PNNL-24049

Pacific Northwest NATIONAL LABORATORY

Proudly Operated by Battelle Since 1965

DUSTRAN 2.0 User's Guide:

A GIS-Based Atmospheric Dust Dispersion Modeling System

March 2015

WJ Shaw
FC Rutz
JP Rishel
EG Chapman

U.S. DEPARTMENT OF ENERGY

Prepared for the U.S. Department of Defense Strategic Environmental Research and Development Program under a Related Services Agreement with the U.S. Department of Energy under contract DE-AC05-76RL01830

PACIFIC NORTHWEST NATIONAL LABORATORY
operated by
BATTELLE
for the
UNITED STATES DEPARTMENT OF ENERGY
under Contract DE-AC05-76RL01830

Printed in the United States of America

Available to DOE and DOE contractors from the
Office of Scientific and Technical Information,
P.O. Box 62, Oak Ridge, TN 37831-0062;
ph: (865) 576-8401
fax: (865) 576-5728
email: reports@adonis.osti.gov

Available to the public from the National Technical Information Service
5301 Shawnee Rd., Alexandria, VA 22312
ph: (800) 553-NTIS (6847)
email: orders@ntis.gov <http://www.ntis.gov/about/form.aspx>
Online ordering: http://www.ntis.gov

This document was printed on recycled paper.
(8/2010)

PNNL-24049

DUSTRAN 2.0 User's Guide:

A GIS-Based Atmospheric Dust Dispersion Modeling System

WJ Shaw
FC Rutz
JP Rishel
EG Chapman

March 2015

Prepared for the U.S. Department of Defense Strategic Environmental Research and Development Program under a Related Services Agreement with the U.S. Department of Energy under contract DE-AC05-76RL01830

Pacific Northwest National Laboratory
Richland, Washington 99352

Summary

The U.S. Department of Energy's Pacific Northwest National Laboratory (PNNL) completed a multi-year project to update the DUSTRAN atmospheric dispersion modeling system, which is being used to assist the U.S. Department of Defense (DoD) in addressing particulate air quality issues at military training and testing ranges. This version—DUSTRAN V2.0—includes (1) a complete replacement of the geographic information system (GIS) platform to include the utilization of open-source GIS software (MapWindow) (2) user profiles to allow multiple users to run individual simulations using separate sites (3) integration of the Environmental Protection Agency's (EPA's) AERMOD air-dispersion model for modeling near-field (less than 50 km) releases, including an option to use a formulation for near-field deposition derived from new work by the Desert Research Institute; and (4) updates to the dust source-term module for military vehicles to include additional vehicle and soil types, as well as enhanced formulations for particle deposition. The project was funded by DoD's Strategic Environmental Research and Development Program (SERDP).

DUSTRAN V2.0 includes widely used, scientifically defensible atmospheric dispersion models and model components that are coupled with state-of-science dust-emission formulations in one, easy-to-use interface. The DUSTRAN modeling platform is built on the MapWindow open-source GIS software, and includes the EPA-approved AERMOD and CALPUFF dispersion models for modeling active-source dust emissions, as well as the CALGRID dispersion model for modeling wind-blown dust from large areas. DUSTRAN includes the necessary terrain and meteorological preprocessors that are required to run the dispersion models. Execution and data transfer between the preprocessors and dispersion models is managed automatically by the interface at runtime. DUSTRAN also includes dust-emission modules for generating source terms from both tracked and wheeled military vehicle activities as well as wind-blown dust generation from larger areas. The primary features of DUSTRAN include:

- A modeling domain that is graphically specified and size-selectable (20 km to 400 km).
- Dispersion models for treating near-field (AERMOD) and far-field (CALPUFF) dispersion from active emission sources and a gridded dispersion model (CALGRID) for modeling passive, wind-blown dust emissions from large areas.
- An "Add Site" utility for creating new modeling sites and the supporting files and data structure needed for a simulation.
- Easily specified single- or multi-station meteorology.
- Multiple point, area, and line sources created graphically within a scenario.
- Easily specified simulation and release times (typically a few hours to a few days) within the user interface.
- Resulting concentrations and deposition contours that are viewable within the GIS interface and can be animated to view the time-progression of the plume.
- Simulations of multiple particle sizes and gaseous species in a single run.
- Treatment of dry deposition as well as complex terrain effects.

This manual documents DUSTRAN V2.0 and includes installation instructions, a description of the modeling system, and detailed example tutorials.

Acknowledgments

This research was supported in part by the U.S. Department of Defense through the Strategic Environmental Research and Development Program (SERDP Project RC–1729) under a Related Services Agreement with the U.S. Department of Energy (DOE) under Contract DE-AC05-76RL01830. Pacific Northwest National Laboratory (PNNL) is operated for DOE by Battelle.

Dr. Jack Gillies of Desert Research Institute (DRI) provided emission-factor data for wheeled military vehicles as well as new observations of deposition velocity incorporated into DUSTRAN as part of this project. The emission factors were developed under a previous Strategic Environmental Research and Development Program project (CP-1191) with DRI.

Dr. John Hall, the SERDP program manager, provided valuable guidance and support during all phases of the development of DUSTRAN.

Acronyms and Abbreviations

AMS	American Meteorological Society
AERMAP	AMS/EPA MAPping program
AERMET	AMS/EPA METeorological model
AERMOD	AMS/EPA Regulatory Model
CALGRID	CALifornia photochemical GRID model
CALMET	CALifornia METeorological model
CALPOST	CALifornia POST-processing program
CALPUFF	CALifornia PUFF model
CST	Central Standard Time
DEM	digital elevation model
DoD	U.S. Department of Defense
DOE	U.S. Department of Energy
DRI	Desert Research Institute
DUSTRAN	DUST TRANsport
EPA	U.S. Environmental Protection Agency
ESRI	Environmental System Research Institute
EST	Eastern Standard Time
FSL	Forecast Systems Laboratory
GIS	geographic information system
GLCC	global land-cover characteristics
M-O	Monin-Obukhov (similarity theory)
MST	Mountain Standard Time
NARCS	number of arc distances
NOAA	National Oceanic & Atmospheric Administration
NWS	National Weather Service
NTC	National Training Center
OWE	Olson World Ecosystem (database)
PGEMS	Pacific Gas and Electric Modeling System
PGT	Pasquill-Gifford-Turner specifications
PM	particulate matter
PNNL	Pacific Northwest National Laboratory
PST	Pacific Standard Time
SERDP	Strategic Environmental Research and Development Program
USGS	U.S. Geological Survey
UTM	Universal Transverse Mercator

Contents

Figures

Tables

1.0 Introduction

1.1 Background

Activities at U.S. Department of Defense (DoD) training and testing ranges can be sources of dust into local and regional air sheds governed by air quality regulations. Activities that could disturb the soil surface, and thus generate dust, include vehicle and troop maneuvers, convoy movement, helicopter activities, munitions impacts, roadway preparations, and wind erosion. Other sources of particulates include smokes and obscurants, controlled burns, and engine operations.

From January 2001 through August 2006, the Pacific Northwest National Laboratory (PNNL), operated by Battelle for the U.S. Department of Energy (DOE), carried out a multi-year research project funded primarily by DoD's Strategic Environmental Research and Development Program (SERDP) to develop an atmospheric dispersion modeling system to assist the DoD in addressing particulate air quality issues at military training and testing ranges. The culmination of that work was the development of the DUST TRANsport, or DUSTRAN, modeling system V1.0 (Allwine et al. 2006). DUSTRAN V1.0 functioned as a console application within the ArcMap Geographic Information system (GIS) and included the U.S. Environmental Protection Agency (EPA)-approved CALPUFF dispersion model for modeling active sources of dust emissions from military vehicular activities and the widely used CALGRID dispersion model for modeling wind-blown dust generation. Source terms for vehicular and wind-blown dust are both native to the DUSTRAN modeling system.

In 2010, SERDP funded additional development of DUSTRAN to 1) replace the GIS platform with open-source GIS software called MapWindow; 2) enable user profiles to allow multiple users to run individual simulations using separate sites; 3) add the EPA's AERMOD air-dispersion model for modeling near-field (less than 50 km) releases; and 4) update the vehicular dust source-term module to include additional vehicle and soil types, as well as enhanced formulations for particle deposition. The culmination of this work is the DUSTRAN V2.0 modeling system. Unless otherwise indicated, subsequent references to "DUSTRAN" in this document refer to DUSTRAN V2.0.

The primary objectives in formulating DUSTRAN have been to 1) identify and construct the system from widely available, scientifically defensible models and model components; 2) couple and integrate the models within a user-friendly, open-source GIS interface; 3) develop and implement an advanced dust-emission model into the modeling system; and 4) document the system through technical articles and a supporting user's guide. This user's guide supports that final objective.

Figure 1.1 identifies the primary components of DUSTRAN, which include the following:

- The MapWindow GIS interface, which allows the user to launch the DUSTRAN console application, view model data layers, and navigate (e.g., zoom, pan) the model domain.
- The DUSTRAN console for entering user inputs, including the model domain, release period, sources, and meteorology and selecting model output display options.
- The dust-emissions modules for estimating active and wind-blown dust-emission rates as a function of time and location.
- The atmospheric dispersion models for estimating near-field (AERMOD) or far-field (CALPUFF and CALGRID) dust concentrations and deposition patterns.

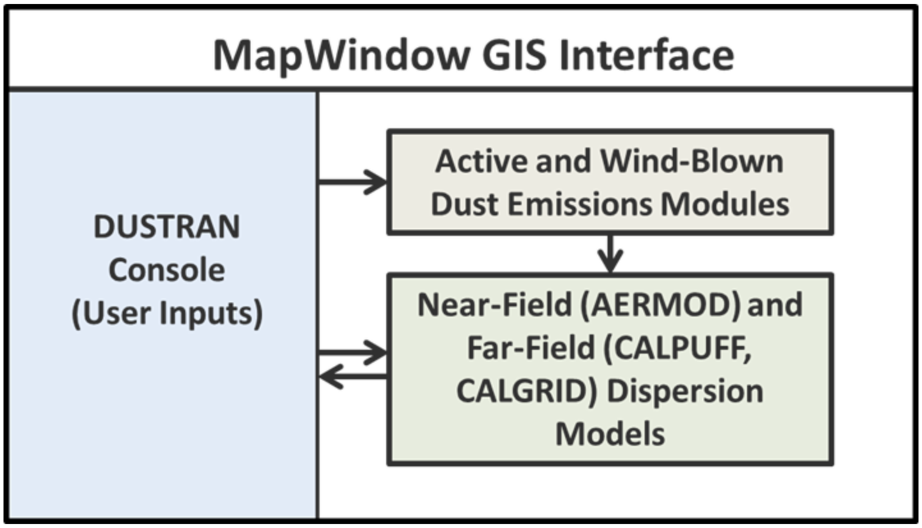


Figure 1.1. Primary DUSTRAN components**.**

1.2 Features

DUSTRAN V2.0 functions as a console application within the open-source MapWindow GIS software and includes near-field (AERMOD) and far-field (CALPUFF) dispersion models for simulating active dust emissions from vehicular sources. In addition, DUSTRAN includes the CALGRID dispersion model for wind-generated dust over the model domain. DUSTRAN includes the dust-emission modules for creating the necessary source-term factors from both the vehicular and wind-blown dust-generation activities. Dust-emission factors from wheeled and tracked military vehicles and from certain civilian vehicles were developed during SERDP projects CP-1191 and SI-1399 (Gillies et al. 2005a, 2005b; Gillies et al. 2010). DUSTRAN V2.0 also incorporates the widely used AP-42 emission factors (EPA 2005) for paved and unpaved roads using EPA online updates dated January 2011 and November 2006, respectively.

The EPA's AERMOD (EPA 2014a) dispersion model is used to model near-field (within 50 km) air concentration and ground deposition from active sources of dust. DUSTRAN includes and automatically runs AERMOD's terrain (AERMAP) and meteorological (AERMET) preprocessors. Data from the preprocessors are seamlessly merged with other user inputs prior to AERMOD's execution. The EPA-approved CALPUFF dispersion model (Scire et al. 2000a) is used to model far-field (greater than 50 km) air concentration and ground deposition from active sources of dust. DUSTRAN includes and automatically runs the CALPUFF terrain (TERREL), land-use (CTGPROC), and meteorological (CALMET) preprocessors. The CALifornia photochemical GRID (CALGRID) dispersion model (Scire et al. 1989) is used to model wind-blown dust from large areas. The CALMET meteorological model (Scire et al. 2000b) provides the meteorological fields (e.g., winds and mixing height) for the CALPUFF and CALGRID dispersion models. The primary features of DUSTRAN include the following:

- The modeling domain is graphically specified and size-selectable (20 to 400 km).
- DUSTRAN operates at any U.S. geographic location and has an "Add Site" wizard that generates a new site's supporting files and data structure for use in a simulation.
- Single-station or multiple-station meteorology can be used and easily specified.
- Multiple point, area, and line releases can be accommodated and specified graphically.
- Simulation and release times are easily specified in the user interface.

- The output concentrations and deposition contours can be viewed graphically, and the output can be animated to view the progression of the dust plume across the modeling domain.
- Multiple particle sizes can be simulated at one time.
- Simulation periods typically range from a few hours to 1 day.
- The atmospheric models process dry deposition using both native formulations and near-field formulations based on SERDP-funded experimental work carried out by DRI under this project..
- The atmospheric models process complex terrain effects.

The DUSTRAN interface allows the user to graphically create a model domain, which defines the area in which a simulation will be performed. For a wind-blown dust simulation, the domain defines the area of potential dust emissions. For source emissions, the domain contains specific emission locations defined by point-, area-, or line-source types. Area and line sources are integrated with a vehicular dust-emission module that allows the user to specify the type, speed, and number of vehicles. The DUSTRAN interface allows the user to specify the time and duration of the sources, size of the modeling domain, duration of the simulation, and source of the meteorological data. After running a simulation, the ground-level dust air concentrations and deposition fields are displayed graphically within the MapWindow GIS interface.

2.0 Technical Overview of the DUSTRAN Modeling System

DUSTRAN is a comprehensive dispersion modeling system, consisting of a dust-emissions module, a diagnostic meteorological model, and dispersion models that are integrated seamlessly into MapWindow's open-source GIS. DUSTRAN functions as a console application within MapWindow and allows the user to interactively create a release scenario and run the underlying models. Through the process of data layering, the model domain, sources, and results—including the calculated wind vector field and plume contours—can be displayed with other spatial and geophysical data sources to aid in analyzing and interpreting a scenario.

Fundamental to the DUSTRAN modeling system is a dust-emissions module that includes algorithms for calculating dust emissions from both active and natural sources. Active sources include vehicular dust generation from paved and unpaved roadway surfaces as well as emission factors for various wheeled and tracked military vehicles (Gillies et al. 2005a, 2005b; Kuhns et al. 2010) and the widely used AP-42 emission factors (EPA 2005). Natural sources include wind-blown dust generation from a user-specified domain and are a function of surface wind stress, soil type, and vegetation type. In either case, dust emissions are calculated for explicit particulate matter (PM) size classes, including $PM_{2.5}$, PM_{10}, PM_{15}, and PM_{30}.

In DUSTRAN, dust transport, diffusion, and deposition are simulated using one of three dispersion models—AERMOD, CALPUFF, or CALGRID. Three dispersion models are used due to their frames-of-reference in calculating plume transport, allowing DUSTRAN users to take advantage of each model's inherent strengths in simulating different source types and transport distances:

- AERMOD (EPA 2014a). A steady-state plume model for near-field (<50 km) dispersion and particle deposition from active-source emissions (i.e., point, area, or line). AERMOD incorporates planetary boundary layer turbulence structure and scaling concepts, including treatment of surface and elevated sources and simple and complex terrains.
- CALPUFF (Scire et al., 2000a). A non-steady-state puff dispersion model for long-range (>50 km) dispersion and particle deposition from active-source emissions (i.e., point, area, or line). CALPUFF simulates the effects of time- and space-varying meteorological conditions on pollution transport, transformation, and removal in both simple and complex terrains.
- CALGRID (Scire et al., 1989). An Eulerian dispersion model used for wind-blown dust emissions, where the entire model domain is a potential emission source.

In addition to the AERMOD and CALPUFF regulatory versions, modified versions of both models have been included in DUSTRAN. The modified versions implement a dry-deposition algorithm based on experimental data from the Desert Research Institute (DRI), which is described further in Section 2.1.3.2.

Each dispersion model has a suite of preprocessors included in DUSTRAN for extracting terrain and meteorological data for use in the dispersion calculations. All the model components are dynamically linked by the DUSTRAN interface.

AERMOD utilizes two data preprocessors that are regulatory components of the modeling system: AERMET (EPA 2014b), a meteorological data preprocessor that incorporates planetary boundary layer turbulence structure and scaling concepts, and AERMAP (EPA 2011), a terrain data preprocessor that incorporates complex terrain using U.S. Geological Survey (USGS) Digital Elevation Model (DEM) data. CALPUFF and CALGRID utilize a suite of geophysical data preprocessors for extracting land-use and terrain data as well as CALMET (Scire et al., 2000b) for creating gridded fields of wind and boundary-layer parameters from observed meteorological data. These gridded fields are then supplied to the CALPUFF and CALGRID dispersion models where they are used in plume dispersion and deposition calculations.

Figure 2.1 shows the primary components of the AERMOD modeling system as implemented within DUSTRAN. Similarly, Figure 2.2 shows the primary components of the CALPUFF and CALGRID modeling systems as implemented within DUSTRAN. Table 2.1 and Table 2.2 list the version numbers of each model component.

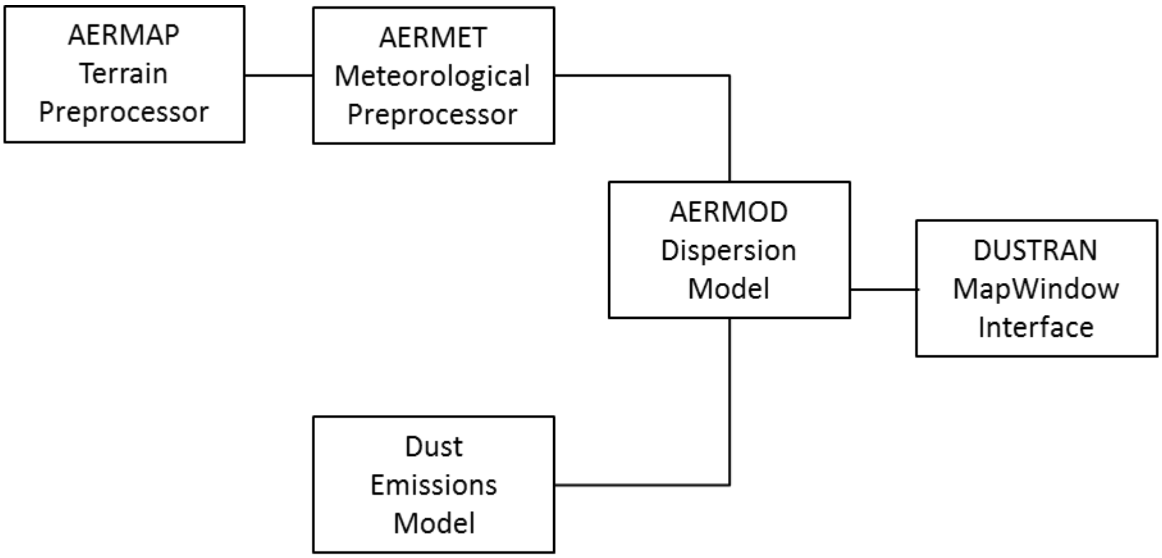


Figure 2.1. AERMOD Modeling Components within DUSTRAN

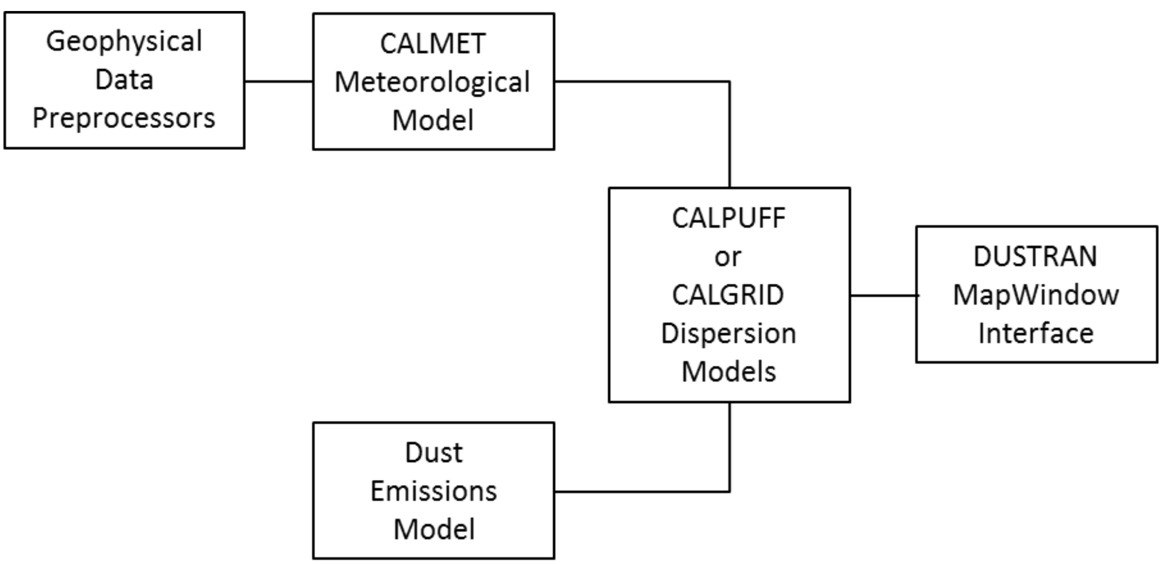


Figure 2.2. CALPUFF/CALGRID Modeling Components within DUSTRAN

Table 2.1. AERMOD Model Components and Version Numbers Implemented in DUSTRAN

Component	Purpose	Version Number
AERMOD	Dispersion Model	14134
AERMET	Meteorological Model	14134
AERMAP	Terrain Preprocessor	11103

Table 2.2. CALPUFF/CALGRID Model Components and Version Numbers Implemented in DUSTRAN

Component	Purpose	Version Number
CALPUFF	Dispersion Model	5.5
CALGRID	Dispersion Model	1.6
CALMET	Meteorological Model	5.2
CALPOST	Post-processing Program	5.2
TERREL	Terrain Preprocessor	2.1
CTGPROC	Land Use Preprocessor	1.2
MAKEGEO	Merges Terrain/Land Use Datasets	1.1
READ62	Meteorological Preprocessor for Extracting Standard Upper-Air Formats	4.0

The following sections provide a brief technical overview of the DUSTRAN model components. Numerous documents on the AERMOD (e.g., EPA 2004a, 2004b, 2004c, 2011, 2014a, 2014b) and CALPUFF/CALGRID (Scire et al. 1989, 2000a, 2000b) modeling systems discuss the theoretical and technical basis of the dispersion and meteorological models used within DUSTRAN; readers are referred to these documents for detailed information on each model component. This user's guide provides a cursory overview of the model components and their integration within the DUSTRAN framework. A more detailed discussion of the vehicular and wind-blown dust-emissions factor module is given in Section 2.5, and serves as the technical documentation and reference for this component.

