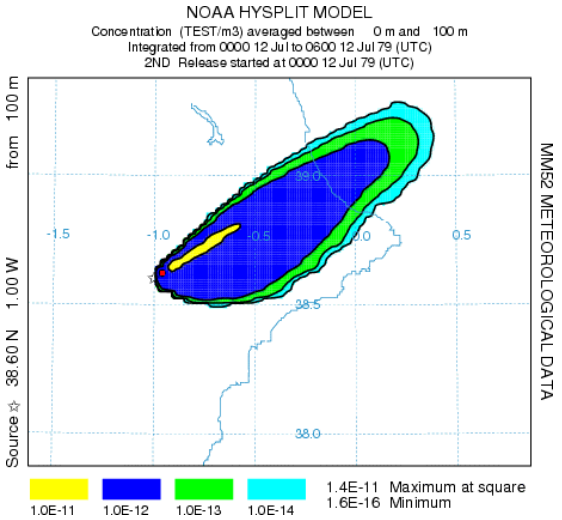
Chemistry Conversion Modules

Normally pollutants are treated independently – one pollutant per particle. However multiple pollutants per particle can be defined by enabling the simple in-line-chemical conversion module through the "advanced configuration" menu. To demonstrate this capability first run the base simulation for one pollutant, configured similar to the previous example (36-km MM5, 6-h duration). Next configure the model for two pollutants, through the "Pollutant, Grid, Deposition" setup menu. Give each pollutant a unique name and configure the 2nd pollutant for no emissions. Running the model with this configuration will give the same result as before. Enable the 10%/hour chemical conversion module through the "advanced configuration" menu and running the model will now produce concentrations in the second pollutant.



Multiple pollutants can be selected from the "Display Concentration" menu. The "All" option sums pollutants to one map. The default 10% per hour conversion is not very interesting. The rate can be modified by creating a "chemrate.txt" file to define the species index and for this example a 50% conversion rate. If the file is placed in the model startup directory, the conversion module will use these values and produce the following results for the two pollutants.



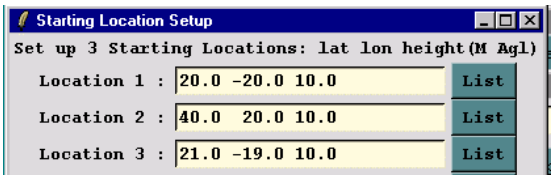


Previous Page | Next Page

Modeling PM10 Emissions from Dust Storms

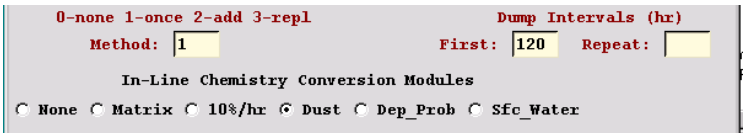
The model contains a PM10 emission algorithm that will emit particles from grid-cells with a desert land-use classification and a friction velocity that exceeds a certain threshold value. More detail on this approach can be found in the on-line HYSPLIT references.

To set up the model for such a simulation, an approach similar to the matrix configuration is employed. Set up three starting locations representing the domain limits over which dust will be emitted and the third representing the grid resolution.

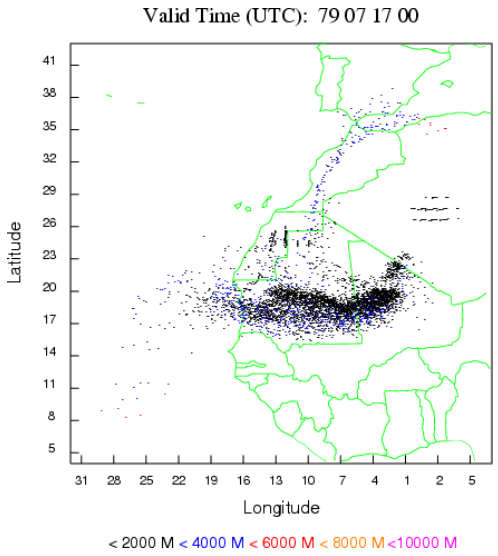


For this example set up the model to run for a region over North Africa, starting on July 12, for a duration of 5 days. Use the ECMWF data and set the concentration grid resolution to 0.2 degrees to speed up the calculation. The emission duration should also be set to 120 hours.