2.1 AERMOD Modeling System

The AERMOD modeling system (EPA 2014a, 2014b) is used to model near-field (<50 km) dispersion from active-source emissions (i.e., point, area, or line). The following sections describe the AERMOD modeling system components and their implementation within DUSTRAN.

2.1.1 AERMET

AERMET (EPA 2014b) is a meteorological preprocessor that uses hourly meteorological observations to calculate certain boundary-layer parameters (e.g., mixing height and friction velocity) for use in AERMOD. AERMET requires surface and upper-air meteorological observations as well as certain surface characteristics (i.e., albedo, Bowen ratio, and surface roughness). These data are supplied through the "Meteorology" tab within DUSTRAN (see Section 4.8). In addition, lookup tables for typical values of albedo, Bowen ratio, and surface roughness are included in DUSTRAN. These values are based on various surface characteristics tables from Appendix A of the EPA's AERSURFACE User's Guide (EPA 2008).

Table 2.3 lists meteorological observations required by AERMET. Surface data are hourly observations, whereas upper-air vertical profiles are required less frequently, normally twice daily (00Z and 12Z). A single surface and upper-air station must be used. The input meteorological data are written to formatted surface and upper-air files by the DUSTRAN interface for use in AERMET.

Table 2.3. AERMET Meteorological Input Requirements

Surface Data (Hourly)	Upper-Air Data (Usually Twice Daily)
Wind Speed and Direction	Wind Speed and Direction
Temperature	Temperature
Relative Humidity	Pressure
Station Pressure	Measurement Height
Total Sky Cover	
Ceiling Height	
Measurement Height	

Since upper-air data can be difficult to obtain, profiling equations have been incorporated into DUSTRAN as an alternative to approximate upper-air soundings. Using the single observation data entered by the user, and certain parameters listed in the Cal.par file, DUSTRAN automatically generates an upper-air sounding file, which is used by the AERMET model. The sounding data consist of wind speed/direction, temperature, and pressure at several heights above the ground, where the lower and upper heights and the number of heights are specified in the Cal.par file (see Section A.2.1). The height spacing is logarithmic to allow narrower spacing close to the surface. The DUSTRAN profiling equations are described in the following sections.

2.1.1.1 Wind Speed/Direction Profile

For simplicity, the wind direction is assumed constant with height, and the wind speed is assumed to increase with height using the power-law relationship:

$$U_n = U_1 \left(\frac{Z_n}{Z_1} \right)^P \tag{2.1}$$

where U_n = wind speed at sounding height "n" (m/s),
U_1 = wind speed at lowest sounding height (m/s),
Z_n = sounding height "n" (m),
Z_1 = lowest sounding height (m),
P = power-law exponent depending on atmospheric stability.

The power-law exponents in Equation 2.1 follow from Table 4.6 of Turner (1994) and are listed in the Cal.par file (Section A.2.1) for application of DUSTRAN in either "rural" or "urban" areas.

2.1.1.2 Temperature Profile

The temperature sounding is developed from temperature lapse rates specified as a function of stability in the Cal.par file and the surface temperature specified in the user-input window. Consequently, the temperature sounding is determined as:

$$T_n = T_1 + T_{LR}\left(Z_n - Z_1\right) \tag{2.2}$$

where T_n = temperature at sounding height "n" (K),
T_1 = temperature at lowest sounding height (K),
T_{LR} = temperature lapse rate depending on stability (K/m)
Z_n = "n" sounding height (m)
Z_1 = lowest sounding height (m).

2.1.1.3 Pressure Profile

The atmospheric pressure as a function of sounding height is determined from the hydrostatic relationship as:

$$P_n = P_1 \ EXP\left[-\frac{a(Z_n - Z_1)}{(T_n + T_1)/2}\right] \tag{2.3}$$

where P_n = pressure at sounding height "n" (mb),
P_1 = pressure at lowest sounding height (mb),
a = 0.0342 K/m.

2.1.2 AERMAP

AERMAP (EPA 2011) is a terrain preprocessor that uses publicly available terrain data to extract elevations for sources and receptors as well as calculate the receptor "hill-height scale" for AERMOD dispersion calculations. DUSTRAN utilizes the U.S. Geological Survey's (USGS) Global Multi-resolution Terrain Elevation Data 2010 (GMTED2010) data files for use within AERMAP. These files provide global coverage at a resolution around 7.5 arc seconds (about 150 m). The GMTED2010 terrain files are clipped at site creation within DUSTRAN's "Add Site" utility, resulting in a smaller terrain file for use within a given AERMAP domain. This eliminates the need for users to define or supply multiple DEM files, which may not completely cover the AERMOD model domain.

2.1.3 AERMOD

AERMOD is the EPA's preferred regulatory dispersion model (see 40 CFR Part 51, Appendix W) for most modeling applications. The model is useful or simulating short-range dispersion (<50 km) from discrete source-type configurations (e.g., point, line, and area sources). Within DUSTRAN, area and line sources are integrated using AERMOD to simulate particle dispersion and deposition from paved or unpaved roadways due to various vehicle types.

AERMOD is a steady-state plume model; meteorological conditions can be updated hourly and are assumed to be spatially homogenous across the entire model domain. In the stable boundary layer, the plume concentration is assumed to be Gaussian. In the convective boundary layer, the horizontal plume concentration is assumed to be Gaussian, but the vertical plume concentration is described by a bi-Gaussian probability density function. The model calculates average plume concentration and deposition-flux values at defined receptor locations. The receptor field is automatically defined and created by DUSTRAN based upon the model domain size and source input configuration.

The AERMOD input file (Aermod.inp) defines the "Control," "Source," "Meteorology," "Receptor," and "Output" options for a given run. Every site in DUSTRAN has a "StaticData" directory that stores a template Aermod.inp to be used for that site. The template file is merged with user input from the DUSTRAN interface before running the model. Unlike CALPUFF (see Section 2.2.2), the AERMOD input file does not contain multiple options for controlling transport and diffusion; rather, a single, regulatory method is implemented within the model. A complete description of AERMOD's scientific basis is described in the model formulation's document (EPA 2004a).

Two versions of AERMOD have been implemented within DUSTRAN—"Source Emission – AERMOD" and "Source Emission – AERMOD (DRI)." The former version is the EPA's regulatory version of AERMOD (EPA 2014); the latter version has been modified to implement a dry-deposition algorithm based on experimental data from DRI.

In both versions of AERMOD, the dry-deposition flux, F_d, is calculated as the product of the concentration, χ_d, and a dry-deposition velocity, V_d, computed at a reference height, z_r:

$$F_d = \chi_d \cdot V_d \tag{2.4}$$

where F_d = the dry-deposition flux onto the ground (µg/m²/s),
χ_d = the air concentration (µg/m³) calculated at reference height z_r,
V_d = the particle dry-deposition velocity (m/s),
z_r = the dry-deposition reference height (m) $= z_o + 1$, and
z_o = the site surface roughness length from the meteorological file.

The two versions only differ in their method for estimating V_d. The following sections discuss how V_d is estimated in the EPA and DRI versions of AERMOD; all other model formulations are the same.

2.1.3.1 Source Emissions – AERMOD

The EPA version of AERMOD (i.e., "Source Emissions – AERMOD) (EPA 2014) computes a particle dry-deposition velocity to estimate the dry-deposition flux of particles onto the ground. The dry-deposition velocity is estimated using a resistance model, whereby the atmosphere is treated as a series of resistance layers to the depositing particles. The method is analogous to electrical resistance and is expressed as the inverse sum of a series of resistance layers near the ground plus a gravitational settling velocity (EPA 2004d):

$$V_d = \frac{1}{R_a + R_b + R_a R_b V_g} + V_g \tag{2.5}$$

where V_d = the dry-deposition velocity (m/s),
R_a = the aerodynamic layer resistance (s/m),
R_b = the quasi-laminar layer resistance (s/m), and
V_g = the gravitational settling velocity (m/s).

The aerodynamic layer resistance (R_a) occurs in the shallow layer (~10 m) next to the ground and depends on several meteorological parameters (e.g., wind speed, atmospheric stability, and surface roughness); the more turbulent the atmosphere, the smaller the aerodynamic resistance. The quasi-laminar layer resistance occurs in the thin, non-turbulent layer just above the depositing surface and depends on molecular, rather than turbulent, properties. EPA (2004d) summarizes formulations for R_a and R_b; these formulations are used in the EPA's version of AERMOD to estimate dry-deposition velocities for various particle sizes.

2.1.3.2 Source Emissions – AERMOD (DRI)

The DRI version of AERMOD (i.e., "Source Emissions – AERMOD (DRI)) also computes a particle dry-deposition velocity to estimate the dry-deposition flux of particles onto the ground. The dry-deposition velocity is estimated from an empirical function derived from DRI experimental data. DRI collected data over five different surface types (i.e., short grass, long grass, steppe, sage, and bare) with five different surface roughness lengths (Table 2.4). For each surface type, DRI provided a mean deposition velocity with its standard deviation in six size bins centered in a range from approximately 0.7 to 18 µm. The roughness values for each measurement site ranged from approximately 0.01 to 0.20 m. Further, the minimum deposition velocity did not drop below 1 cm/s. Therefore, in the curve fitting described below, 1 cm/s was adopted as a lower bound for deposition velocity.

Table 2.4. Data used to calculate functional form for deposition.

Size Bin Range (um)	0.7-1	1-2	2-2.5	2.5-5	5-10	>10	Avg Z0 (m)
Dp (um)	0.837	1.414	2.236	3.536	7.071	15.166	
Vd Short grass (cm/s)	1.78	1.65	1.21	2.29	4.27	9.63	0.026
Vd Long Grass (cm/s)	3.49	3.66	5.76	5.47	7.08	10.95	0.206
Vd Sagebrush (cm/s)	3.84	4.06	4.75	4.80	5.39	8.51	0.188
Vd Steppe grass (cm/s)	1.49	1.81	1.78	2.51	3.98	4.76	0.039
Vd No Vegetation (cm/s)	1.07	1.59	1.60	2.14	2.38	3.61	0.012
Size Bin Range (um)	0.7-1	1-2.5	2.5-5	5--10	10--15	>15	
Dp (um)	0.837	1.581	3.536	7.071	12.247	18.574	

Visual inspection of the data suggested that while a linear fit in surface roughness was sufficient, some curvature was suggested in the deposition velocity dependence on particle aerodynamic diameter. Because the derived function is entirely empirical, a curve-fitting routine was applied to the general quadratic function:

$$V_d(D_p, Z_0) = aZ_o^2 + bD_pZ_0 + cD_p^2 + dZ_0 + eD_p + f \quad (2.6)$$

where V_d = the dry-deposition velocity (cm/s),
D_p = the aerodynamic particle diameter (µm),
Z_0 = the surface roughness length (m),
$a, b, c, d, e,$ and f are determined by regression.

An initial fit using this general quadratic form yielded large uncertainty and near-zero values for coefficients a and c, while explaining approximately 80 percent of the variance. The terms including a and c were dropped, and the fit was repeated with:

$$V_d(D_p, Z_0) = bD_pZ_0 + dZ_0 + eD_p + f \quad (2.7)$$

The fit using this form also explained approximately 80 percent of the variance, confirming that the dropped terms were not useful in the fit.

On further inspection, a datum for deposition velocity at 15 µm over the bare surface appeared to be anomalously large. Dropping this point reduced the number of observations available from 30 to 29 and increased the variance explained to 93 percent. This fit was adopted for use in DUSTRAN. The resulting coefficients are:

b = 1.43
d = 12.4
e = 0.128

f = 1.16

These coefficients and the above function apply in the region $\{0 \le D_p < 18, 0 \le Z_0 < 0.21\}$. Outside this fitted domain, the following forms are used:

$\{0 \le D_p < 18,\ Z_0 \ge 0.21\}$ $\qquad V_d = V_d(D_p, 0.21) = bD_p \times 0.21 + d \times 0.21 + eD_p + f$ (2.8)

$\{D_p \ge 18, 0 \le Z_0 < 0.21\}$ $\qquad V_d = V_d(18,\ Z_0) = b \times 18 \times Z_0 + dZ_0 + e \times 18 + f$ (2.9)

$\{D_p \ge 18,\ Z_0 \ge 0.21\}$ $\qquad V_d = V_d(18, 0.21) = 11.5$ (2.10)

2.2 CALPUFF/CALGRID Modeling Systems

The CALPUFF modeling system (Scire et al., 2000a) is a non-steady-state puff dispersion model used to model long-range (>50 km) dispersion and particle deposition from active-source emissions (i.e., point, area, or line). The CALGRID modeling system (Scire et al., 1989) is an Eulerian dispersion model used to model wind-blown dust emissions, where the entire model domain is a potential emission source. Both dispersion models utilize the same preprocessor components (e.g., CALMET and CALPOST). The following sections describe the models, their preprocessors, and how they are implemented within DUSTRAN.

2.2.1 CALMET

CALMET is a meteorological model that generates three-dimensional gridded wind fields and two-dimensional fields of boundary-layer parameters for the CALPUFF and CALGRID dispersion models. CALMET is a diagnostic meteorological model and therefore requires surface and upper-air observations to generate the gridded fields. These data are supplied through the “Meteorology” tab within DUSTRAN (see Section 4.8).

Table 2.5 lists required meteorological observations used by CALMET. The surface data are hourly observations whereas the upper-air vertical profiles are required less frequently, normally twice daily (i.e., 00Z and 12Z). Multiple surface and upper-air stations may be used, and the stations are not required to be on the DUSTRAN domain, as CALMET interpolates the data to the domain. These input meteorological data are written to formatted surface and upper-air files by the DUSTRAN interface for use in CALMET.

Table 2.5. CALMET Meteorological Input Requirements in DUSTRAN

Surface Data (Hourly)	Upper-Air Data (Usually Twice Daily)
Wind Speed and Direction	Wind Speed and Direction
Temperature	Temperature
Cloud Cover	Pressure
Ceiling Height	Elevation
Surface Pressure	
Relative Humidity	

Geophysical data are also used by CALMET to derive the gridded meteorological fields. These data—terrain elevations and land use/land cover—are routinely available in datasets from the USGS with varying spatial resolution. In DUSTRAN, CALMET terrain data are supplied through GTOPO30 files, which are DEMs with a horizontal spacing of 30 arc seconds (approximately 1 km). Land-use/land-cover data are supplied through global land-cover characteristics (GLCC) files and are of similar resolution.

Preprocessing programs interface the geophysical datasets with the CALMET meteorological model. These preprocessors, shown in Figure 2.3, are implemented within DUSTRAN and automatically extract the required geophysical data based on the user's domain size. The extracted data are used in the CALMET model formulations and written to the CALMET output file for use in the CALPUFF and CALGRID dispersion models.

The procedures that CALMET uses to derive the gridded meteorological fields are controlled largely by an input file called "Calmet.inp." The input file is a text file with a series of keywords logically grouped based upon their overall function within CALMET. Every site in DUSTRAN has a "StaticData" directory that stores the template Calmet.inp to be used for that site. The template file is merged with user input from the DUSTRAN interface before running the model. The parameter settings within the template file are set to optimized values to produce the most realistic output (see Section A.2.2). Caution should be exercised if the user wishes to change any setting within the template file, as unrealistic results may be produced.

With the meteorological and geophysical input datasets defined, the following subsections provide an overview of the CALMET procedures for deriving gridded meteorological fields. Technical formulations are not provided here, as they are available in the CALMET User's Guide (Scire et al. 2000b); instead, the CALMET processing and creation of gridded meteorological fields are described qualitatively.

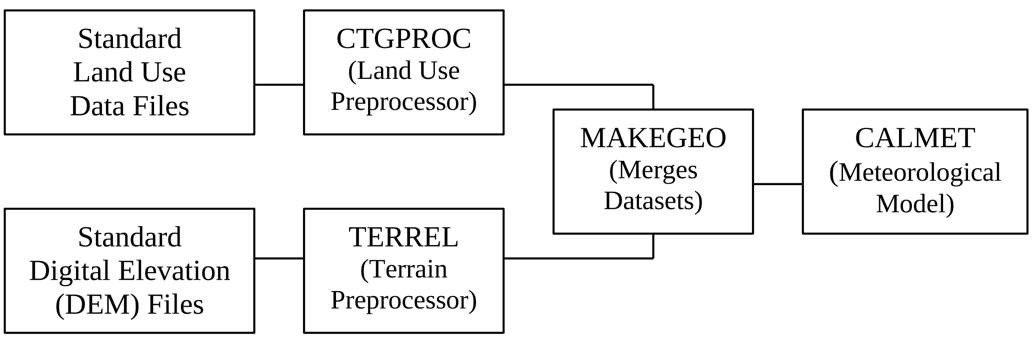


Figure 2.3. Data Flow and Geophysical Preprocessors for CALMET

2.2.1.1 CALMET-derived Wind Field

CALMET uses a two-step process to create the three-dimensional wind field for each hourly time step. In step one, an "initial guess" wind field is modified for terrain effects. In step two, surface and upper-air observations are merged objectively with the step-one, terrain-adjusted winds to create the final flow field. Each step is briefly discussed below.

Step-One Wind Field Formulation

The step-one wind field formulation begins with an "initial guess" wind field. The "initial guess" wind field can be a spatially varying or a constant, domain-mean wind used throughout the grid. In DUSTRAN's implementation of CALMET, the "initial guess" wind field is spatially varying and is based on surface and upper-air observations. The surface observations are extrapolated vertically using a power-law or Monin-Obukhov (M-O) similarity theory, assuming a neutral boundary layer, with M-O extrapolation used by default. The vertically extrapolated surface winds are then merged with the upper-air observations at each node on the grid using a $1/r^2$ interpolation. During the merging, a bias can be applied at each vertical level in the domain, whereby the relative weighting of the surface and upper-air data can be controlled. This level-by-level bias allows for surface data to more greatly influence the flow field in the lowest layers and the upper-air data to dominate the higher layers.

Once the "initial guess" wind field has been created, it is adjusted for terrain effects. CALMET has the option of adjusting the wind field for kinematic effects, slope flow, and flow blocking. Each option can be explicitly treated, and the cumulative effects are merged with the "initial guess" wind field to determine the step-one flow field.

1. Kinematic effects are calculated by assuming an initial zero vertical velocity in the "initial guess" wind field. The vertical velocity is then calculated because of topographic effects, and the horizontal velocity is adjusted using a divergence minimization scheme that iteratively adjusts the horizontal wind components until the three-dimensional divergence is less than a specified value.
2. Slope flow effects (e.g., upslope flows during the day and drainage flows at night) are based on an empirical scheme that is a function of the terrain slope, distance to the crest, and the sensible heat flux. A separate formulation for the sensible heat flux is used for the daytime and nighttime in CALMET and is performed for overland locations only.
3. Flow blocking, which is the result of stable stratification, is determined by calculating the Froude number for each grid node in CALMET. If a critical Froude number is not exceeded, then the flow is blocked by terrain and is adjusted tangent to the land feature (i.e., the flow is forced around the land feature).

Of the three terrain adjustment procedures, the kinematic effects can sometimes lead to unrealistically large horizontal velocities, particularly in complex terrain. Therefore, the kinematic adjustment is rarely implemented in DUSTRAN.

Step-Two Wind Field Formulation

The step-two formulation is an objective merging of the terrain-adjusted, step-one wind field with surface and upper-air observations. The objective analysis is performed level by level by first extrapolating the surface wind observations vertically using a constant power law or M-O as a function of stability. Then, for a given level, observations within a specified radius are weighted equally with the step-one wind field. All other observations at that level have a $1/r^2$ weighting out to a specified radius of inclusion. The radii for equal weighting and inclusion can be specified separately for the surface and all other vertical levels.

Each level of the merged wind field is then smoothed, and the divergence at each grid cell is calculated to provide a new estimate of the vertical velocity. The vertical velocity for the top level of the domain can be set to zero (called the O'Brian adjustment procedure), and the horizontal wind components are readjusted to be mass consistent with the new vertical velocity field using a divergence minimization procedure. The resulting wind field is the final wind field output by CALMET for use in the CALPUFF or CALGRID dispersion calculations.

2.2.1.2 CALMET-Derived Boundary-Layer Parameters

CALMET contains a micrometeorological model based upon an energy balance method, whereby the sensible heat flux is calculated at each grid node by parameterizing the unknown terms—latent heat flux, anthropogenic heat flux, ground storage/soil heat flux, and net radiation—in the surface energy balance equation. Once the sensible heat flux is calculated, gridded fields of other boundary-layer parameters that are functionally dependent on the sensible heat flux, such as the M-O length and surface-friction velocity, are computed.

CALMET has various formulations for calculating the mixing height based on time of day and stability classification. For unstable daytime conditions, the mixing height is thermally driven, and so it is a function of the surface heat flux and the vertical temperature profile from upper-air soundings. For stable nighttime conditions, the mixing height is mechanically driven and, thus, functionally dependent upon the friction velocity.

Because of the explicit use of the surface energy balance method in CALMET, all DUSTRAN simulation start times must be before sunrise. CALMET contains time-validation routines that mandate a start time of 5 a.m. local time, or earlier. So, if a noon-time or evening release is desired, the simulation start time must begin by 5 a.m. even though the source release time may not occur until much later in the day. Normally, this is of little consequence, as the model runtime is extremely fast and efficient.

2.2.1.3 Meteorological Data Input Options

DUSTRAN allows the user to specify four sources of meteorological data to be used by CALMET (see Section 4.8). The four options include the following:

1. Available Data. Use available site-specific meteorological data where data format is known by DUSTRAN and data-ingest utilities are available in DUSTRAN. Currently, this feature applies to data from the DOE's Hanford site meteorological network and data from the DoD's Fort Irwin site meteorological network.
2. Single Observation. Use single-point meteorological observations specified in DUSTRAN through an input window. DUSTRAN creates one "surface" data file and one "upper-air" data file from the user input in the format needed by CALMET. The "dummy" stations are located at the center of the modeling domain and are assumed to persist for the duration of the simulation.
3. User Defined. Use surface and upper-air meteorological data files (surf.dat and up_1.dat, up_2.dat,… up_n.dat) that have already been prepared for being directly read by CALMET. These files were created outside of DUSTRAN using CALMET utilities.
4. National Oceanic & Atmospheric Administration (NOAA) Archived. Use meteorological data archived from web-site-accessible National Weather Service (NWS) surface and upper-air data stations.

Of the four methods, option two, "Single Observation," is the only method that relies on user input for defining basic meteorological conditions. These inputs are then used by DUSTRAN to construct all required inputs for use in the CALMET model. The other three options use actual data streams coming from defined sources. The methodology used within DUSTRAN to construct the necessary meteorological inputs for CALMET when using "Single Observation" is described in the following section.

Single Observation Methodology

The Single Observation option provides the user with a very easy and convenient way to quickly view the effects of various configurations (e.g., multiple sources, long-rang transport, and nighttime stable flows) on resulting concentration and deposition fields. Even though the single-point meteorological observation persists and is used for the entire simulation, the model-derived meteorological grids will still vary spatially and temporally, as they are a function of land use, topography, and the surface-sensible heat flux (i.e., time of day). CALMET contains a solar model for use in determining sensible heat flux (which drives diffusion rates and mixing height growth) as a function of time.

The user specifies wind speed, wind direction, mixing height, ambient temperature, relative humidity, ambient pressure, and atmospheric stability through the DUSTRAN meteorological input window (see Section 4.8.2). CALMET also needs other surface quantities for completeness (i.e., ceiling height, opaque sky cover, and precipitation code), which are specified near the start of the Cal.par DUSTRAN setup file (see Section A.2.1). The Cal.par is a static text file used to initialize certain parameters in DUSTRAN. The file can be edited in a standard text editor; however, because it allows many features of DUSTRAN to be controlled, caution should be exercised if modifications to this file are desired. The default values in cal.par for ceiling height, opaque sky cover, and precipitation code are 100 (units are hundreds of feet), 0 (units are tenths of coverage), and 0 (no precipitation), respectively. The default values for the meteorological variables specified through the DUSTRAN user-input window are also given in the Cal.par file.

CALMET requires at least one upper-air sounding for operation. Using the single observation data entered by the user and parameters listed in the Cal.par file, DUSTRAN automatically generates an upper-air sounding file, which is used by the CALMET model. The sounding data consist of pressure, temperature, wind speed, and wind direction at several heights. Section 2.1.1 discusses the profiling methods used for AERMET; these same methods are used for generating vertical profiles for CALMET.

2.2.2 CALPUFF

The CALPUFF dispersion model is ideal for simulating long-range dispersion (>50 km) from discrete source-type configurations (e.g., point, line, and area sources). The latter two sources—area and line—are integrated with a dust-emissions model and can simulate particle dispersion and deposition from paved or unpaved roadways due to various vehicle types.

As the name implies, CALPUFF is a puff model; it transports and diffuses source material as a series of discrete puffs using gridded meteorological fields from CALMET. The model calculates average plume concentration and deposition-flux values at defined receptor locations. The receptor field is automatically defined and created by DUSTRAN based on the model domain size and source input configuration.

The use of spatially varying meteorological fields makes CALPUFF ideal for medium- and long-range transport applications where domain sizes often exceed 50 km and the assumption of "spatially homogenous" meteorology used in straight-line plume models often fails. As a result, CALPUFF has gained widespread acceptance and has been approved by the EPA as a regulatory model (see 40 CFR Part 51, Appendix W) for applications involving long-range transport. Domain sizes in DUSTRAN can be up to 400 km; thus, CALPUFF is an appropriate selection for use in the modeling system.

The procedures that CALPUFF uses to define plume transport and dispersion are controlled largely by an input file called "Calpuff.inp." The input file is a text file with a series of keywords that are logically grouped based upon their overall function within CALPUFF. Every site in DUSTRAN has a "StaticData" directory that stores the template Calpuff.inp to be used for that site. The template file is merged with user input from the DUSTRAN interface before running the model. The parameter settings within the template file are set to optimized values to produce the most realistic output (see Section A.2.3).

Two versions of CALPUFF have been implemented within DUSTRAN—"Source Emission – CALPUFF" and "Source Emission – CALPUFF (DRI)." The former version is the EPA's regulatory version of CALPUFF (Scire 2000a) and the latter version has been modified to implement a dry-deposition algorithm based on experimental data from the DRI. The dry-deposition formulations in CALPUFF are the same as those described for the AERMOD dispersion model (see Section 2.1.3).

The sections that follow review some of the more important parameters used by CALPUFF to control puff transport and dispersion. Recommendations are made for the various parameter settings based on experience, guidance documents (e.g., Irwin 1998), and CALPUFF's specific implementation within the DUSTRAN system. Users should exercise extreme caution if changing any setting within the template file, as unrealistic results may be produced.

2.2.2.1 Near-Field Release Approximation

Puff models are often computationally expensive when used for near-field applications involving continuous releases because the puffs are still relatively small, and so enough puffs must be released to approximate the source. In addition, sampling problems may arise near the source if too few puffs are released in a given time step, especially during rapidly varying meteorological conditions. To address these issues, CALPUFF can use an elongated puff, called a slug, to approximate the release. As the slug is transported downwind and its crosswind dimensions become larger because of dispersion, CALPUFF can transition the slug back to a puff. The slug method is an input parameter set in the CALPUFF input file and is recommended in most DUSTRAN applications.

2.2.2.2 Dispersion Coefficients

CALPUFF is a Gaussian model and therefore approximates atmospheric diffusion through the specification of dispersion coefficients. The dispersion coefficients are a function of atmospheric stability and affect the vertical and lateral growth of a puff as it is transported downwind. CALPUFF provides many methods for defining the dispersion coefficients, including the following:

- direct measure of the horizontal and vertical velocity variances
- similarity theory formulations
- Pasquill-Gifford-Turner (PGT) specifications.

Of the listed methods, the similarity theory formulations are recommended and implemented in DUSTRAN, as the parameters used in their formulation are explicitly calculated by CALMET.

2.2.2.3 Plume Rise

CALPUFF can account for plume rise, especially from point sources, which are often used to approximate releases from stacks. With stack-type releases, plume buoyancy (due to increased exhaust temperature) and momentum (from exhaust flows) can loft plumes into the air. Plume lofting can result in a phenomenon called partial plume penetration, whereby part of the plume is ejected into a stable layer (called an inversion) above the release. The overall effect of these parameters is to increase the release height and remove material from the initial plume, all of which act to reduce surface concentration and deposition-flux values downwind of the release, particularly near the source. Because DUSTRAN domain sizes tend to be large (e.g., >50 km), these effects play a smaller role and only act to increase computation time. They are not recommended for use unless near-field effects are of concern.

2.2.2.4 Receptor Grids

Receptors are locations where the model performs concentration and deposition calculations. In CALPUFF, a primary receptor grid is used for calculating values across the entire domain. The grid is Cartesian and has uniformly spaced receptors in the X and Y directions. By default, 50 receptors are specified for both directions, so a 100-km domain, for example, has a receptor spacing of 2 km in the X and Y directions. The number of receptors in the primary grid can be changed within the Cal.par file (see Appendix A.2.1).

In DUSTRAN, secondary receptor grids, or sub-grids, are automatically generated in and around sources to increase the resolution of the calculated concentration and/or deposition fields very near the source. These sub-grids are treated as discrete receptors in CALPUFF, and up to 4,000 discrete receptors are allowed. For each point source, a polar receptor sub-grid is used. For each area source, a rectangular receptor sub-grid is used. No sub-grid is currently implemented for a line source. The size and resolution of the receptor sub-grids are defined according to parameters set within the Cal.par file (see Section A.2.1).

Figure 2.4 presents an example of the various receptor grids implemented in DUSTRAN for a CALPUFF model simulation. Receptors are displayed as blue dots, with the primary Cartesian grid spaced uniformly across the domain and a polar and a rectangular sub-grid centered over their respective source types.

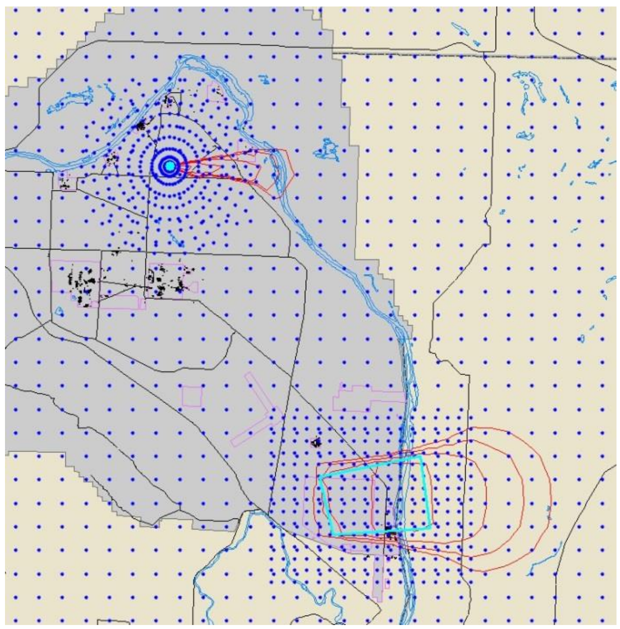


Figure 2.4. Example of the Primary Cartesian Receptor Grid and a Polar and Cartesian Sub-Grid Used Within CALPUFF

2.2.2.5 Representing Moving Vehicles as Line and Area Dust Sources

DUSTRAN does not treat the motion of individual vehicles, but rather takes a “bulk” approach to dust emissions from vehicle activities. That is, dust emissions from all vehicles active on a road segment or within a training area over a specified time are assumed to be released uniformly from the road or area at a constant rate throughout the duration of the activity. Therefore, the input fields on the DUSTRAN “Vehicle Parameters” window should not be interpreted as representing the specific motion of individual vehicles, but rather as a convenient approach for providing the vehicle information needed by DUSTRAN. Details of specifying the vehicle characteristics and activities for road segments and for training areas are given in Sections 4.6.2.2 and 4.6.3.2, respectively.

As described in Section 2.3.1, dust emissions from a moving vehicle are proportional to the vehicle momentum (i.e., vehicle weight × vehicle speed). Therefore, if some vehicles of one type travel at significantly different speeds than other vehicles of the same type, another vehicle type will need to be added to the DUSTRAN vehicle list such that the other speed(s) can be specified. Additional vehicles can be specified in DUSTRAN by editing the Cal.par file (see Section A.2.1).

2.2.3 CALGRID

The CALGRID dispersion model has been implemented within DUSTRAN to simulate the dispersion of wind-generated dust; it is activated by setting the "Simulation Type" to "Wind-blown Dust" within the DUSTRAN interface (See Section 4.4). In the wind-blown dust mode, DUSTRAN creates gridded dust-emission factors for the entire model domain, which are then supplied to the CALGRID model to simulate the downwind dispersion and deposition. The dust-emission factors calculated by DUSTRAN are a function of wind stress, soil texture, and vegetation type across the domain and are discussed further in Section 2.3.2.

CALGRID is an Eulerian model and uses mass continuity to track material throughout a gridded volume. In DUSTRAN, the volume boundaries are defined by specifying a domain in which the user would like to simulate wind-blown dust dispersion. The amount of dust in a given volume is the sum of dust being generated by the wind or lost by deposition as well as the transfer of dust between volumes through wind transport and atmospheric diffusion. The gridded nature of the model makes it ideal for examining releases from large areas (e.g., wind-blown dust over a large domain).

The CALMET-derived spatially and temporally varying meteorological fields are used in CALGRID to transport and diffuse material throughout the domain. Horizontal transport requires the two-dimensional gridded fields of the velocity components (U and V) for each vertical layer. Terrain-following vertical velocities are used to determine the vertical transport through each of the vertical cell faces in CALGRID. Horizontal diffusion is a function of the CALMET-gridded PGT stability classification, modified for wind speed within each cell and distortion or shear between horizontal cells. Vertical diffusion is calculated from CALMET-gridded similarity fields and is functionally dependent upon the height above ground and stability.

Emissions are introduced into the CALGRID domain depending on the source type. For area sources, which include the model domain for wind-blown dust simulations, emissions are injected into CALGRID using emission layers, with each layer containing a fraction of the total emissions. In DUSTRAN's implementation of CALGRID, area sources have one emission layer, bounded between the surface and 20 meters. For other source types, such as point sources, material is injected into one or more CALGRID layers based on the height of the stack, plume rise due to buoyancy and momentum, and the plume overlap with the model layers.

The procedures that CALGRID uses to define plume transport and dispersion are controlled largely by an input file called "Calgrid.inp." The input file is a text file with a series of keywords that are logically grouped based upon their overall function within CALGRID. Every site in DUSTRAN has a "StaticData" directory that stores the template Calgrid.inp to be used for that site. The template file is merged with user input from the DUSTRAN interface before running the model. The parameter settings within the template file are set to optimized values to produce the most realistic output (see Section A.2.4). Users should exercise extreme caution if changing any setting within the template file, as unrealistic results may be produced.

2.2.3.1 Receptor Grid

In CALGRID, the primary receptor grid is Cartesian and has uniformly spaced nodes in the X and Y directions. The nodes serve to both define the horizontal extent of a given cell and specify receptor locations where concentration and deposition values are calculated. Because CALGRID is an Eulerian model, inherent problems exist for situations when the horizontal grid cell size is small and the wind speed is large, as material may be transported through more than one grid cell in a single time step. To minimize this possible issue, 20 nodes in the X and Y directions are recommended and are set as the default. For example, the cell size (and receptor spacing) for a 100-km grid should be 5 km. The number of nodes in the primary grid can be changed within the Cal.par file (see Section A.2.1).

It should be noted that the outer band of grid cells in CALGRID serve to initialize the inner grid cells within the domain. These cells are considered boundary cells and serve as storage locations for the lateral boundary conditions of the grid; no calculations (e.g., transport, diffusion, or deposition) are performed within these cells, and so no values are available for contouring. Therefore, the number of receptors in the X and Y directions available for contouring will always be two less than the actual number of nodes.

2.2.4 CALPOST

CALPOST is designed to interface with and summarize the output from the CALPUFF or CALGRID models. In DUSTRAN, the CALPOST post-processing module is used to create user-specified time-averaged values from standard hourly outputs generated by the models. In addition, CALPOST is used to create Top 50 tables, which are tabular values of the highest 50 concentration and deposition values during a simulation for the averaging period of interest. The averaging periods are set within the DUSTRAN interface (i.e., currently 1-, 3-, 8-, and 24-hour averages are available) and calculated for the length of the run.

2.3 Dust-Emission Modules

Dust is injected into the atmosphere through active and natural processes. Active processes primarily involve human activity that directly disturbs the surface (e.g., vehicle activity on dirt roads and other unpaved areas or from re-suspension of loose material covering paved roads). Natural processes include wind erosion, which occurs primarily in arid or semiarid environments and may be enhanced by soil disturbance following recent human activity or following natural disasters (e.g., range fires). The dust-emission modules incorporated into DUSTRAN account for both vehicular and wind-blown dust-generation processes.

2.3.1 Emission by Vehicular Activity

The vehicular dust-emission module represents dust emissions as the product of an empirically formulated emission factor and the vehicle activity, the latter taken as the total vehicle distance traveled (summed if there are multiple vehicles) in a given period of interest. Explicitly, it can be written as

$$F_j = E_j \cdot A \tag{2.11}$$

where
- Fj = dust emission due to vehicle activity for particulate size class j [g]
- Ej = emission factor for particulate size class j [g/VKT]
- A = vehicle activity [VKT]
- VKT = vehicle kilometers traveled.

The relations used to determine E_j are entirely empirical and are usually available for only some of the standard particulate size classes (e.g., $PM_{2.5}$, PM_{10}, PM_{15}, and PM_{30}). Variables on which various authors have expressed an E_j dependency include the silt content of the surface, the number of vehicle axles, vehicle weight, vehicle speed, and soil moisture. The emission factor, E_j, is determined as a product of some combination of these variables, each raised to an empirically determined power and a fitted constant. The paved and unpaved road-emission factors in EPA's AP-42 (EPA 2005) are based on this approach and are available for use in DUSTRAN.

Emission factors have been measured for specific vehicles or classes of vehicles. The particulate emission factors for wheeled and tracked military vehicles used in DUSTRAN were provided through SERDP research projects. In observations carried out using a variety of wheeled vehicles (primarily military) at Ft. Bliss, Texas, Gillies et al. (2005a, 2005b) found that the only two variables that matter significantly in calculating the PM_{10} emission factor for unpaved roads are vehicle weight and vehicle speed. Moreover, when weight and speed are properly accounted for, a single empirically derived functional form may be used to calculate a vehicle-specific emission factor. This function may be expressed using

$$E_{PM10} = 0.003 \cdot W \cdot S \tag{2.12}$$

where W = vehicle weight (kg)
S = mean vehicle speed (km/h).

Combining Equations 2.11 and 2.12 and summing over the types of wheeled vehicles operating on an unpaved road of length, L, during time period, T, gives the total emission from the road for that time period as:

$$F_{PM10} = 0.003 \cdot \sum_{i=1}^{k} W_i \cdot S_i \cdot A_i \tag{2.13}$$

where i = vehicle type (e.g., Humvee; specifies vehicle weight)
k = total number of vehicle types
A_i = $L \cdot N_i$
N_i = total number of vehicles of type i.

Similarly, using results from Kuhn et al. (2010), the total emission from an unpaved road from tracked military vehicles for a given time period can be expressed as:

$$F_{PM10} = 0.0014 \sum_{i=1}^{k} W_i \cdot S_i \cdot A_i \tag{2.14}$$

where *F, W, S, A, i* and *k* are defined as above.

The DUSTRAN vehicle-activity dust-emission module produces total emissions for each road segment over the time period, T. Vehicular dust emissions are then passed to the CALPUFF dispersion model where they are released into the modeling domain uniformly along each road segment in both space and time for the duration of the activity.

The vehicle emissions module requires the Universal Transverse Mercator (UTM) easting and northing coordinates to describe the starting and ending points of each road segment as well as the activity duration. Within DUSTRAN, the roadways are created graphically by drawing each segment within MapWindow. For a given line segment, DUSTRAN prompts the user to enter the weight and mean speed for various vehicle types traveling on the roadway. For paved surfaces, emission factors are based on EPA AP-42 recommended values (EPA 2005) and are available for $PM_{2.5}$, PM_{10}, PM_{15}, and PM_{30}. For unpaved road surfaces, the user has the option of specifying whether to use emission factors derived from

EPA AP-42 (EPA 2005) or from Gillies et al. (2005a, 2005b) and Kuhn et al. (2010). Because the Gillies et al. (2005a, 2005b) and Kuhn et al. (2010) work is specifically for E_{PM10}, emission factors for other size classes under this option are estimated by computing the ratio of Gillies et al. E_{PM10} to EPA AP-42 E_{PM10} and applying this ratio to values for EPA AP-42 unpaved road $E_{PM2.5}$ and E_{PM30} particle class sizes. As of this writing, AP-42 does not include recommendations for unpaved road PM_{15} emission factors; PM_{15} emissions under both options are thus estimated via a linear interpolation between PM_{10} and PM_{30} emissions. Average fleet weight and average fleet speed, required for AP-42 formulations, are calculated automatically from information input by the user in the DUSTRAN interface.

During military training exercises, off-road activities can occur within specific training areas where numerous vehicles can move around the area (both on- and off-road) during a period of time where the specific paths of the vehicles are not known. DUSTRAN treats this area-wide training activity as an area source (see Section 4.6.3.1). The total area-wide dust emissions for each particle size range during the period of the training are determined using the same method as for roads described above. Knowing the total distance traveled by each vehicle type during the training period, the total dust emissions for each particle size range are determined using the emission factors described above for unpaved roads. No distinction is made between dust emissions from vehicles operating on unpaved roads versus vehicles operating during off-road maneuvers. At this time, dust-emissions factors for off-road activities are not available; however, emission factors for unpaved roads should be a reasonable surrogate for off-road vehicular activities.