The initial control file will only have 3 starting locations. From the "Advanced Configuration" menu, set the conversion module to "Dust", output a particle dump file after 120 hours, and set the default mode to 3D particle. The particle dump output file will also be used in the next example. Other parameters in the "namelist" file can have their default value.



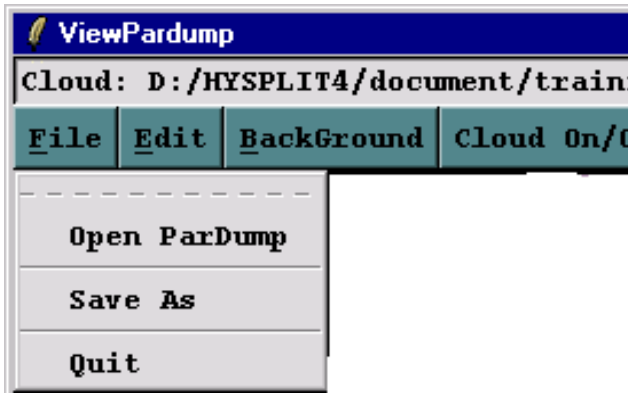
The model is run from the "Run Dust Storm" tab of the "Special Simulation" menu. This causes the execution of a pre-processor that writes out the value of all the desert land-use locations to the control file that will be used for the simulation. Only grid cells where the friction velocity exceeds the threshold will emit dust. Upon completion use the particle display program to show the dust distribution. Note the dust at higher elevations over southern Spain.

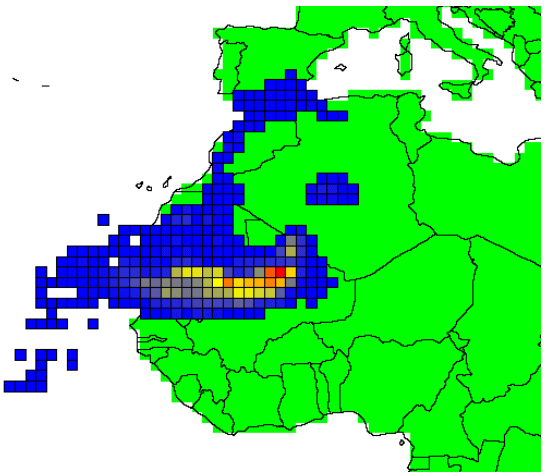


Previous Page	Next Page

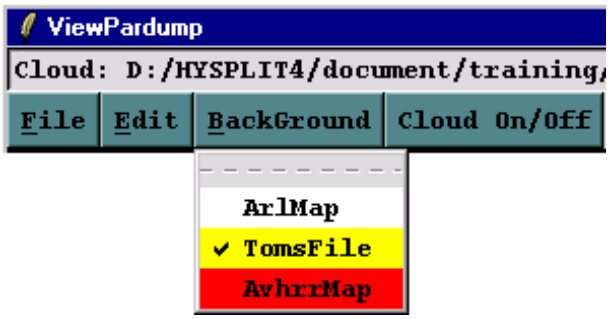
Restarting the Model from a Particle Dump File

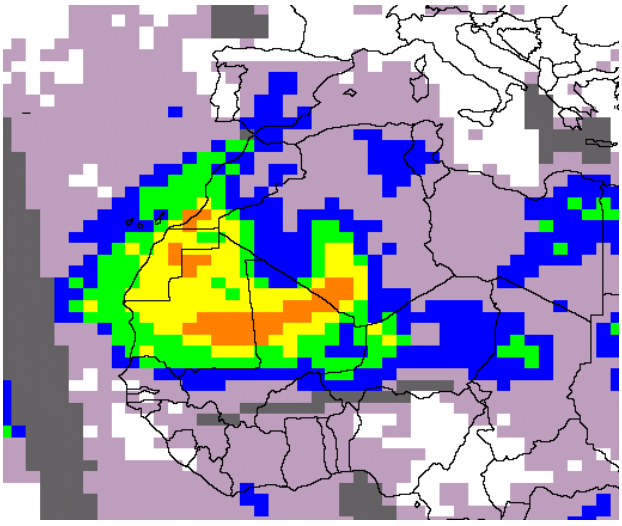
The “Particle Editor” tab from the “Advanced” menu has several functions. In addition to viewing the particle dump file, it can also be used to move those particles to a different location, and then re-write the file. The model simulation can then be restarted from the adjusted particle dump file from the configuration menu. The menu opens a blank map with several buttons. Selecting “File” is used to open the particle dump file. Use the file created in the dust storm example. Particle densities are shown at one-degree resolution.