2.3.2 Windblown Dust

The windblown dust formulation in DUSTRAN provides a measure of the dust emission from the modeling domain caused by wind erosion of the surface. These emissions are a function of the surface wind stress, vegetation class, and soil texture across the modeling domain. The surface wind stress, as approximated by the friction velocity, is calculated as a function of time and location from the CALMET meteorological model. Vegetation class and soil texture coverage come from well-established global databases and are discussed further in Sections 2.3.2.3 and 2.3.2.4, respectively. An evaluation of the wind-erosion model described in this section has been reported by Shaw et al. (2008).

The “Add Site Wizard” (see Section 7.0) in DUSTRAN automatically creates vegetation class and soil texture files for use in a wind-blown dust simulation whenever a new site is created. These characteristic files, which are a subset of the original global datasets, can be used for any domain specified within a site and are the default files used for generating dust emissions in a wind-blown dust simulation. In addition, the user has the option of specifying finer-resolution characteristic files (see Section 4.6.9) that can be ranked by order of use in a given simulation. When used in this way, high-resolution files can provide detailed information in user-specific regions within the domain and the default files provide information where there is no user-specified information. High-resolution files can be created using the “Polygon Layer Creator” (see Section 5.0), which is an easy way to build vegetation class and soil texture files or to explore the effects of vegetation removal or soil disruption (e.g., off-road vehicle traffic disturbing soils in new areas or field plowing) on dust emissions.

The general approach in DUSTRAN for computing PM_{10} concentrations resulting from wind-blown dust is to first calculate gridded fields of wind-generated dust emissions over the modeling domain for each model time step. The time- and space-varying dust emission is then provided to the CALGRID dispersion model, which uses winds from CALMET for transporting, dispersing, and depositing the emitted dust throughout the modeling domain. The wind-blown dust emissions for each model grid cell are calculated using the method given in Sections 2.3.2.1 through 2.3.2.4. The principal information needed to calculate the dust emissions for each model grid cell is the time-varying friction velocity (from

CALMET), the area-weighted average vegetation mask, and the area-weighted average fraction of total dust emissions by particle size category for each grid cell. Section 2.3.2.5 gives the method for calculating the fractional distribution of vegetation class and soil texture for each grid cell.

2.3.2.1 Dust Flux as a Function of Friction Velocity

Numerous authors over the past three decades have made laboratory and field measurements of dust flux from wind erosion and empirically related those measurements to the friction velocity, u_*, which is a measure of wind stress on the surface. Some efforts have been made to provide a theoretical foundation for the functional form of the flux in terms of friction velocity, but observations continue to have a great deal of scatter and do not yet validate particular theoretical results. The primary difference among published relations is whether the flux depends on u_* raised to the third or fourth power. Because of their field measurements of dust flux, G (g cm^{-2} s^{-1}) under a variety of conditions, DUSTRAN uses the formulation of Gillette and Passi (1988):

$$G = Cu_*^4\left(1 - \frac{u_{*_t}}{u_*}\right) \qquad (2.15)$$

where u_{*_t} (cm s^{-1}) is a threshold friction velocity below which dust emission does not to occur. In addition to the uncertainty in the exponent of u_*, there has also been significant experimental variation in the values of u_{*_t}.

Gillette and Passi did not actually publish values of C and u_{*_t} in the above relation. However, they did graphically present a variety of observations of G versus u_*. Data digitized from their graph was used to compute the root-mean-square (rms) difference between the function above and the data for a variety of combinations of C and u_{*_t}. Those results are shown as a contour plot in Figure 2.5, which shows a broad region over which the rms differences are not much different from the absolute minimum value that occurs near a fitted threshold friction velocity of about 33 cm s^{-1}. Compromise values of $C = 1.0\times10^{-14}$ g cm^{-6} s^{3} and $u_{*_t} = 20$ cm s^{-1} were selected, which place the threshold friction velocity below the lowest reported value of u_*, because some of the observations of Gillette and Passi showed dust flux occurring for values of $u_* < 30$ cm s^{-1}. Figure 2.6 shows the fit using PNNL's coefficients.

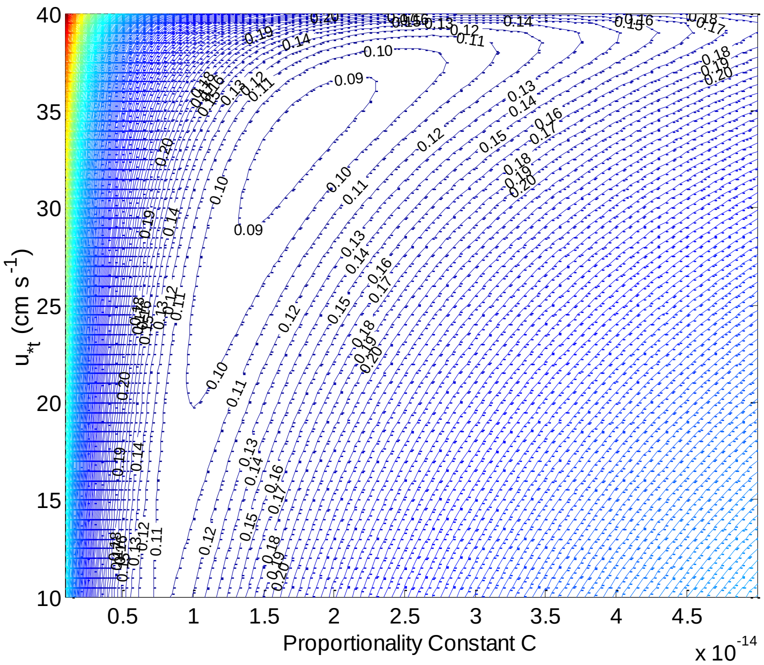


Figure 2.5. Contour Plot of RMS Differences Between Equation (2.15) and Observations of Dust Flux G Discussed by Gillette and Passi (1988)

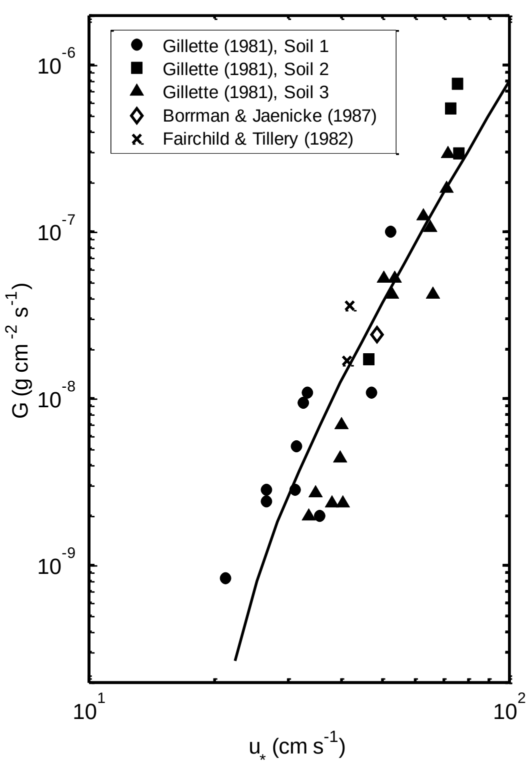


Figure 2.6. Observations of G Versus u* after Gillette and Passi (1988). The solid line is Equation (2.15) with C = 1.0 × 10^{-14} g cm^{-6} s^3 and u_{*t} = 20 cm s^{-1}.

2.3.2.2 Effect of Soil Moisture

Soil moisture, if a measure is available, is taken into account in our approach using the method of Fecan et al. (1999) as cited by Nickovic et al. (2001). Basically, soil moisture is incorporated through a wetness factor, f_w, that multiplies the threshold friction velocity to increase it. Consequently, the total dust flux from the surface accounting for soil moisture is

$$G = Cu_*^4\left(1-\frac{f_w u_{*t}}{u_*}\right) \tag{2.16}$$

where the soil wetness factor is given as

$$f_w = \begin{cases} \left[1+1.21(w-w')^{0.68}\right]^{1/2} & w > w' \\ 1 & w \le w' \end{cases} \tag{2.17}$$

w is the gravimetric soil moisture (i.e., mass of water/mass of soil; %), and w' is the maximum amount of water that can be adsorbed by the soil (%), given as a function of the fraction of clay in the soil, β_1 (see Section 2.3.2.4). w' is given as

$$w' = 14\beta_1^2 + 17\beta_1 \tag{2.18}$$

At this time, soil moisture, w, is not a function of time or location in DUSTRAN, but is currently specified as a constant value in the Cal.par model static file described in Section A.2.1.1. The default value is zero, leading to a wetness factor of one, which is dry soil.

2.3.2.3 Vegetation Cover Effects on Dust-Emitting Potential

Equation 2.8 gives the maximum wind-generated dust flux from the surface not accounting for the effects of different types of vegetation cover on dust-emitting potential. Essentially, more vegetation cover results in less wind-blown dust generated from the surface. This effect of vegetation cover on wind-blown dust is treated by simply multiplying the dust flux from Equation 2.16 by a vegetation mask, α, that ranges from zero to one. The total dust flux from the surface accounting for vegetation is then given as

$$G = \alpha Cu_*^4\left(1-\frac{f_w u_{*t}}{u_*}\right) \tag{2.19}$$

The vegetation mask is determined using the Olson World Ecosystem (Olson 1992) that defines 59 distinct classes of vegetation. Of the 59 classes, only 4 have been sufficiently exposed to soil to allow for wind erosion, and they include two desert categories and two semi-desert categories. Because of the sparseness of vegetation in deserts, α for those categories has a value of 1.0. Because of the more widespread presence of shrubs and grasses in the semi-desert categories, α is assigned a value of 0.5. Table 2.6 lists the Olson identification number, vegetation class description, and α for these four classes. All other Olson categories are assigned a value of zero for α and, therefore, do not contribute to wind-blown dust emissions.

Within DUSTRAN, the default Olson vegetation class file is derived from the original 10-minute resolution global database at the time of the sites creation (see the "Add Site Wizard" in Section 7.0).

The default dataset can be supplemented with higher-resolution, Olson-based vegetation class files created by the user using the "Polygon Layer Creator" (see Section 5.0).

Table 2.6. Olson Vegetation Classes Used in the Wind-blown Dust-Emission Model

ID #	Olson Vegetation Class Description	α
8	Desert, mostly bare stone, clay and sand	1.0
50	Sand desert, partly blowing dunes	1.0
51	Semi-desert/desert, scrub/sparse grass	0.5
52	Cool/cold shrub, semi-desert/steppe	0.5

The value of α actually used in Equation 2.19 to calculate the dust emissions for a model grid cell is the area-weighted average of α *'s* for all the vegetation categories that fall within the grid cell. The area-weighted average vegetation mask, $\bar{\alpha}$, is calculated as

$$\bar{\alpha} = \sum_{i=1}^{4} f_i^V \alpha_i \tag{2.20}$$

where α_1 is the vegetation mask for the ith Olson vegetation class (Table 2.6), and f_i^V is the area fraction of the ith Olson vegetation class in a grid cell. The area fractions for the four Olson vegetation classes will sum to one or less. The sum may be less that one because, as noted previously, many other Olson vegetation classes exist that have α *'s* equal to zero. See Section 2.3.2.5 for the determination of the area fractions.

2.3.2.4 Approximating the Size Distribution of Windblown Dust

The formulation given above estimates the total mass of dust produced by wind. However, it is also useful to know the size distribution and in particular how much of the dust consists of particles smaller than 10 µm in diameter. To do this, global databases of soil texture class have been used to estimate the fraction of the emitted dust in four separate particle size ranges. Soil textures are typically defined in terms of their fractions of clay, small silt, large silt, and sand (Tegen and Fung 1994). Table 2.7 gives typical properties of particles for each soil texture class.

Table 2.7. Features of Typical Dust Particles (after Nickovic et al. 2001)

k	Soil Texture Class	Range of Particle Diameters (µm)	Typical Particle Diameter (µm)	Particle Density (g cm^{-3})	γ_k
1	clay	1–2	1.5	2.50	0.08
2	small silt	2–20	12	2.65	1.00
3	large silt	20–50	36	2.65	1.00
4	sand	50–100	76	2.65	0.12

The approach for determining the soil texture class (and thus the particle size distribution) follows from Nickovic et al. (2001) using a Zobler soil categories database. Table 2.8 lists the fractions, $\beta_{j,k}$, of the four (k-index) soil texture classes within each of the seven (j-index) Zobler soil categories. Note that the fractions of soil texture classes for each Zobler category sum to one. In addition, note that the size fractions for small and large silt given by Nickovic et al. (2001) were too large by a factor of two. The values in Table 2.8 are corrected.

Table 2.8. Fractions, $\beta_{j,k}$, of the Soil Texture Classes in each Zobler Soil Category

		k=1	2	3	4
j	Zobler Soil Categories	Clay	Small Silt	Large Silt	Sand
1	coarse	0.12	0.04	0.04	0.80
2	medium	0.34	0.28	0.28	0.10
3	fine	0.45	0.15	0.15	0.25
4	coarse-medium	0.12	0.09	0.09	0.70
5	coarse-fine	0.40	0.05	0.05	0.50
6	medium-fine	0.34	0.18	0.18	0.30
7	coarse-medium-fine	0.22	0.09	0.09	0.60

Currently, the soil texture category for a desired location in the modeling domain is being read from an ASCII, comma-delimited text file that was derived from the Zobler raster image with a 1-degree resolution (Staub and Rosenzweig 1992). Sources for higher-resolution soil textures that are spatially complete (e.g., cover the continental U.S.) are being investigated for possible inclusion in a future version of DUSTRAN. In addition, the user has the option of creating high-resolution Zobler soil texture files using the "Polygon Layer Creator" (see Section 5.0). The user-specific files can be associated with a given simulation (see Section 4.6.9) to supplement or replace the soil textures derived from the default Zobler global file.
To accomplish the size partitioning of the dust flux from each model grid cell, DUSTRAN uses the dust-productivity factor as defined by Nickovic et al. (2001). For each of the four particle size classes, DUSTRAN defines a dust-productivity factor δ_k so that the dust flux in the kth particle size class, G_k, is

$$G_k = \delta_k G \tag{2.21}$$

where G is from Equation 2.19. The dust-productivity factor for a grid cell is determined by

$$\delta_k = \gamma_k \sum_{j=1}^{7} f_j^Z \beta_{\mathbf{j,k}} \tag{2.22}$$

where γ_k = the ratio of mass available for uplift to total mass in that soil texture class (size range)
$\beta_{j,k}$ = from Table 2.8
f_j^Z = the area fraction of the jth Zobler soil category in a grid cell.

The area fractions for the seven Zobler categories sum to one. See Section 2.3.2.5 for the determination of the area fractions. Table 2.7 lists the values of γ_k, which are those used by Nickovic et al. (2001). Because the values of δ_k represent a partitioning of the total flux, G, they should sum to unity. However, the values of G used by Gillette and Passi (1988) to develop Equation 2.15 were for particle sizes <40 μm in diameter. This essentially excludes the larger sand category. This exclusion of the larger sand category is accounted for by actually using an "enhanced" total dust flux, G', in Equation 2.21. Therefore, the "actual" dust flux for each particle size (soil texture class) within a grid cell is

$$G_k = \delta_k G' \tag{2.23}$$

where G' is determined as

$$G' = \frac{\sum_{k=1}^{4} \delta_k}{\sum_{k=1}^{3} \delta_k} G \qquad (2.24)$$

Substituting Equation 2.24 into 2.23 gives the actual dust flux by particle size category as

$$G_k = \delta_k \frac{\sum_{i=1}^{4} \delta_i}{\sum_{i=1}^{3} \delta_i} G \qquad (2.25)$$

Currently in DUSTRAN, only the PM_{10} size particles from wind-blown dust are provided as a gridded output. The emission flux of PM_{10} for each grid cell is the sum of the first two particle size categories, or

$$G_{PM10} = \sum_{k=1}^{2} G_k \qquad (2.26)$$

2.3.2.5 Calculating Soil and Vegetation Types for Use in Wind-Blown Dust Simulations

This section provides specific information on how the Olson vegetation class and Zobler soil-texture values are sampled and mapped to model grid cell average values in DUSTRAN. Mapping is necessary because the characteristic files provide vegetation and soil-texture information as a series of polygon shapes whereas CALGRID requires emissions from a regular array of cells within a Cartesian grid. This section is not necessary to understand how wind-blown dust is determined, but is given here for completeness of the calculations and operations performed within DUSTRAN.

For calculating wind-blown dust emissions, DUSTRAN requires the fractional area coverage of the four Olson vegetation classes and seven Zobler soil texture categories for all the model grid cells within the model domain. The fractional area coverage of either the vegetation or soil textures class for each model grid cell is calculated by “sampling” the respective base characteristic file. The default base characteristic files are created automatically at the time of the site’s creation (using the “Add Site Wizard,” Section 7.0) and are derived from the original Olson World Ecological and Zobler soil textures global database files. The default files are assigned automatically to a given wind-blown dust scenario using the “Sources” tab (see Section 4.6) within DUSTRAN and can be supplemented with higher-resolution, user-specific characteristic files created using the “Polygon Layer Creator” (see Section 5.0).

The format of the two base data files includes headers containing the number of data columns and rows found in the two site-wide files as well as headers detailing the UTM coordinates of the lower left and upper right corners of the site extent. Typically, site extents are greater than 600 km square to accommodate modeling domains centered at various locations and of various sizes. Following the headers are data records containing the soil or vegetation data. Each data record contains five fields separated by commas that represent one grid point in the base file. The first two fields of each record contain the column and row indices of the data’s location in the data file. The third element contains the soil or vegetation type ID. A 9999 in the type field represents a grid point that does not contain any soil or vegetation data. Lastly, the fourth and fifth elements are the UTM easting and northing coordinates of the grid point. The [0,0] indices in the file represent the X and Y values of the lower left (southwest) corner of the site extent. X indices increase to the right (east), and Y indices increase to the top (north).

The fractional area coverage, f_k^A, within a model grid cell for the k[th] category of surface property A (e.g., soil texture) calculates as

$$f_k^A = \frac{N_k}{N_G N_G} \tag{2.27}$$

where N_k is the number of occurrences of the k[th] category of property "A," and N_G is the number of sampling points in the x- and y-directions in a square model grid cell. Currently in DUSTRAN, N_G is equal to 5 for a total of 25 sampling points per model grid cell.

The desired property value for each model grid cell sampling point is read from the appropriate site-wide file containing the specific property (e.g., soil texture or vegetation cover) of interest. The property value at a grid cell point located at $[X_m,Y_n]$, where m = 1 to N_G and n = 1 to N_G, is determined by knowing the nearest site-wide data point to the sampling point. The indices $[i_m, j_n]$ of the nearest site-wide point to the [m,n] grid cell point are determined as

$$i_m = NINT\left[\frac{X_m - X_0}{X_d - X_0}\left(N_x - 1\right)\right] \tag{2.28}$$

$$j_n = NINT\left[\frac{Y_n - Y_0}{Y_d - Y_0}\left(N_y - 1\right)\right] \tag{2.29}$$

where
- i_m = 0 to N_x-1; X-direction index of m[th] grid cell sampling point in site-wide file
- j_n = 0 to N_y-1; Y direction index of n[th] grid cell sampling point in site-wide file
- X_m = UTM Easting coordinate of the model grid cell sampling point
- Y_n = UTM Northing coordinate of the model grid cell sampling point
- X_0 = UTM Easting coordinate of the southwest corner of the site-wide file
- Y_0 = UTM Northing coordinate of the southwest corner of the site-wide file
- X_d = UTM Easting coordinate of the northeast corner of the site-wide file
- Y_d = UTM Northing coordinate of the northeast corner of the site-wide file
- N_x = Number of data points in the X-direction in the site-wide file
- N_y = Number of data points in the Y direction in the site-wide file
- $NINT$ = get nearest integer of quantity in brackets.

The X_m and Y_n coordinates for each of the grid cell sampling points are determined as

$$X_m = X_0^G + \frac{\Delta}{2N_G}\left[1 + 2(m-1)\right] \tag{2.30}$$

$$Y_n = Y_0^G + \frac{\Delta}{2N_G}\left[1 + 2(n-1)\right] \tag{2.31}$$

where
- Δ = Length of square grid cell along one side
- X_0^G = UTM Easting coordinate of the model grid cell southwest corner
- Y_0^G = UTM Northing coordinate of the model grid cell southwest corner

As mentioned previously, the spatial resolution of the Olson vegetation class database is 10 minutes (roughly 15 km at mid-latitudes) and the Zobler soil texture database is 1 degree (roughly 80 km at mid-latitudes). In general, these data are at a much coarser resolution than the grid resolution that will be likely used in a windblown dust simulation. For example, a 200 km domain with 20 grid cells in the X and Y direction will have a 10 km resolution, which is a finer resolution than either the default vegetation class or soil texture database files. Nevertheless, the 25 (5 by 5) sampling points per grid cell method is not computationally expensive and is employed for situations where the user may supplement a simulation with higher-resolution characteristic files.

3.0 DUSTRAN Installation Instructions

DUSTRAN requires that both the Microsoft .NET 4.5 framework and the MapWindow GIS be installed on the system. The installation instructions below explain how to install these if they are not already present. These instructions will cover installing DUSTRAN and setting up access to DUSTRAN from within MapWindow.