The background tab is used to select either a TOMS aerosol index file, which can also be downloaded through the menu from the NASA web site, or a NOAA AVHRR file of optical depth over the oceans. More detail on these files is provided in the User’s Guide. Loading the TOMS image for July 16th yields a picture remarkably similar to the model calculation, including a weak dust plume over southern Spain. In this particular situation the model and measurements are so close that no adjustment is necessary.



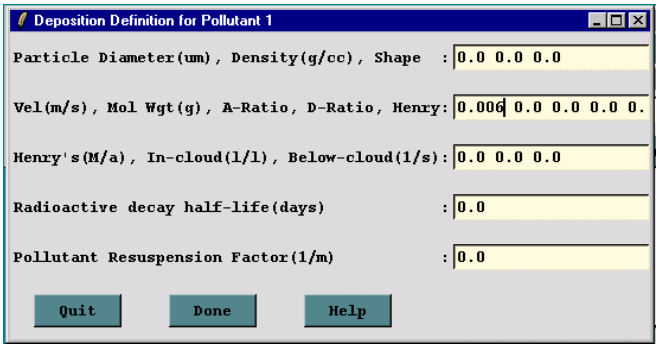


Previous Page | Next Page

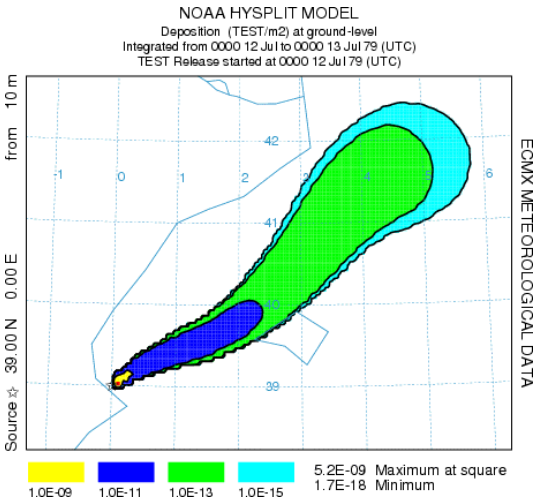
Pollutant Deposition

The deposition (D) from a particle is expressed as a fraction of the mass (m) computed as the sum of different time constants, $D_{wet+dry} = m \{1-\exp[-\Delta t (\beta_{dry}+\beta_{gas}+\beta_{inc}+\beta_{bel})] \}$.

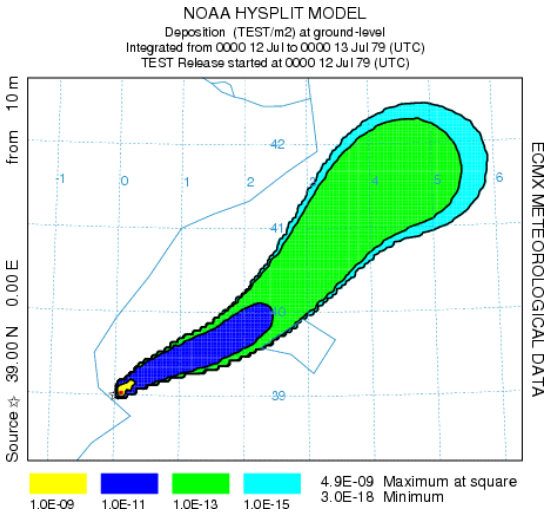
Configure the model to run for 24 hours with default values, from 39N-0W using ECMWF data, releasing one unit over one hour. Set the concentration grid for 0.05 degrees resolution with one output level at zero meters (for deposition). This simulation will yield no results because deposition has not been turned activated. Open up the deposition menu and set the velocity to 0.006 m/s. When entered directly as a velocity (Vd), then $\beta_{dry} = V_d \Delta Z_p^{-1}$. The results show the deposition left by the puff as it moved across the domain.



The dry deposition of particles due to gravitational settling can also be computed from the particle diameter and density: $V_g = d_p^2 g (\rho_g - \rho) (18 \mu)^{-1}$. Enter a density of 5 g/cc and a diameter of 6 um, which should result in a settling velocity close to previous 0.6 cm/s. The results from this configuration are almost identical to the previous calculation.