3.1 Installing DUSTRAN

- DUSTRAN V2.0 runs within the MapWindow Version 4.7 application.
- Note that the installation of this software requires that the user has administrative privileges on the machine in which it is being installed.
- Before installing DUSTRAN, be sure all other applications are closed.
- If a prior version of DUSTRAN has been installed, it will need to be removed prior to installation of the new version.

3.2 Installing DUSTRAN and Setting Up Access to DUSTRAN from Within MapWindow.

1. Start the installation of the DUSTRAN modeling system by double-clicking on the “Dustran_Installer.exe” executable file.
2. If the Microsoft .NET 4.5 framework is not currently installed on the machine, the DUSTRAN installation will prompt for the start of the .NET 4.5 installation process. Click the “Install” button to start the installation.

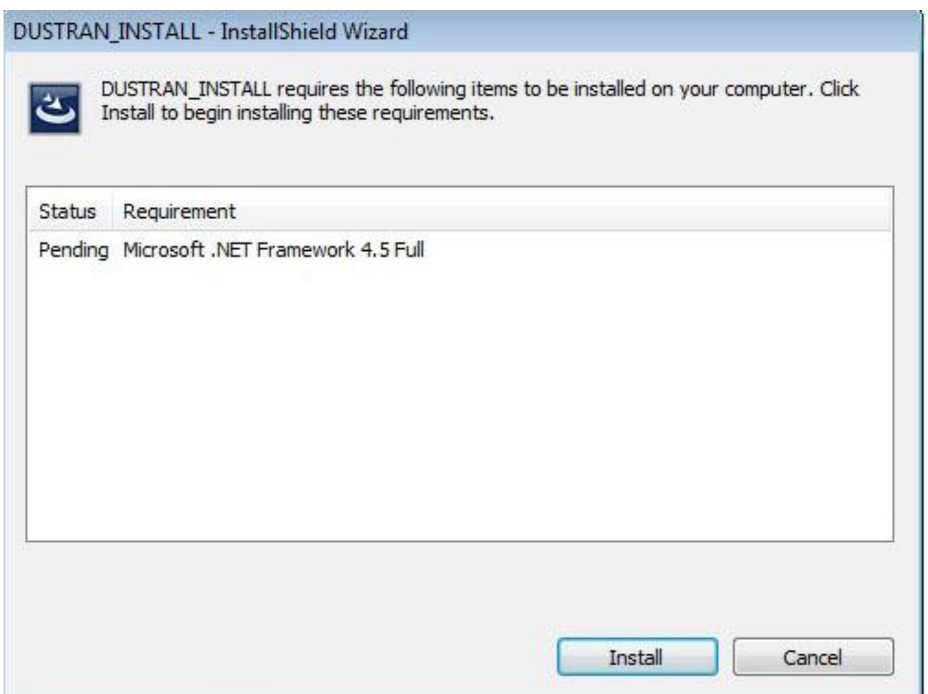


Figure 3.1. .NET Framework 4.5 Installation Start

3. During the installation of the .NET framework, a number of dialog windows will be displayed showing the extracting and installation progress. These windows will close on their own and do not require user action.

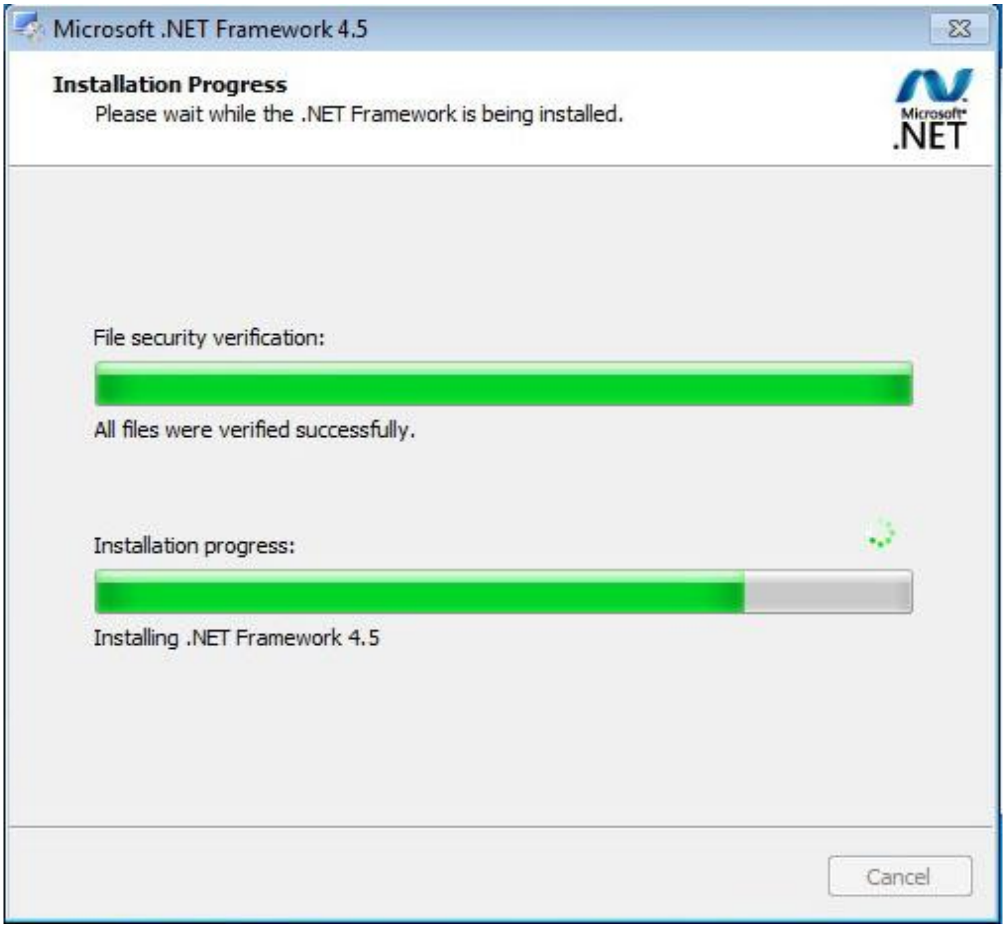


Figure 3.2. .NET 4.5 Installation Progress

4. Once the .NET framework has been installed, the DUSTRAN install wizard will start. Click the “Next” button to continue with the installation.

Figure 3.3. DUSTRAN Install Start

5. A “Destination Folder” prompt will be displayed showing a default path of “C:\DUSTRAN\ as the installation path.

Figure 3.4. DUSTRAN Destination Select

6. Click the “Change…” button to select a different destination folder or click on the “Next” button to select the default path and continue the installation.

7. The "Ready to Install the Program" dialog window will be displayed. Click the Install button to start the installation process.

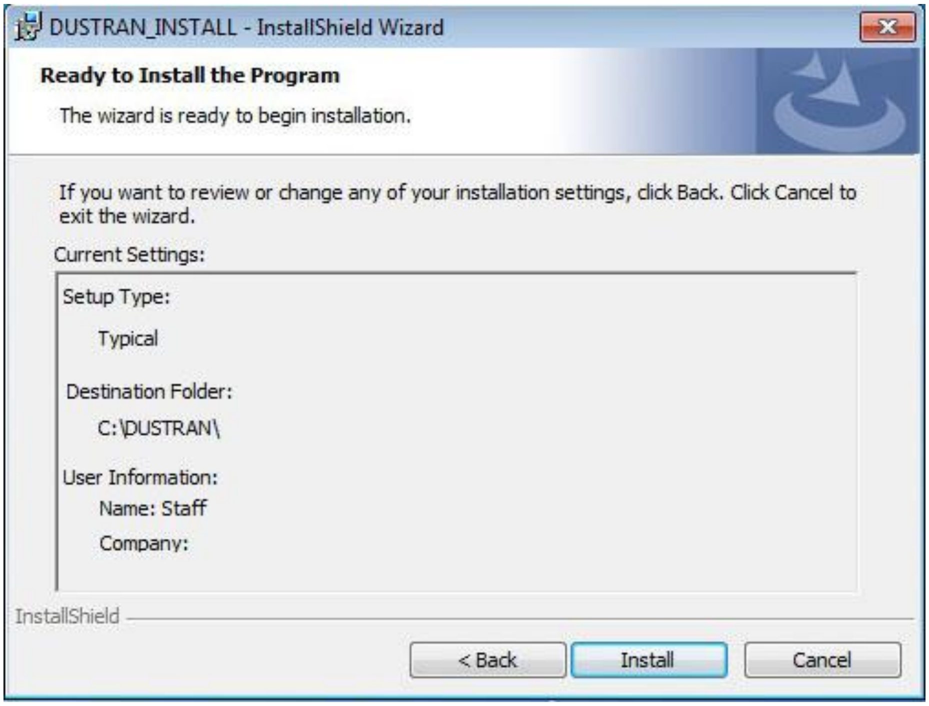


Figure 3.5. Start DUSTRAN Installation

8. When "InstallShield Wizard Completed" is displayed, the DUSTRAN portion of the install has completed, click on the "Finish" button to start the "DUSTRAN Configuration Manager."

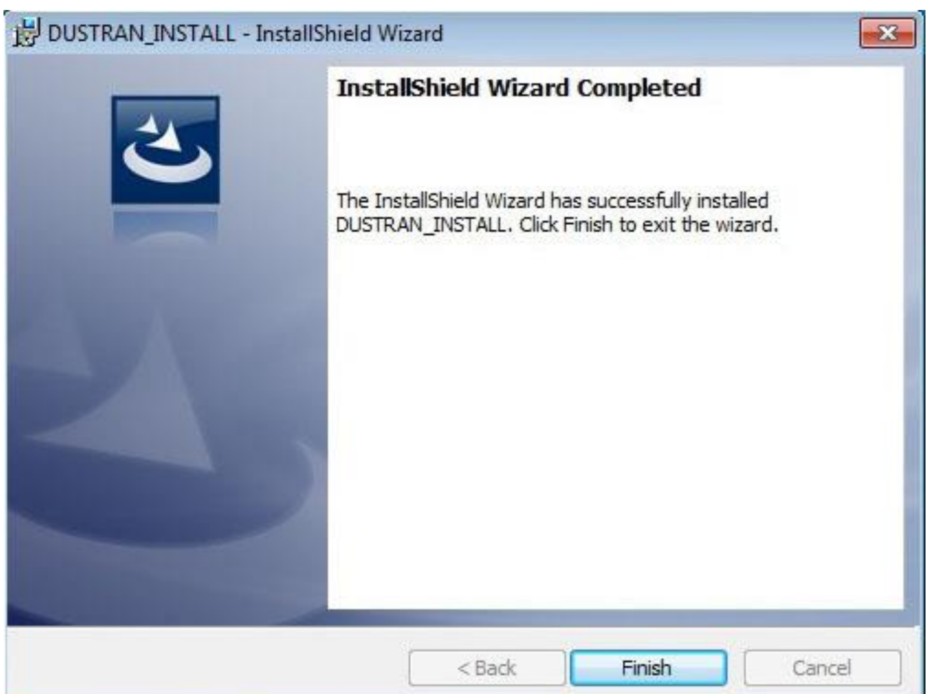


Figure 3.6. DUSTRAN Install Finished

9. Following the DUSTRAN install, the installer will start an install of Microsoft Chart Controls.

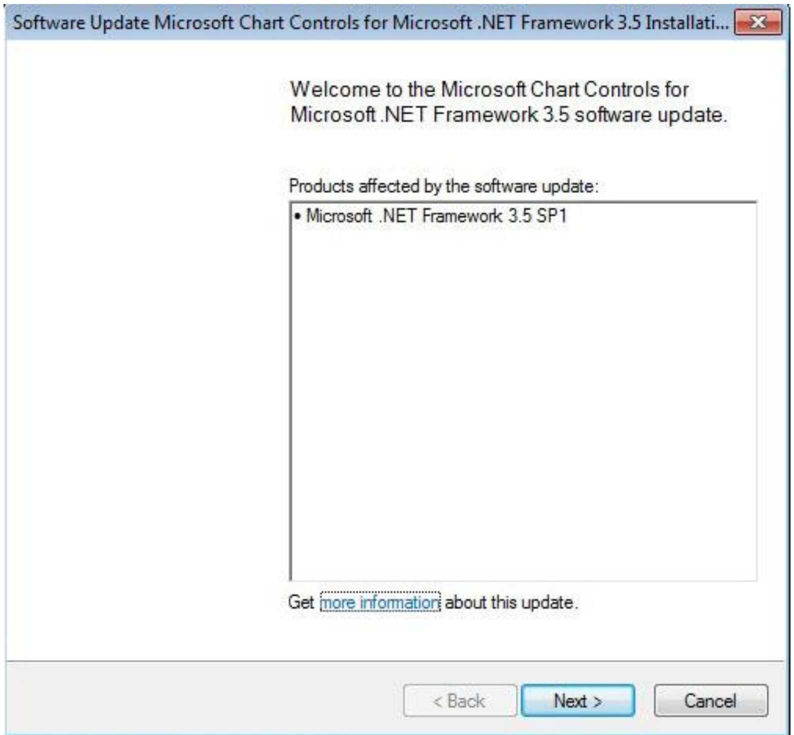


Figure 3.7. Microsoft Chart Control Installation Start Window

10. Review and accept license terms for the chart control software and then click "Next."

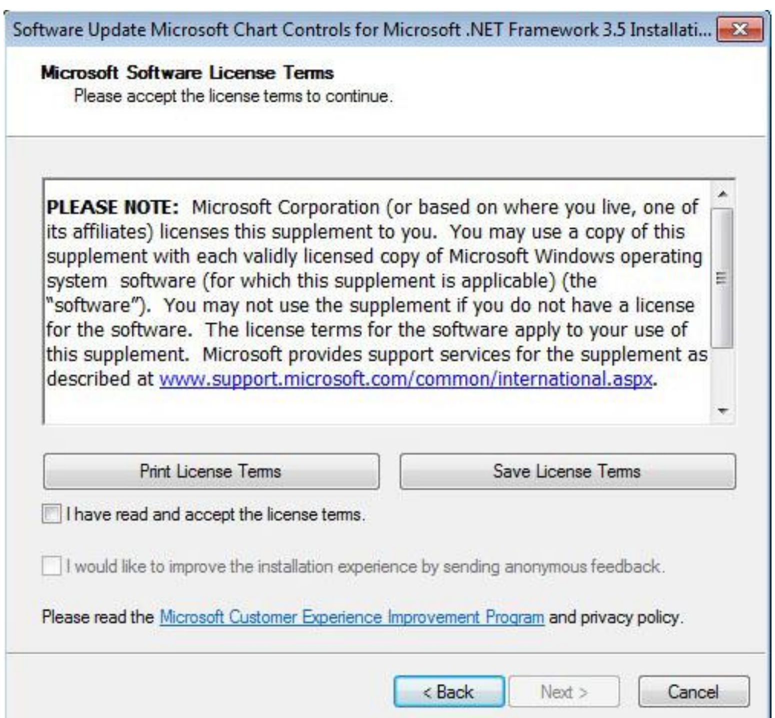


Figure 3.8. Chart Control License Agreement

11. During the installation of the chart control a progress dialog window will be displayed and then shut down once the files have been installed.

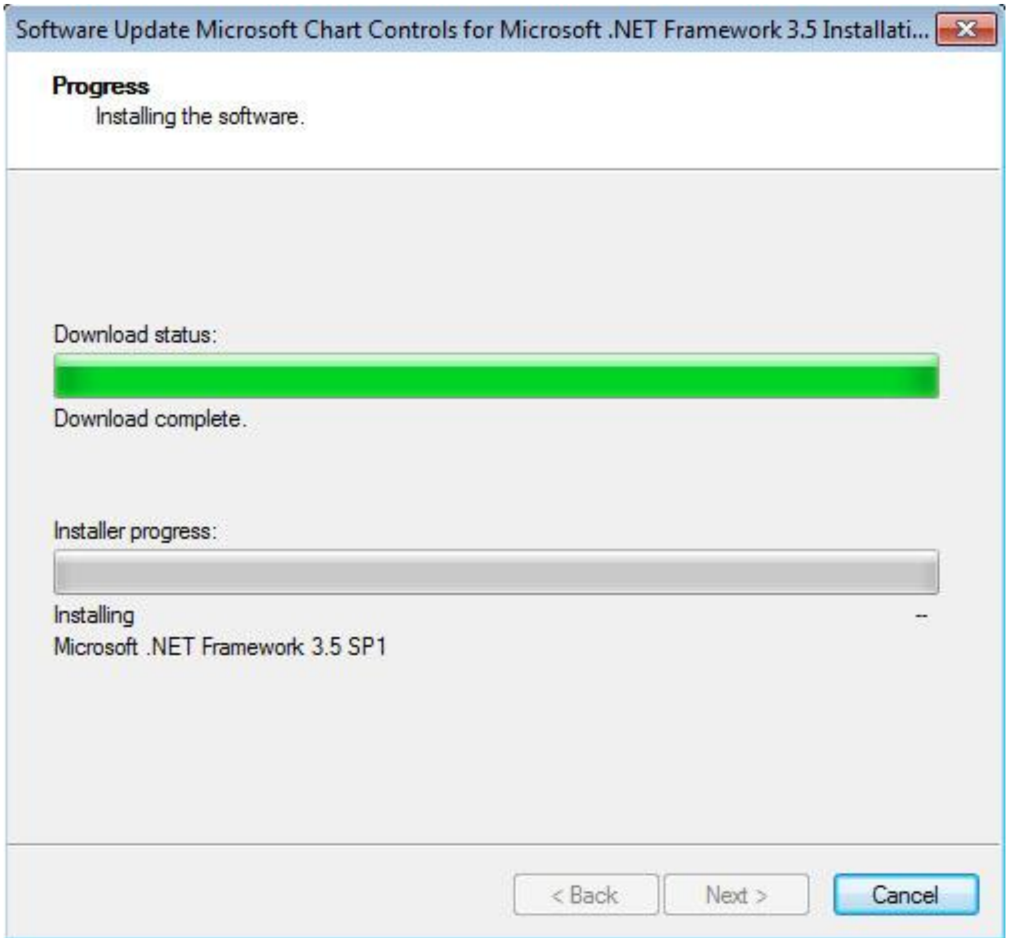


Figure 3.9. Chart Control Installation Progress Window

12. Upon completion, the installer will display a finish dialog. Click on the "Finish" button to close the window and complete the chart control install.

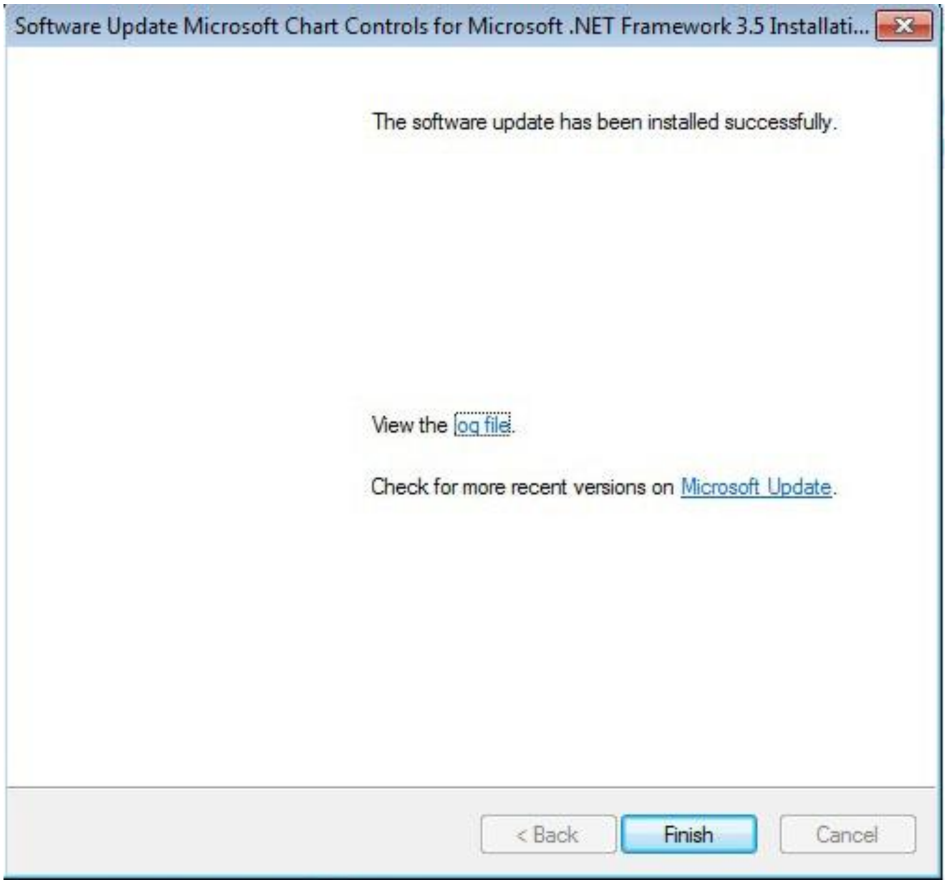


Figure 3.10. Chart Control Installation Complete Window

13. In order to use DUSTRAN there must be a copy of MapWindow GIS software present on the installation machine.
14. If MapWindow has not been previously installed, click on the "Install MapWindow" button of the configuration manager which will start the MapWindow installation wizard.