Deposition Velocity



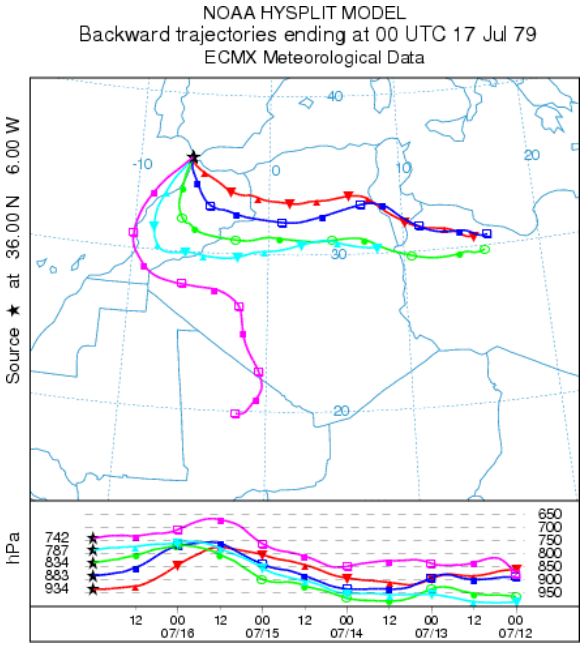
Settling Velocity

The default approach is for the number of particles and puffs to remain the same but lose mass due to deposition. If the "Deposition Probability" option is selected in the advanced menu, then particles will be lost if $R < \beta_{dry} \Delta t$, and where R is a random number (0-1). The model must be configured for the 3D particles for this option. If a sufficient number of particles are released the results will be identical to the other deposition options.

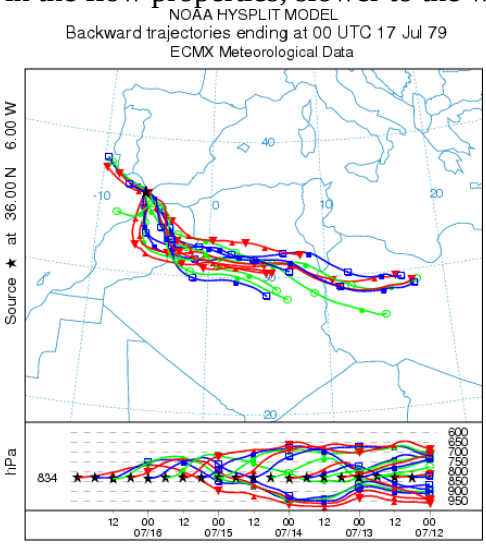
Previous Page	Next Page

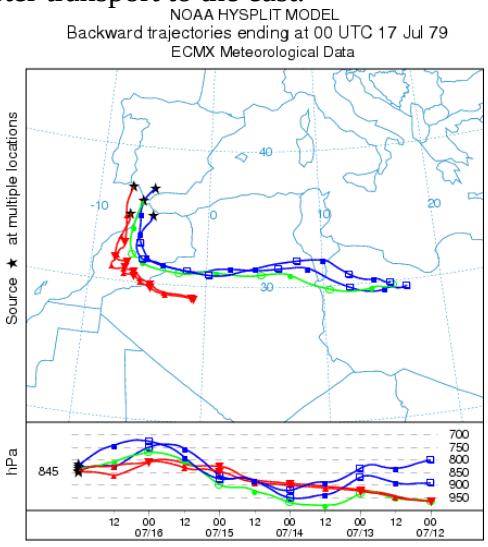
Source Attribution via Back Trajectories

Frequently it is necessary to attribute a pollutant measurement to a specific source location. One approach is to compute a backward trajectory to determine the air's origin. Although it is not uncommon to see sources identified by a single trajectory, the uncertainties inherent in a single-trajectory can preempt its utility. One way to reduce those uncertainties would be to compute multiple trajectories, in height, time, and space. For instance if we use the previous example of dust observations (from TOMS) over southern Spain on July 17th as a starting point, a backward trajectory calculation for multiple heights clearly shows transport at all levels from North Africa.



Quickly changing meteorological conditions also contribute to uncertainty, especially if a pollutant sample represents an average rather than a snapshot concentration. Set the height (1500 m) and from the advanced menu set the restart interval to 6 hours. Over the 5-day period, most of the trajectories originated in North Africa. Closer examination of the trajectory output file would determine when the temporal shift occurred. The third variation is to examine the spatial sensitivity. In this simulation, set four additional starting points, offset by 1 degree from the central location. The result shows a division in the flow properties, slower to the west and faster transport to the east.



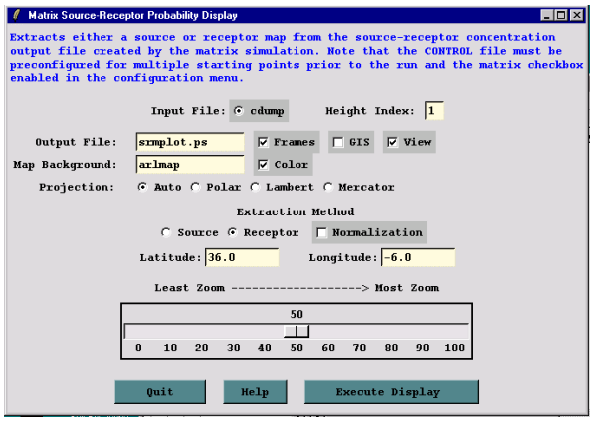


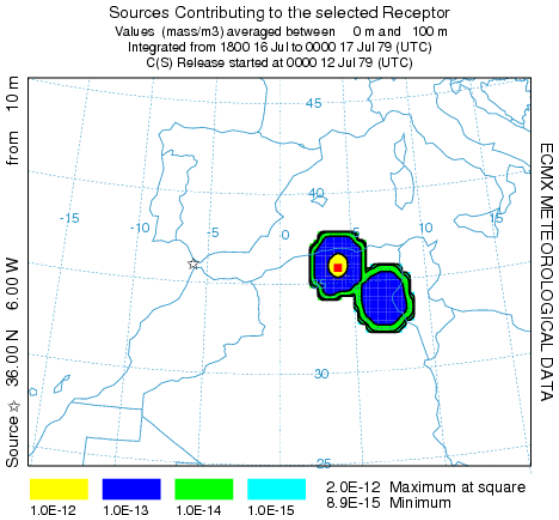
Previous Page	Next Page

Source Attribution via Source-Receptor Matrices

The matrix approach to source attribution is set up in a similar manner to the previous discussion on to run the model with multiple sources. Use the same 3 starting locations as with the dust storm simulation to define the source region (20N-40N and 20W-20E). Use the ECMWF data, run the 3D-particle model for 120 hours with concentration output every 6 hours. Reduce the resolution of the concentration grid to 0.50 reduce memory requirements. Prior to executing the model through the "Run Matrix" tab of the "Special Simulations" menu, it is necessary to check the "Matrix" button in the advanced configuration menu. This causes the concentration grid to be reconfigured so that every source location with the matrix (861 in this example) will have its own concentration grid.

The final concentration output file can only be displayed through the "Matrix" tab of the display menu. The term "matrix" has two connotations with respect to HYSPLIT. In the earlier application, a matrix of sources was created. These results are summed to a single concentration file. In this application, defining a concentration grid for each source creates a matrix of sources and receptors. This requires a special preprocessor for the display program. In this example, selecting the "receptor" button causes the entered latitude-longitude value to be treated as the receptor point. The dispersion factors from each source to that receptor are then interpolated to a grid and displayed by time period. The last time period, coincident with the TOMS image, is shown below.