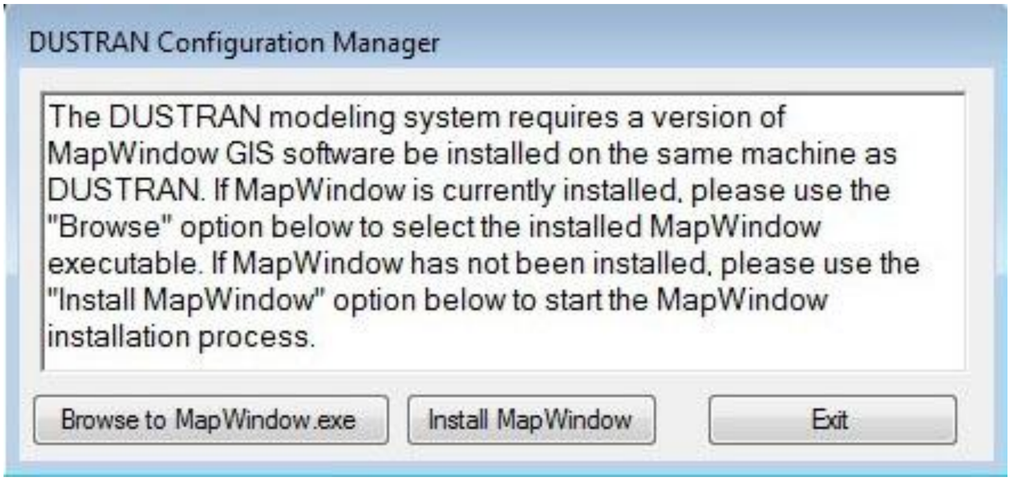


Figure 3.11. DUSTRAN Configuration Manager

15. Once started, the MapWindow installation wizard will walk through the steps required to install the software.
16. Once the MapWindow installation has finished, return to the DUSTRAN Configuration Manager dialog window and click on the "Browse to MapWindow.exe" button.
17. Use the Open dialog window to browse to and select the MapWindow.exe. When the file has been selected click the "Open" button to close the browse window.

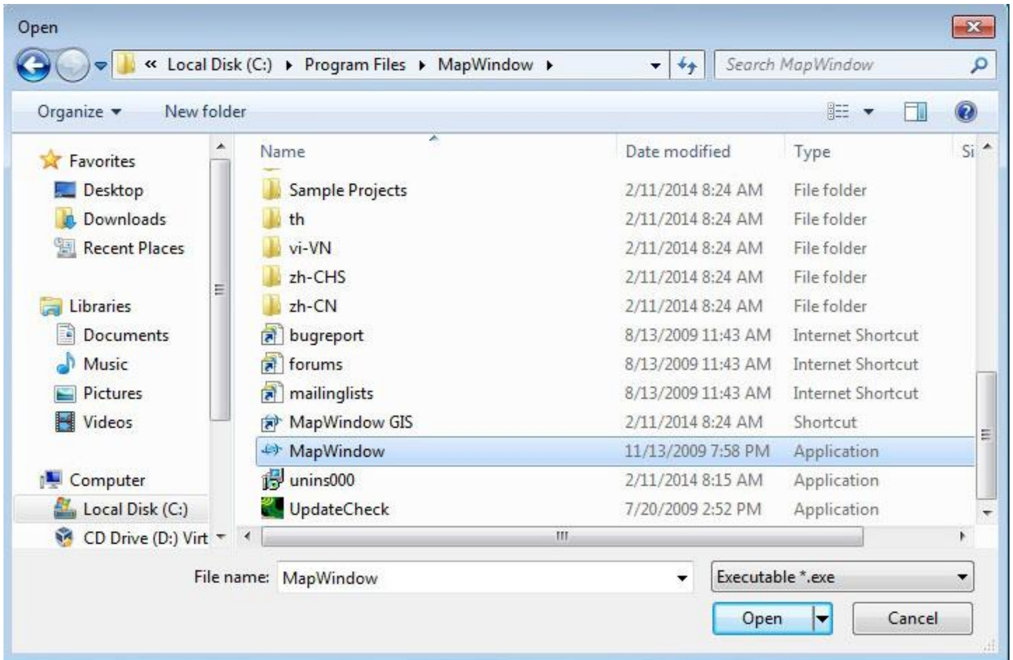


Figure 3.12. Select MapWindow.exe File

18. Once the MapWindow.exe has been selected, a command prompt window will be displayed showing the extraction of the .tif raster file used in the creation of new sites. The extraction will take a few minutes to complete.

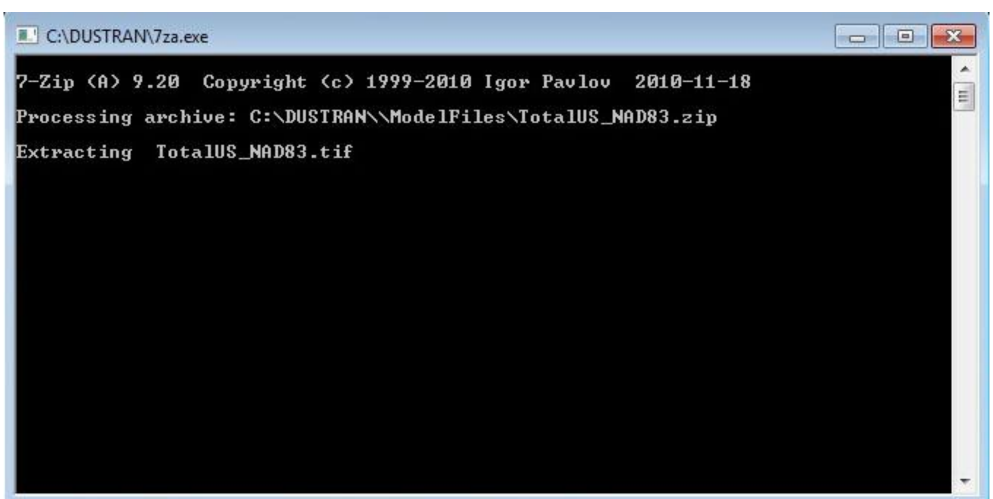


Figure 3.13. .tif File Extraction Window

19. When the extraction completes, “Configuration Complete” will be displayed. Click on the OK button to close the install application.

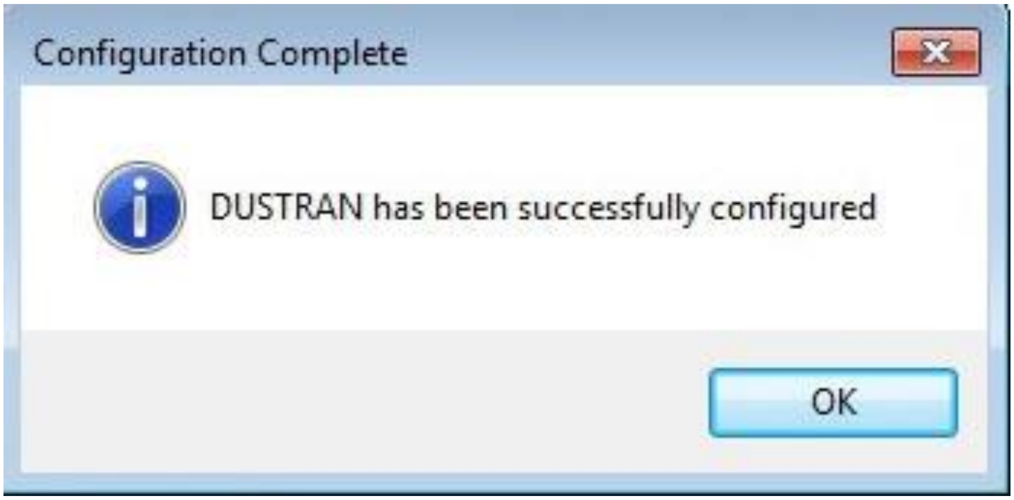


Figure 3.14. DUSTRAN Configuration Complete

20. Double-click on the “MapWindow GIS” desktop icon to start the MapWindow application after the installation has successfully completed.

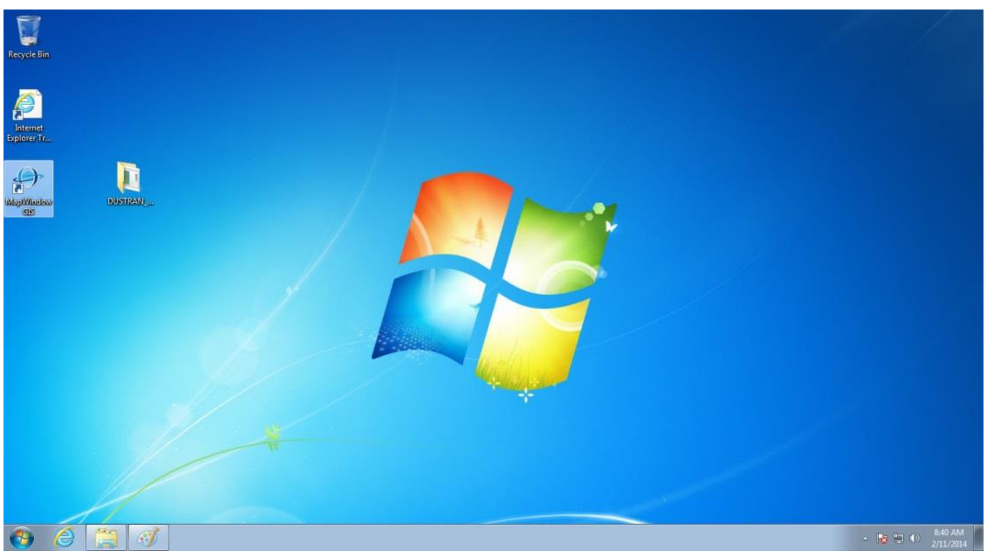


Figure 3.15. MapWindow Desktop Icon

21. Under the “Plug-ins” menu inside of the MapWindow application, select “DUSTRAN.”

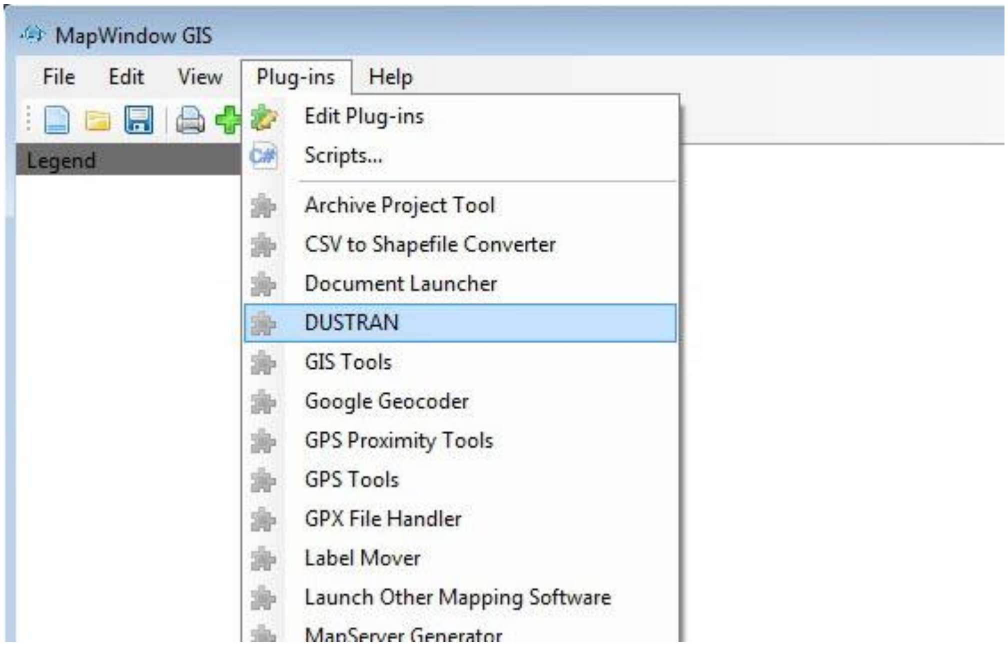


Figure 3.16. Selecting DUSTRAN Plug-in

22. Once the plug-in has been loaded, select the "DUSTRAN" menu item, the "D" icon on the toolbar, or press "alt-d" to open the DUSTRAN user interface.

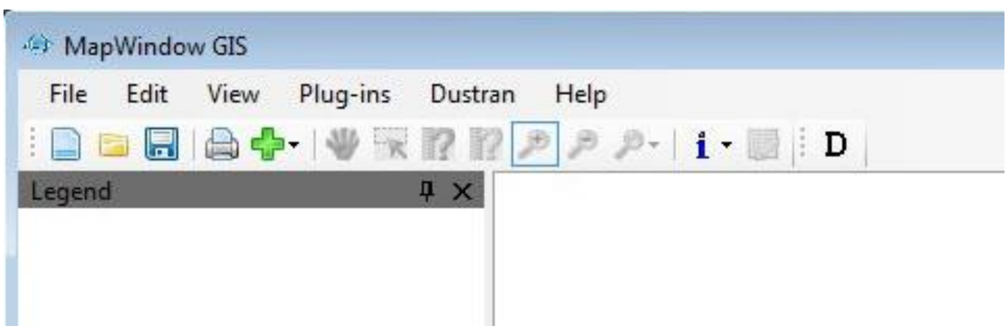


Figure 3.17. MapWindow DUSTRAN Menu Items

23. To add a site, click on the "Add Site" button in DUSTRAN and follow the steps outlined in Chapter 6 of this user's guide to create a new simulation site.

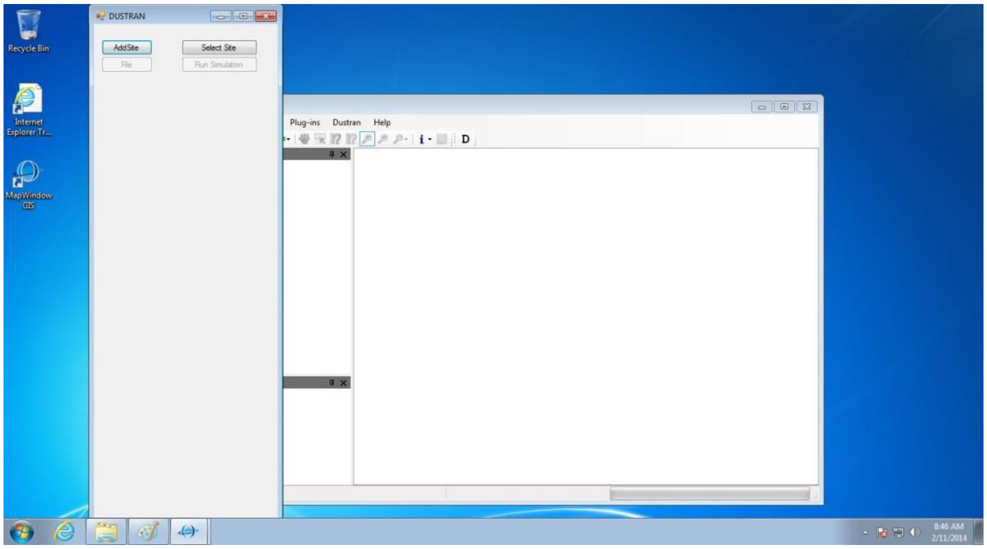


Figure 3.18. DUSTRAN Startup

4.0 DUSTRAN User Interface

DUSTRAN functions as a plug-in within the MapWindow GIS application. To perform a simulation in DUSTRAN, several fields within the interface must be selected, entered, or completed. This section details the various user-entry fields within the interface and is intended as a reference guide for the application. This section assumes that both MapWindow and DUSTRAN are already installed, and the user has some familiarity with the MapWindow application.

4.1 Starting DUSTRAN and Loading a Site

DUSTRAN functions as a plug-in within the MapWindow application and is accessible through the MapWindow toolbar (see Section 3.0, “DUSTRAN Installation Instructions”). To start DUSTRAN, first start the MapWindow application and then start DUSTRAN by clicking on the “D” button located on the MapWindow toolbar.

Once the DUSTRAN interface has been opened, click on the “Select Site…” button to open a dialog box that contains a list of available sites. Select a site from the “Current Site” list and click “Open” to open the site in DUSTRAN. The GIS map files associated with the site are automatically loaded and displayed in the map window. For example, Figure 4.1 shows the Yakima site loaded into the DUSTRAN modeling system showing the Hanford site within Washington State.

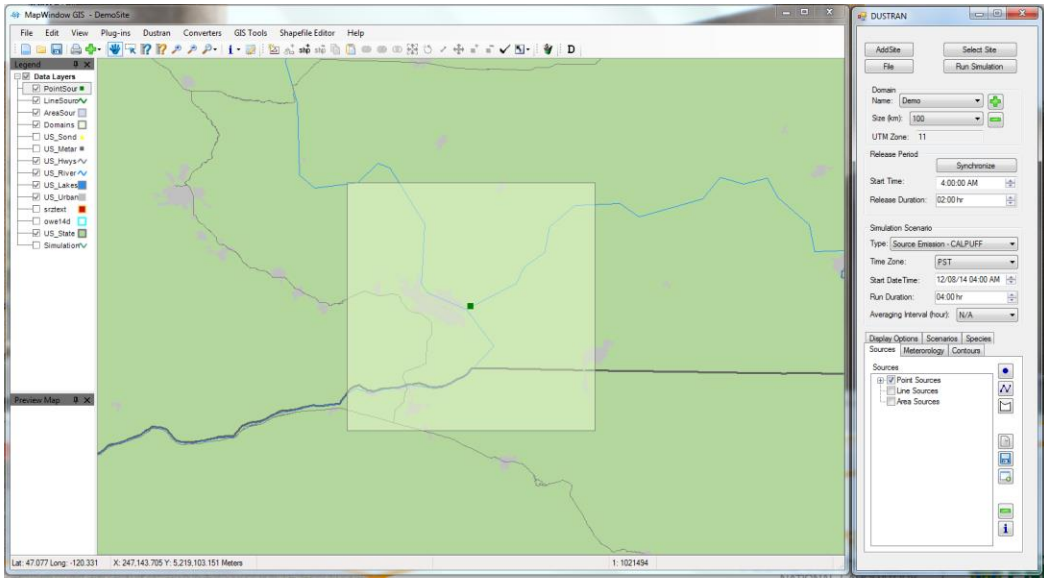


Figure 4.1. Site Loaded into the DUSTRAN Modeling System

4.2 Domain Panel

The “Domain” panel within the DUSTRAN interface is used to set the location and size of the modeling domain that will be used in a model simulation. The modeling domain is the area where both the meteorological and dispersion calculations are performed.

The “Domain” panel allows for domains to be selected, added, and deleted. Figure 4.2 displays an example of the “Domain” panel with an existing domain name and size selected.

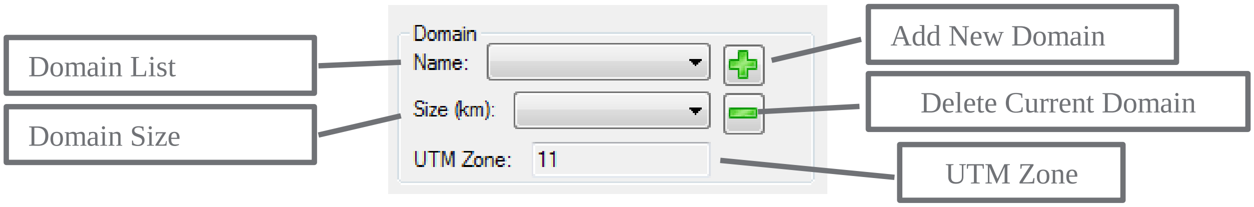


Figure 4.2. Example Domain Panel with Controls Labeled

4.2.1 Creating a New Domain

Multiple model domains can be created for a site, but only one model domain can be used in a given simulation. Use the following steps to create a model domain:

1. Within the “Domain” panel, click on the “Add Domain”  button.
2. Select the location of the center of the new domain by clicking on a map location within MapWindow.
3. When prompted, enter a name for the domain. This name will be added to the list of domains that are stored within the “Name” list; these domains are available for use in a given simulation.
4. Set the domain size by selecting a size from the “Size” list. Domain sizes available for use within DUSTRAN include square areas that are 20, 50, 80, 100, 150, 200, 250, 300, 350, and 400 km on a side.
5. Following the selection of a domain size, MapWindow’s map display will refresh automatically to the selected domain location, and the domain boundary will be sized appropriately.

4.2.2 Selecting an Existing Domain

Domains that currently exist for a site are available for selection under the “Name” list. To select an existing domain:

1. Within the “Domain” panel, select a desired domain from the “Name” list.
2. The domain will automatically display in MapWindow. To change the size of the domain, select a size from the “Size” list.
3. Following the selection of a domain size, MapWindow’s map display will refresh automatically to the selected domain location, and the domain boundary will be sized appropriately.

4.2.3 Deleting a Domain from a Site

A previously created domain can be deleted from a site so that it no longer appears as an available domain under the “Name” list. To delete a domain:

1. Within the “Domain” panel, select the name of the domain to delete from the “Name” list.
2. Click on the “Delete Domain”  button to permanently delete the domain from the site.

4.3 Release Period Panel

The "Release Period" panel is used to set a default start time and duration for all newly created sources. The start time and duration for a given source can be set independently of the default value at the time of the source's creation (see the Section 4.6 for information on creating sources). In addition, the "Release Period" panel can be used to synchronize each selected source's start and end times. Source synchronization is normally performed after all sources have been created and is a convenient method for assigning sources the same release time and duration, if desired. Figure 4.3 displays an example of the "Release Period" panel with the input controls labeled.