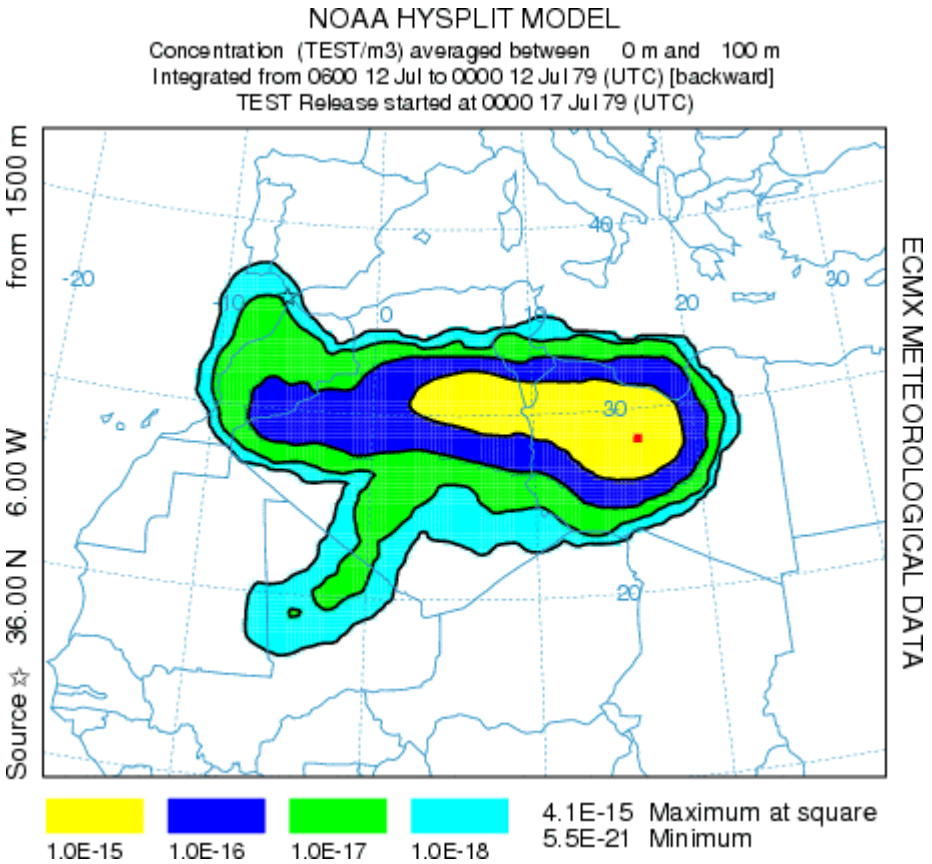




Previous Page | Next Page

Source Attribution Functions

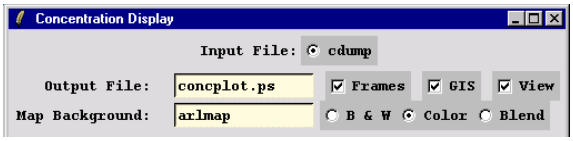
Running the air concentration prediction model backwards is comparable to a back trajectory calculation but it includes the dispersion component of the calculation. The result, although it looks like an air concentration field is more comparable to a source attribution function. If the atmospheric turbulence were stationary and homogeneous then this attribution function would yield the same result from receptor to source as a forward calculation from source to receptor. Configure the model as the default top-hat-particle to run from the receptor location (36N-6W-1500 m) for 120 hours backward from July 17th using a 1-hour emission and with concentration output every 6 hours. The elevated release corresponds to the elevated nature of the TOMS signal. The concentration grid level should be 100 m to correspond to lower-level emissions. The results are shown below for the last 6-hour period in the simulation. They would be interpreted to mean that the emissions in the "yellow" region between 0000 July 12 and 0600 July 12 were most likely to have contributed to the measurements on the 17th at 1500 m at 36N-6W.