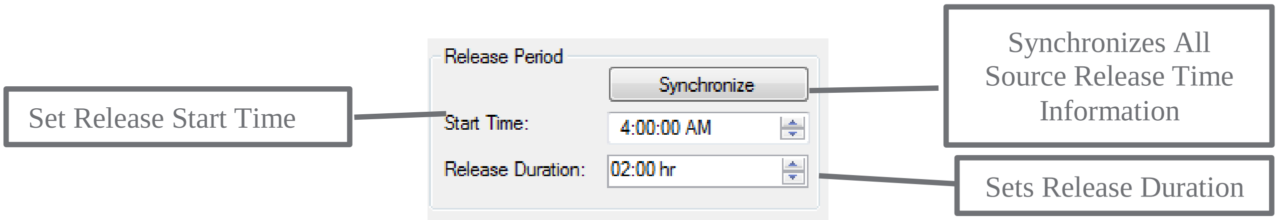


Figure 4.3. Example Release Period Panel with Controls Labeled

4.3.1 Setting the Default Release Start Time and Duration

A default "Start Time" and "Release Duration" can be set for sources on the "Release Period" panel. The "Start Time" is the default starting time in which sources begin releasing material. The "Release Duration" is the default period for which material is released. These values are used as defaults whenever a source is created and can be changed specifically for each source on the "Release Parameters" form for that source.

- To change the default start time, enter an hour value or click the increase or decrease arrow buttons for the "Start Time" input box. Clicking the up-arrow (down-arrow) will increase (decrease) the time by 1 hour. Note, start times are limited to hour increments, and any minutes entered will be ignored by the simulation.
- To change the default release duration, enter an hour value or click the increase or decrease arrow buttons for the "Release Duration" input box. Clicking the up-arrow (down-arrow) will increase (decrease) the time by hourly increments.

4.3.2 Synchronizing the Release Start Time and Duration for All Sources

All start times and durations can be synchronized to the same time and duration for all selected sources. However, synchronization should be used with caution to avoid inadvertently resetting release start times and durations to unintended values. Synchronization is typically performed after all sources have been created via the "Source" tab, but only if the user wants all source start times and release durations to be the same. To synchronize source times:

- Set the default "Start Time" and "Release Duration" on the "Release Period" panel.
- Click on the "Synchronize" button. A dialog box will be displayed asking the user to confirm changing the time data for all of the sources. Selecting the "OK" option will result in the time data for all sources to be changed to the default values found in the "Release Period" panel. Selecting the "Cancel" option will cancel the synchronize operation, and no changes will occur.

4.4 Simulation Scenario Panel

The “Simulation Scenario” panel is used to set the simulation type, start date, time, and run duration. The start date, time, and run duration correspond to the time that the AERMOD, CALPUFF, or CALGRID models are run and do not need to correspond with source release start times or durations. Often, the run duration is chosen so that it is longer than the release duration to continue simulating the plume movement and diffusion on the domain after all sources have finished releasing material. Figure 4.4 shows an example of the “Simulation Scenario” panel with sample entries.

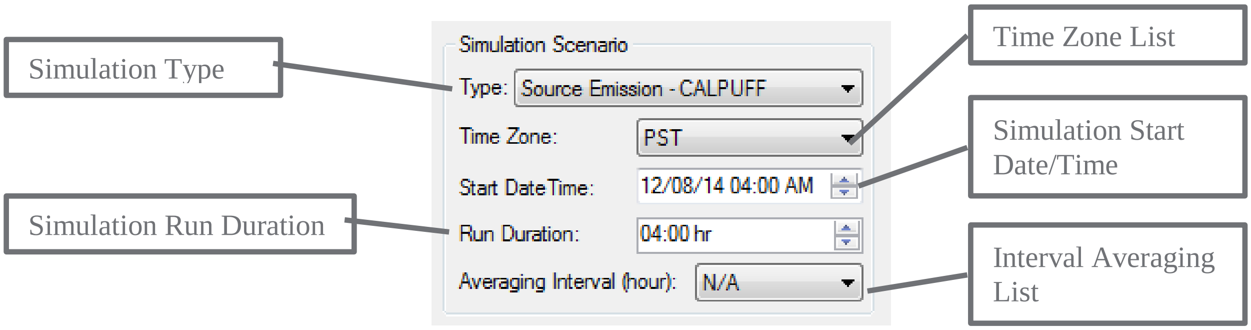


Figure 4.4. Example Simulation Scenario Panel with Controls Labeled

4.4.1 Setting the Simulation Type

The simulation type is used to distinguish between which dispersion model (AERMOD, CALPUFF, or CALGRID) is used to run a given simulation. AERMOD and CALPUFF are used to model standard “Source Emissions” and includes emissions that can be quantified using point-, area-, or line-source types. Vehicular dust emissions, which can be quantified using area- or line-source configurations, is an example source type that should be run using this mode. CALGRID is used to model wind-blown dust dispersion over the modeling domain. A qualitative overview of the models is provided in Sections 2.2.2 and 2.2.3, respectively.

To set the dispersion model to use for the current simulation type, click on the “Type” field and select one of the following:

- “Source Emissions – CALPUFF” to use an EPA version of the CALPUFF dispersion model for the current simulation. CALPUFF is ideal for modeling discrete sources that can be quantified using point-, area-, or line-source types. Emissions from stacks or roadways are example sources that would normally be run using this simulation type.
- “Source Emissions – CALPUFF (DRI)” to use the CALPUFF dispersion model with dry-deposition factors developed by DRI for the current simulation. CALPUFF is ideal for modeling discrete sources that can be quantified using point-, area-, or line-source types. Emissions from stacks or roadways are example sources that would normally be run using this simulation type.
- “Source Emissions – AERMOD” to use the EPA-approved version of the AERMOD dispersion model for the current simulation. AERMOD is ideal for modeling discrete sources that can be quantified using point-, area-, or line-source types. Emissions from stacks or roadways are example sources that would normally be run using this simulation type.
- “Source Emissions – AERMOD (DRI)” to use the AERMOD dispersion model with dry-deposition factors developed by DRI for the current simulation. AERMOD is ideal for modeling

discrete sources that can be quantified using point-, area-, or line-source types. Emissions from stacks or roadways are example sources that would normally be run using this simulation type.

- “Wind-blown Dust” to use the CALGRID dispersion model for the current simulation. The wind-blown dust emissions provided to CALGRID are automatically calculated for each model grid cell within the modeling domain using the approach described in Section 2.3.2.

4.4.2 Setting the Time Zone

The time zone is set through the “Time Zone” listbox within the “Simulation Scenario” panel. Available time zones include: Pacific Standard (PST), Central Standard (CST), Mountain Standard (MST), and Eastern Standard (EST). “Time Zone” is used to correct upper-air soundings to local time and is also used by the CALMET meteorological model to calculate sunrise and sunset for sensible heat flux calculations.

4.4.3 Setting the Start Date

The start date for the simulation is entered in the “Start Date” textbox within the “Simulation Scenario” panel.

- To enter the date manually, click on the month, day, or year portion of the date displayed in the control. Once the appropriate portion of the date has been selected, the new value can be entered. All values should be entered as integers and are automatically checked for correctness. In the case of the month, it will convert the integer to the appropriate month name abbreviation.
- To enter the date using the calendar control, first click on the dropdown arrow. A calendar will be displayed for the date that is currently entered in the “Start Date” control. To change years, click on the year label within the calendar and click on the up (down) arrows to increment (decrement) by one year. To change months, click on the right or left arrow buttons located at the top of the calendar control. Clicking on the left (right) arrow buttons decreases (increases) the month by one. To select a day, click on the desired day within the calendar, and the calendar control will then close with the “Start Date” input box automatically updating to the new date.

4.4.4 Setting the Start Time

The start time for the simulation is entered in the “Start Time” textbox within the “Simulation Scenario” panel. The simulation start time can begin *before* any sources begin releasing material. In fact, the meteorological model, CALMET, requires the simulation to begin *before* sunrise, which for most of the year generally occurs after 05:00 a.m. local time at most continental U.S. locations. This requirement exists because the CALMET-derived boundary-layer parameters and mixing height are a function of the sensible heat flux, which in turn is function of sunrise. For most simulations, setting the “Start Time” to 04:00 a.m. adequately meets CALMET requirements.

To set the start time, click the “Start Time” textbox and enter the hour value manually or by clicking on the increase or decrease arrow buttons found to the right of the input box. Clicking the up-arrow (down-arrow) will increase (decrease) the time by one hour. Start times are currently limited to hour increments and minutes are ignored.

4.4.5 Setting the Run Duration

The run duration for the simulation is entered in the “Run Duration” textbox within the “Simulation Scenario” panel. For CALPUFF or CALGRID, the run duration is typically longer than any source release duration to allow for the dispersion models to continue calculating concentration and deposition values for any residual plume material in the modeling domain.

To set the run duration, click the “Run Duration” textbox and enter an hour value manually or by clicking on the increase or decrease arrow buttons found to the right of the input box. Clicking the up-arrow (down-arrow) will increase (decrease) the time by 15-minute intervals. Currently, the models simulate hour increments, and any minutes are ignored.

4.4.6 Setting the Averaging Interval

The averaging interval of a simulation is the number of hours to use when processing the average of the results generated by the CALPUFF or CALGRID dispersion models. The following averaging intervals are available:

- N/A: No averaging will be performed for the simulation. **Note:** the “N/A” interval corresponds to a model-default, 1-hour average
- 3 Hours: 3-hour averaging will be used for the processing of results
- 8 Hours: 8-hour averaging will be used for the processing of results
- 24 Hours: 24-hour averaging will be used for the processing of results
- Run Length: Averages will be calculated using the entire length of the simulation run

To set the averaging interval, select the desired interval from the “Averaging Interval” list found in the “Simulation Scenario” panel. If the averaging interval selected is greater than the “Run Duration,” a message box will be displayed explaining the time mismatch.

4.5 Species Tab

The “Species” tab is used to view, add, and delete species from the current model scenario. Species can be either particles or gases and their properties (diameter or molecular weight) can be set to allow for deposition calculations. Species that are “checked” are available for selection on all source forms when creating new or editing existing sources. Figure 4.5 shows an example of the “Species” tab with four species selected and the controls labeled.

4.5.1 Selecting an Existing Species

Existing species that are available for selection in a simulation are shown in the “Available Species” list on the “Species” tab. Species that are selected will appear on each source’s “Release Parameter” form.

To select an existing species, check the box located to the left of the species. There are four default PM species—PM10, PM2.5, PM15, and PM30. The number after the PM designation represents the mean particle diameter, in microns. Emissions for these four particle size categories are calculated automatically by the dust-emission module within DUSTRAN. However, emission rates for user-specified species are not determined by DUSTRAN and must be entered directly for each source.

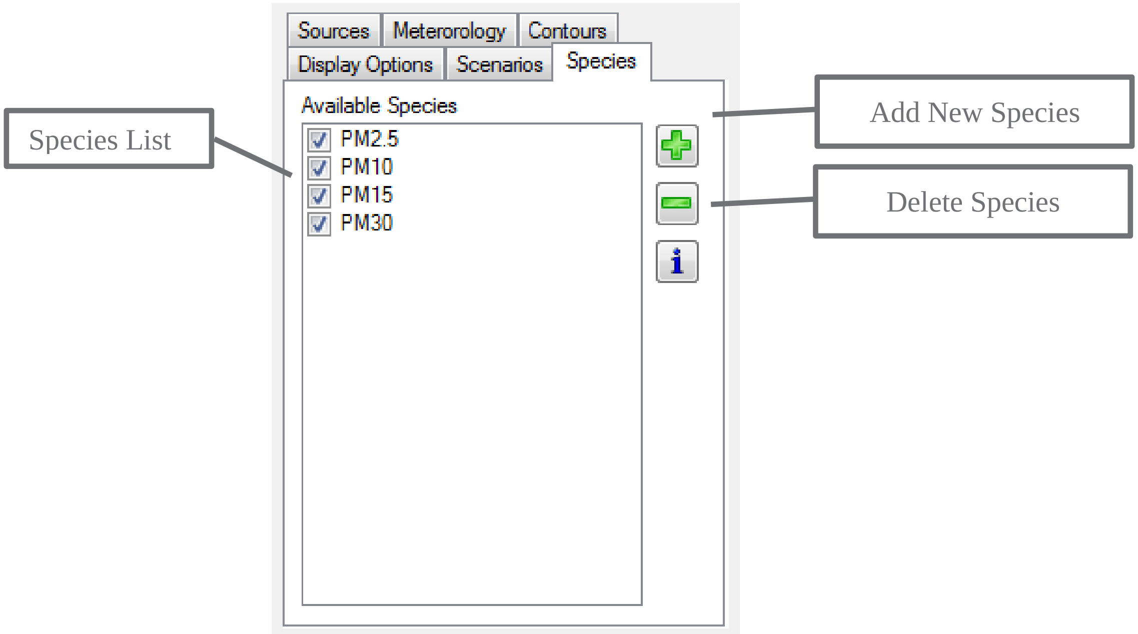


Figure 4.5. Example Species Tab

4.5.2 Modifying an Existing Species

Existing species data can be modified. For example, the mean diameter for a particle or the molecular weight for a gas can be changed. To modify an existing species:

- Double-click on the name of the species in the "Available Species" list on the "Species" tab. The "Species Input Data" window will appear.
- To change the existing species, click "Yes."
- Indicate whether the species is a gas by clicking "Yes" or particle by clicking "No."
- If deposition calculations are desired, click "Yes"; otherwise click "No." For deposition, the program will prompt for the species' mean diameter (particle) or molecular weight (gas).

4.5.3 Adding a New Species

New species, either particles or gases, can be added to a simulation. The species will then be available on each source's "Release Parameter" form. To add a new species:

- Click the "Add"  button located on the "Species" tab. When prompted, enter a name for the new species.
- Next, indicate whether the species is a gas by clicking "Yes" or particle by clicking "No."

If deposition calculations are desired, click "Yes"; otherwise click "No." For deposition, the program will prompt for the species' mean diameter in μm (particle) or molecular weight (gas).

4.5.4 Deleting an Existing Species

Existing species can be deleted from a simulation. Release information from existing sources for that species will also be deleted.

To delete a species, select the species from the list and click the "Delete"  button located on the "Species" tab.

4.6 Sources Tab

The "Sources" tab is used to add, edit, and delete sources within DUSTRAN. In addition, the sources tab is used to associate characteristic data files, such as vegetation and soil texture layers, which are used by the DUSTRAN modeling system to generate gridded dust emissions for a model domain when simulating wind-blown dust. Various source types are available, including point, area, and line sources. Once a source has been created, it will appear on the source list within the "Sources" tab; each source appearing on the list can be edited and can be selected or deselected for use in the current simulation. Figure 4.6 shows an example "Sources" tab with the controls labeled and existing sources populated in the source list.

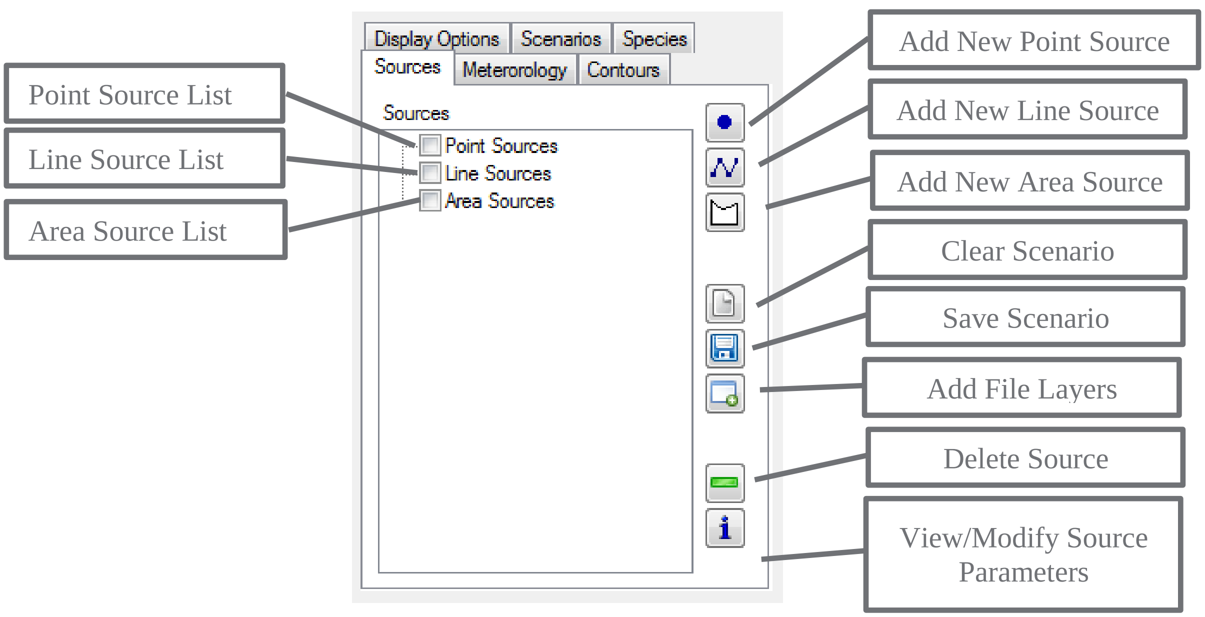


Figure 4.6. Example Sources Tab with Point, Line, and Area Sources Entered

4.6.1 Adding a New Point Source

A point source, such as a stack emission, is modeled as a discrete point that can include the effects of plume rise due to buoyancy and momentum. A new point source can be added to a DUSTRAN simulation by doing the following:

- Click on the "Add New Point Source"  button. A prompt will appear directing the user to select a location in the MapWindow map display for the point source. Click "OK."

- The mouse cursor will change to a crosshair; move the crosshair to the desired location for the point source on the map within MapWindow and click. Enter a name for the point source in the dialog box that appears and click "OK."
- The point source will appear in the source list under "Point Sources" and will also be displayed within MapWindow.

A "Source Input" window will appear with two tabs—"Release Parameters" and "Coordinates." "Release Parameters" are editable parameters that describe the release for the point source. "Coordinates" are the UTM easting and northing coordinates for the source and are not editable.

4.6.1.1 Setting Point-Source Release Parameters

Once a new point source has been added or the name of an existing point source has been double-clicked on the "Sources" tab, the point "Source Input" window will be displayed. The "Release Parameters" tab is selected by default and is used to enter the characteristics of the point source, such as the stack diameter, release time period, and the emission rates of species emitted by the source. Figure 4.7 displays an example "Release Parameters" form for a point source.

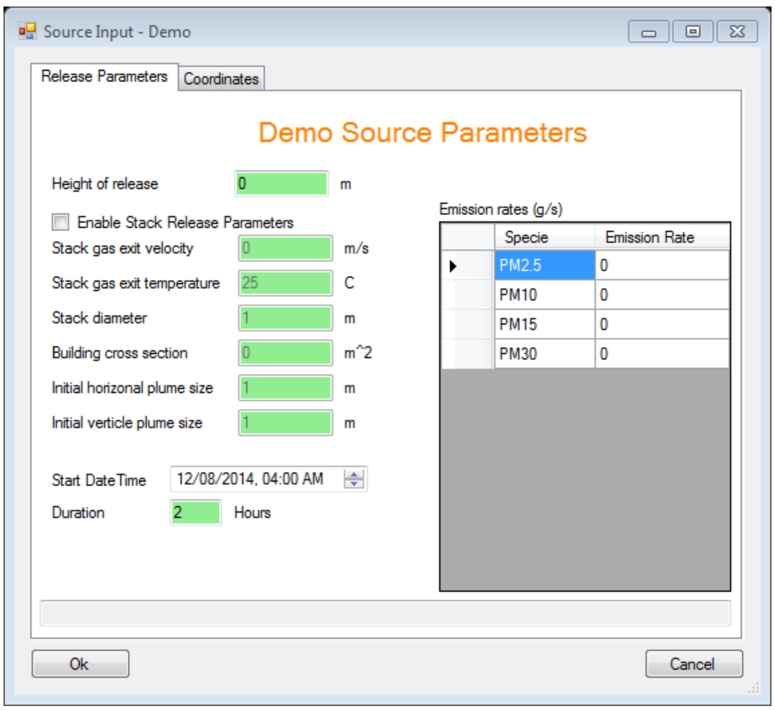


Figure 4.7. Example Point-Source Input Window

The following input parameters are available on the "Release Parameters" tab for a point source:

- Height of release
- Stack gas exit velocity
- Stack gas exit temperature
- Stack diameter
- Building cross section
- Initial horizontal plume size
- Initial vertical plume size.

Default values are provided for each parameter. To change a parameter's value, click on the textbox and enter a value. For quality assurance, valid textbox entries will appear green and invalid textbox entries will appear red. A range of valid entries appears in the yellow label at the bottom of the form for a given textbox entry.

The point-source "Start Date," "Start Time," and "Duration" must also be entered on the "Release Parameter" tab. This is the date, time, and total period the source emits material in the simulation.

- Setting the point-source start date:
  - To enter the date manually, click on the month, day, or year portion of the date displayed in the control. Once the appropriate portion of the date has been selected, the new value can be entered. All values should be entered as integers and are automatically checked for correctness. In the case of the month, it will convert the integer to the appropriate month name abbreviation.
  - To enter the date using the calendar control, first click on the dropdown arrow. A calendar will be displayed for the date that is currently entered in the "Start Date" control. To change years, click on the year label within the calendar and click on the up (down) arrows to increment (decrement) by one year. To change months, click on the right or left arrow buttons located at the top of the calendar control. Clicking on the left (right) arrow buttons decreases (increases) the month by one. To select a day, click on the desired day within the calendar, and the calendar control will then close with the "Start Date" input box automatically updating to the new date.
- Setting the point-source start time:
  - Click the "Start Time" textbox and enter the hour value manually or by clicking on the increase or decrease arrow buttons found to the right of the input box. Clicking the up-arrow (down-arrow) will increase (decrease) the time by one hour. Start times are currently limited to hour increments, and any minutes are ignored.
- Setting the point-source duration:
  - Click the "Duration" textbox and enter an hour value manually or click on the increase or decrease arrow buttons found to the right of the input box. Clicking the up-arrow (down-arrow) will increase (decrease) the time by hourly intervals.