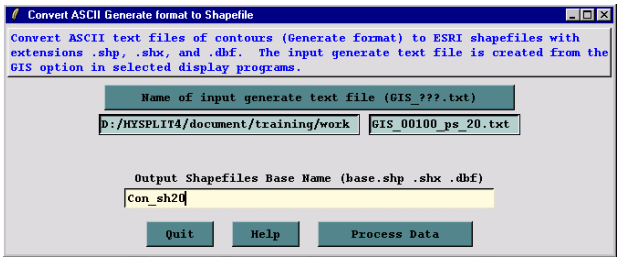


Previous Page	Next Page

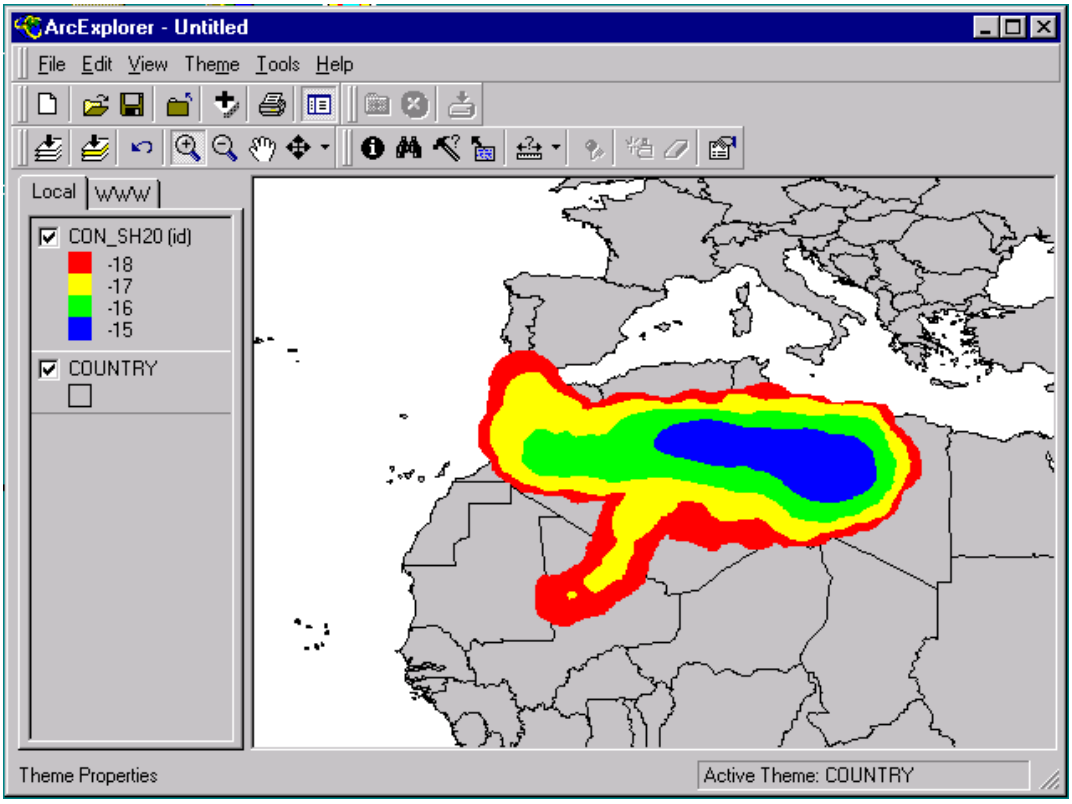
Shapefiles for ArcExplorer

Graphical output from most GUI programs can be converted into an ESRI GIS format for programs such as ArcExplorer. To convert the graphic generated in the last example, check the “Frames” and “GIS” boxes in the “Concentration Display” menu. This will result in a unique Postscript and text file for each time period. The “Generate format” text output file contains the latitudes and longitudes for each contour.





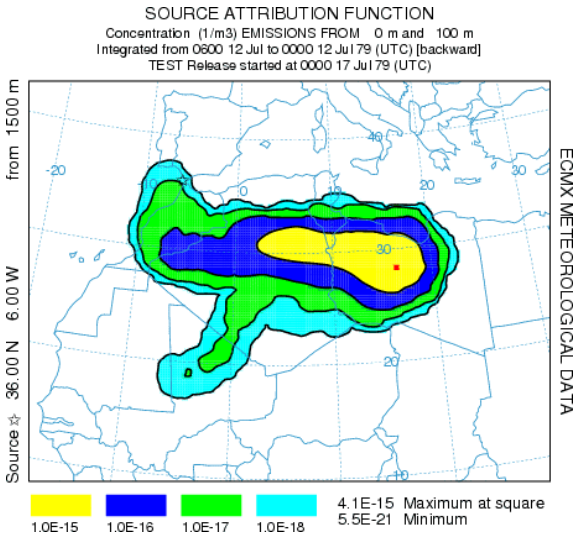
From “Utility Programs”, use the ASCII to Shapefile menu to select text files. They are named by level and time sequence. Select the last file (#20) and rename the output file to reflect the same number. When finished there will be a series of files with the suffix “shp”, “shx”, and “dbf”. Open ArcExplorer and add the Con_sh20 theme and a map background theme, then select both, and center the map.



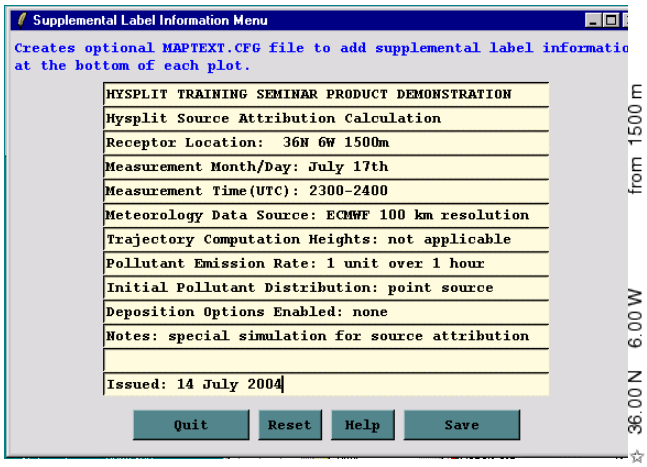
Previous Page	Next Page

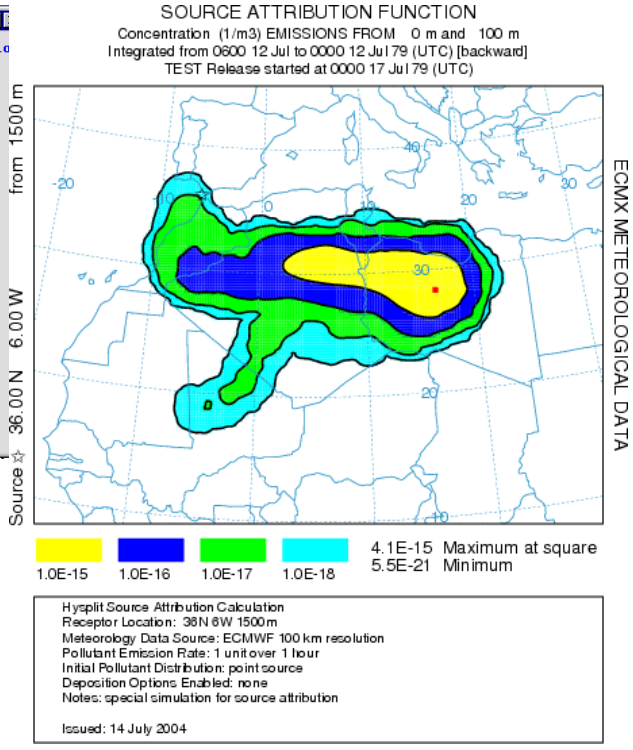
Customizing Map Labels

The concentration display menu only contains a “mass label” entry that can be used to customize map labels. This changes the text on the second line for the mass units of concentration. This would normally be used in conjunction with the concentration multiplier if for instance emission units were grams but display units of micro-grams were desired. Additional label information can be changed if the concentration display program finds a file called “LABELS.CFG” as shown in the adjacent graphic taken from the previous example.



Supplemental text information can be added at the bottom of each plot by entering information on the “Extra Label” menu called from the “Advanced” menu tab. This creates a file called “MAPTEXT.CFG” which is read by both the trajectory and concentration plotting programs.





Previous Page	Next Page

Scripting for Automated Operations

Both the /trajmdl and /concmdl directories contain example TCL scripts that can be used to automate computations. Familiarity with the command line options is essential in modifying and writing new scripts. Script syntax is very similar to the "C" language. All scripts work in the same manner by writing a new control file for each simulation, running the model, and then renaming the output. In the example shown in the text box below, trajectories are computed at four locations. The key statements are shown in red. As an exercise, this script should be modified to run a new trajectory each day, from the 12th through the 16th, but at only one location.

```
       # Auto_traj.tcl
       # the next line restarts using wish \
       # exec wish "$0" "$@"

       set Start_hgt "10.0"
       set Traj_path  "/hysplit4/exec"
       set Start_time "00 00 00 00"
       set Run_hours  "24"
       set Vert_coord "0"
       set Top_model  "10000.0"
       set Meteo_path "./"
       set Meteo_file "ERA40_100k"
       set Output_path "./"
       set Output_base "tdump"

       set Output_numb 1
       foreach {Start_lat Start_lon} {36.0 0.0  37.0 0.0  \
                                      38.0 0.0  39.0 0.0} {

          set Start_loc "$Start_lat $Start_lon $Start_hgt"

          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_base"
          close $f

          exec $Traj_path/hymodelt.exe
          exec $Traj_path/trajplot.exe $Output_base
          file rename tdump tdump${Output_numb}
          file rename trajplot.ps plot${Output_numb}.ps
          incr Output_numb
       }
destroy
```

Previous Page