Emission rates for material being released from the point source are also entered on the "Release Parameters" tab. Emission rates can be set for several species (Note: Species are added in DUSTRAN through the "Species" tab [see Section 4.5].)

To set the point-source emission rates, click on the "Emission Rate" cell for a given "Species" and enter its emission rate. If an "Emission Rate" is labeled "Not Selected," the species must be activated on the "Species" tab in DUSTRAN. All emission rates are validated before closing the form.

4.6.2 Adding a New Line Source

A line source, such as emissions from roadways, is modeled as a series of line segments in DUSTRAN. Line-source emissions can be entered explicitly or calculated automatically when using the vehicular dust-emissions module for simulating dust emission from paved or unpaved roadways. A new line source can be added to a simulation by doing the following:

- Click on the "Line Source"  button. A prompt will appear directing the user to select a location in the MapWindow map display for the line source. Click "OK."

- The mouse cursor will change to a crosshair. The line source is drawn as a series of line segments. To create a line segment, click the mouse on the beginning and ending points of the segment. To complete the line source, double-click the mouse on the ending point of the last line segment. Enter a name for the line source in the dialog box that appears and click "OK."
- The line source will appear in the source list under "Line Sources" and will also be displayed within MapWindow.

A "Source Input" window will appear with two tabs—"Release Parameters" and "Coordinates." The "Release Parameters" tab contains editable parameters that describe the release for the line source. "Coordinates" are the UTM easting and northing coordinates for the source and are not editable. A third tab, called "Vehicle Parameters," is available if the "Emission Model" option is selected on the "Release Parameters" tab. The "Vehicle Parameters" tab is used to enter characteristic vehicle information for the line-source dust-emissions model.

4.6.2.1 Setting Line-Source Release Parameters

Once a new line source has been added or the name of an existing line source has been double-clicked on the "Sources" tab, the line "Source Input" window will be displayed. By default, the "Release Parameters" tab is selected and is used to enter characteristics of the line source, such as the release height above ground level. Figure 4.8 displays an example "Release Parameters" form for a line source with the "Emission Model" option selected.

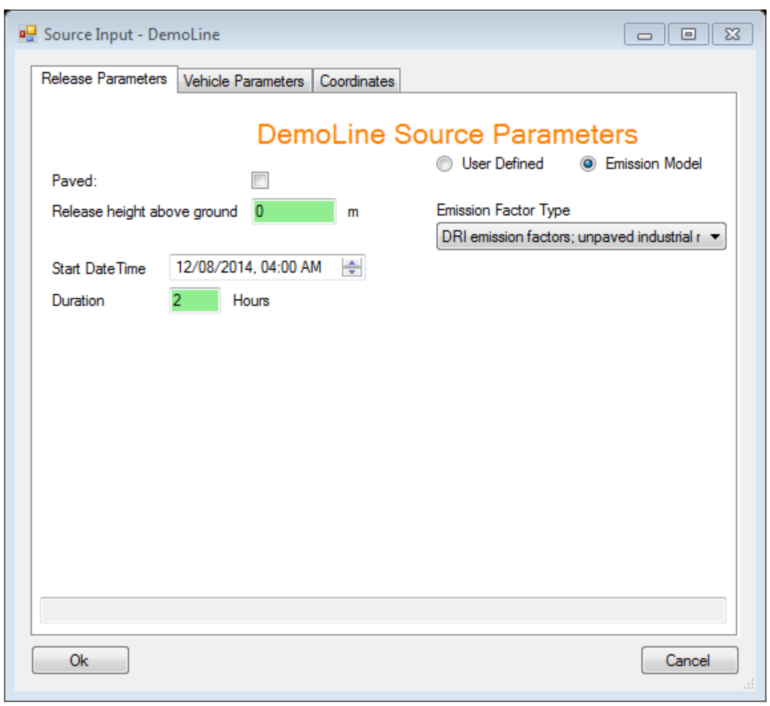


Figure 4.8. Example Line-Source Input Window Using the User-Defined Emission Option

The "Release height above ground" setting is required for each line source and can include heights to accommodate elevated releases. For road emissions, the release height should be zero. To change the parameter's value, click on the textbox and enter a value.

The line-source “Start Date,” “Start Time,” and “Duration” must also be entered on the “Release Parameter” tab. This is the date, time, and total period the source emits material in the simulation.

- Setting the line-source start date:
  - To enter the date manually, click on the month, day, or year portion of the date displayed in the control. Once the appropriate portion of the date has been selected, the new value can be entered. All values should be entered as integers and are automatically checked for correctness. In the case of the month, it will convert the integer to the appropriate month name abbreviation.
  - To enter the date using the calendar control, first click on the dropdown arrow. A calendar will be displayed for the date that is currently entered in the “Start Date” control. To change years, click on the year label within the calendar and click on the up (down) arrows to increment (decrement) by one year. To change months, click on the right or left arrow buttons located at the top of the calendar control. Clicking on the left (right) arrow buttons decreases (increases) the month by one. To select a day, click on the desired day within the calendar, and the calendar control will then close with the “Start Date” input box automatically updating to the new date.
- Setting the line-source start time:
  - Click the “Start Time” textbox and enter the hour value manually or by clicking on the increase or decrease arrow buttons found to the right of the input box. Clicking the up-arrow (down-arrow) will increase (decrease) the time by one hour. Start times are currently limited to hour increments, and any minutes are ignored.
- Setting the line-source duration:
  - Click the “Duration” textbox and enter an hour value manually or click on the increase or decrease arrow buttons found to the right of the input box. Clicking the up-arrow (down-arrow) will increase (decrease) the time by 15-minute intervals. Currently, the models simulate hour increments, and any minutes are ignored.

Emissions for line sources can either be “User Defined” or calculated by the “Emissions Model” using EPA AP-42 (EPA 2005) or DRI (Gillies et al. 2005a, 2005b) emission factors. Selecting “Emissions Model” causes the “Vehicles Parameters” tab to appear on the “Source Input” form (see “Setting Line Source Vehicle Parameters” for this option). If “User Defined” is selected, the emission rates can be entered directly on the “Release Parameters” tab. Emission rates can be set for several species (Note: species are added in DUSTRAN through the “Species” tab [see Section 4.5].)

To set the line-source emission rates, click on the “Emission Rate” cell for a given “Species” and enter its emission rate. If an “Emission Rate” is labeled “Not Selected,” the species must be activated on the “Species” tab in DUSTRAN. All emission rates are validated before closing the form.

4.6.2.2 Setting Line-Source Vehicle Parameters

The line-source “Vehicle Parameters” tab is activated on the line “Source Input” form by selecting “Emission Model” on the “Release Parameters” tab. The “Vehicle Parameters” tab is used to enter information about vehicles that will be traveling along the path of the line source. This information, which is used to calculate road dust emissions generated by vehicle traffic, includes the types of vehicles as well as their speed, weight, and the distance traveled. The particulate emission factors for wheeled vehicles operating on unpaved roads are empirically derived functions described in Section 2.3.1. Two calculation options are available: emissions using factors formulated by DRI (Gillies et al. 2005a, 2005b), or emissions using factors calculated using EPA AP-42 recommendations (EPA 2005). Figure 4.9 shows the “Vehicle Parameters” form for a sample line source with vehicles selected.

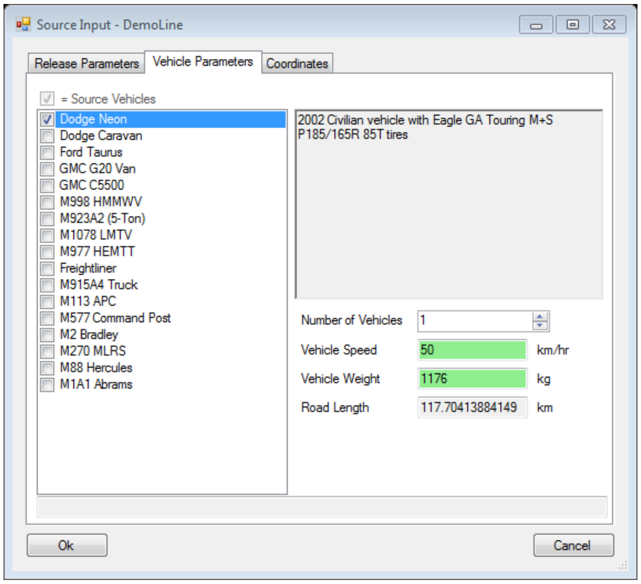


Figure 4.9. Example Vehicle Input Window for a Line Source

A vehicle is used as an emission source by checking the box next to the vehicle's name. By clicking on a vehicle's name, a description of the vehicle is given in the "Vehicle Description" box and a set of input parameters for the vehicle is displayed. The vehicle-specific input parameters include:

- Number of Vehicles
- Vehicle Speed
- Vehicle Weight
- Road Length (non-editable—calculated by the interface).

As discussed in Section 2.2.2.5, DUSTRAN does not treat the motion of individual vehicles. Instead, a bulk approach is used to quantify dust emissions from vehicle activities. That is, the dust emissions from all vehicles active on a road over a specified time are assumed to be released uniformly from the road at a constant rate throughout the duration of the activity. Therefore, the input fields on the "Vehicle Parameters" form (Figure 4.9) should not be interpreted as the specific motion of individual vehicles; the form is simply an approach for providing the information needed by DUSTRAN in a bulk sense.

The approach for determining the input required on the "Vehicle Parameters" form (Figure 4.9) is to first estimate the total distance traveled along the road for all vehicles within a given vehicle type throughout the duration of the activity and then divide this total distance traveled by the road length to get the total effective "Number of Vehicles" on the form (Figure 4.9). Since the number of vehicles must be entered as a whole number, it is recommended that any fractional vehicles be rounded up to err on the side of conservatism in the dust emissions estimated from the vehicle activities. The only constraint on the inputs on the form is that the effective number of vehicles × vehicle speed × activity duration be greater than the road length.

As described in Section 2.3.1, the dust emissions from a moving vehicle are proportional to the vehicle momentum (i.e., vehicle weight × vehicle speed). Therefore, if some vehicles of one type travel at significantly different speeds than other vehicles of the same type, another vehicle type will need to be added to the list such that the other speed(s) can be specified.

For quality assurance, valid textbox entries will appear green and invalid textbox entries will appear red. A range of valid entries appears in the yellow label at the bottom of the form for a given textbox entry.

4.6.3 Adding a New Area Source

An area source, such as emissions from vehicles randomly crossing an off-road region, is modeled as a three- or four-sided polygon in DUSTRAN. Area-source emissions can be entered explicitly or calculated automatically when using the vehicular dust-emissions module for simulating dust emissions from paved or unpaved roadways. A new area source can be added to a simulation by doing the following:

- Click on the "Area Source"  button. A prompt will appear directing the user to select a location in the MapWindow map display for the area source. Click "OK."
- The mouse cursor will change to a crosshair. To create the area source, click the mouse at three or four points, depending on whether you want a triangle or a four-sided polygon. On the last point, double-click the mouse to complete the area source. Enter a name for the area source in the dialog box that appears and click "OK."
- The area source will appear in the source list under "Area Sources" and will also be displayed within MapWindow.

A "Source Input" window will appear with two tabs—"Release Parameters" and "Coordinates." The "Release Parameters" tab contains editable parameters that describe the release for the area source. "Coordinates" are the UTM easting and northing coordinates for the four corners of the area source and are not editable. A third tab, called "Vehicle Parameters," is available if the "Emissions Model" option is selected on the "Release Parameters" tab. The "Vehicle Parameters" tab is used to enter characteristic vehicle information for the area-source dust-emissions model.

4.6.3.1 Setting Area-Source Release Parameters

Once a new area source has been added or the name of an existing area source has been double-clicked on the "Sources" tab, the area "Source Input" window will be displayed. By default, the "Release Parameters" tab is selected and is used to enter characteristics of the area source, such as the effective release height above ground level. Figure 4.10 displays an example "Release Parameters" form for an area source with the "User-Defined" emissions option selected.

The following input parameters are available on the "Release Parameters" tab for an area source:

- Effective height above ground
- Air temperature
- Effective rise velocity
- Effective radius
- Initial vertical spread.

Default values are provided for each parameter. To change a parameter's value, click on the textbox and enter a value. For quality assurance, valid textbox entries will appear green and invalid textbox entries will appear red. A range of valid entries appears in the yellow label at the bottom of the form for a given textbox entry.

The area-source "Start Date," "Start Time," and "Duration" must also be entered on the "Release Parameter" tab. This is the date, time, and total period the source emits material in the simulation.

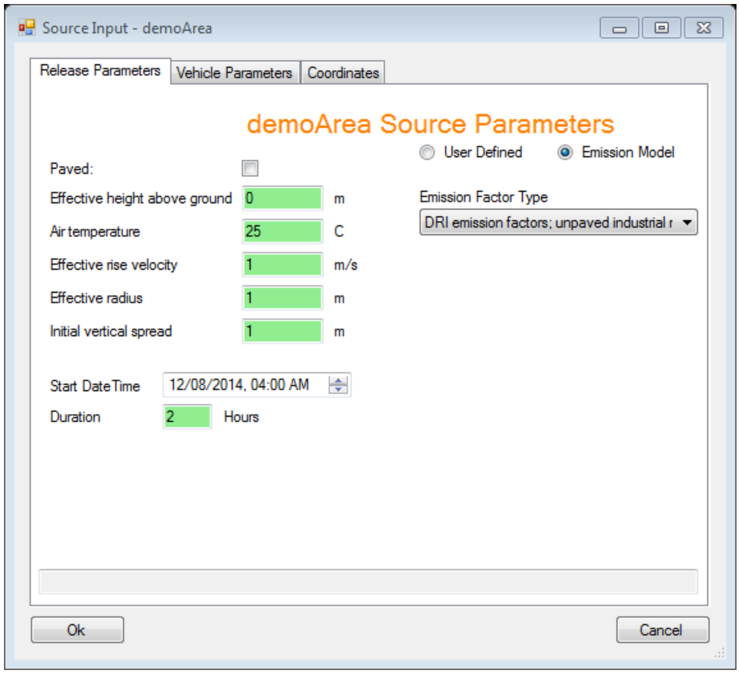


Figure 4.10. Example Area-Source Input Window Using the User-Defined Emission Option

- Setting the line-source start date:
  - To enter the date manually, click on the month, day, or year portion of the date displayed in the control. Once the appropriate portion of the date has been selected, the new value can be entered. All values should be entered as integers and are automatically checked for correctness. In the case of the month, it will convert the integer to the appropriate month name abbreviation.
  - To enter the date using the calendar control, first click on the dropdown arrow. A calendar will be displayed for the date that is currently entered in the "Start Date" control. To change years, click on the year label within the calendar and click on the up (down) arrows to increment (decrement) by one year. To change months, click on the right or left arrow buttons located at the top of the calendar control. Clicking on the left (right) arrow buttons decreases (increases) the month by one. To select a day, click on the desired day within the calendar, and the calendar control will then close with the "Start Date" input box automatically updating to the new date.
- Setting the line-source start time:
  - Click the "Start Time" textbox and enter the hour value manually or by clicking on the increase or decrease arrow buttons found to the right of the input box. Clicking the up-arrow (down-arrow) will increase (decrease) the time by one hour. Start times are currently limited to hour increments, and any minutes are ignored.
- Setting the line-source duration:
  - Click the "Duration" textbox and enter an hour value manually or click on the increase or decrease arrow buttons found to the right of the input box. Clicking the up-arrow (down-arrow) will increase (decrease) the time by 15-minute intervals. Currently, the models simulate hour increments, and any minutes are ignored.

Emissions for area sources can either be "User Defined" or calculated by the "Emissions Model" using EPA AP-42 (EPA 2005) or DRI (Gillies et al. 2005a, 2005b) emission factors. Selecting "Emissions Model" causes the "Vehicles Parameters" tab to appear on the "Source Input" form (see "Setting Area

Source Vehicle Parameters" for this option). If "User Defined" is selected, the emission rates can be entered directly on the "Release Parameters" tab. Emission rates can be set for several species (Note: Species are added in DUSTRAN through the "Species" tab [see Section 4.5]).

To set the area-source emission rates, click on the "Emission Rate" cell for a given "Species" and enter its emission rate. If an "Emission Rate" is labeled "Not Selected," the species must be activated on the "Species" tab in DUSTRAN. All emission rates are validated before closing the form.

4.6.3.2 Setting Area-Source Vehicle Parameters

The area-source "Vehicle Parameters" tab is activated on the area "Source Input" form by selecting "Emission Model" on the "Release Parameters" tab. The "Vehicle Parameters" tab is used to enter information about vehicles that will be traveling within the area source. This information, which is used to calculate road dust emissions generated by the vehicles, includes the types of vehicles as well as their speed, weight, and the distance traveled. The particulate emission factors for wheeled vehicles operating on unpaved roads are empirically derived functions described in Section 2.3.1. Two calculation options are available: emissions using factors formulated by DRI (Gillies et al. 2005a, 2005b) or emissions using factors based on EPA's AP-42 recommendations (EPA 2005). Figure 4.11 shows the "Vehicle Parameters" form for a sample area source with vehicles selected.

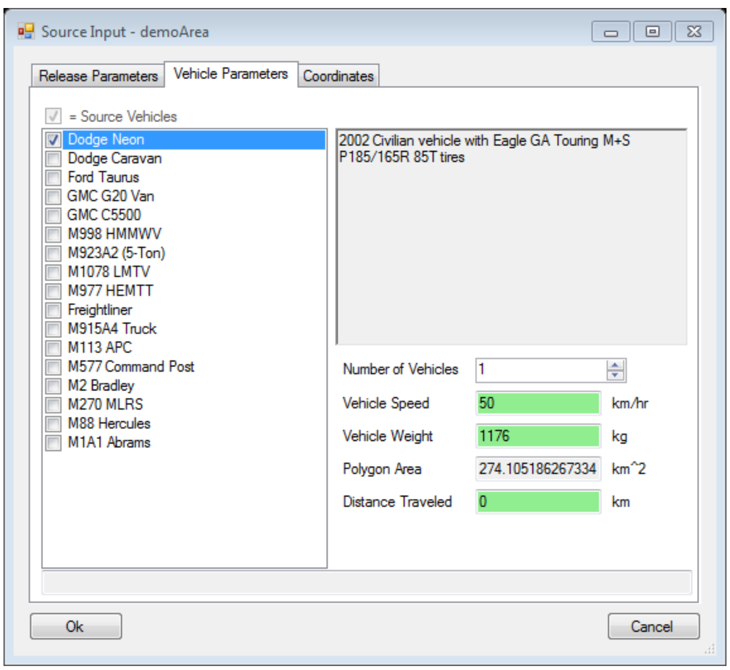


Figure 4.11. Example Vehicle Input Window for an Area Source

A vehicle is used as an emission source by checking the box next to the vehicle's name. By clicking on a vehicle's name, a description of the vehicle is given in the "Vehicle Description" box, and a set for vehicle input parameters is displayed. The vehicle-specific input parameters include:

- Number of Vehicles
- Vehicle Speed
- Vehicle Weight
- Polygon Area (non-editable—calculated by the interface)
- Distance Traveled (by one vehicle during the duration of the release).

As discussed in Section 2.2.2.5, DUSTRAN does not treat the motion of individual vehicles. Instead, a bulk approach is used to quantify dust emissions from vehicle activities. That is, the dust emissions from all vehicles active in an area over a specified time are assumed to be released uniformly from the area at a constant rate throughout the duration of the activity. Therefore, the input fields on the “Vehicle Parameters” form (Figure 4.11) should not be interpreted as the specific motion of individual vehicles; the form is simply an approach for providing the information needed by DUSTRAN in a bulk sense.

The approach for determining the distance traveled by a vehicle within a specific vehicle type (specific weight and speed) is to estimate the total distance traveled for all vehicles within a vehicle type throughout the duration of the activity and then divide this total distance traveled by the total number of vehicles within a vehicle type to get the average distance traveled for one vehicle. This average value is the distance traveled specified in the input window (Figure 4.11). The only constraint on the inputs to the window is that the average distance traveled for one vehicle not be greater than the vehicle speed × activity duration. It is probable that the distance traveled will be less than the vehicle speed × activity duration because of the likelihood that not all vehicles of a particular type will be active during the entire period. For example, some vehicles may move intermittently (at their specified speed) during the period because of their particular function.

As described in Section 2.3.1, the dust emissions from a moving vehicle are proportional to the vehicle momentum (i.e., vehicle weight × vehicle speed). Therefore, if some vehicles of one type travel at significantly different speeds than other vehicles of the same type, another vehicle type will need to be added to the list such that the other speed(s) can be specified.

For quality assurance, valid textbox entries will appear green and invalid textbox entries will appear red. A range of valid entries appears in the yellow label at the bottom of the form for a given textbox entry.

4.6.4 Selecting Existing Sources to Use in a Simulation

Multiple point, area, and line sources can be created for use in a simulation. After a source has been created, it will appear in a list on the “Sources” tab. To use the source in a simulation:

- Check the checkbox next to the name of the source. The source will be displayed in the MapWindow map display and will be used in the model simulation.
- Check the checkbox next to that source type to select all sources for a particular source type (i.e., point, line, or area).

4.6.5 View and Edit Existing Source Information

Existing point, line, and area sources appear in a source list on the “Sources” tab. To view and edit the input parameters for a particular source, double-click on the source’s name or highlight the source name and click on the “Information”  button. The “Source Input” form for that particular source will appear and can be reviewed and edited.

4.6.6 Deleting an Existing Source

Existing point, line, and area sources appear in the source list on the “Sources” tab. To permanently delete a source and its corresponding source information so that it is no longer available for use in a simulation, highlight the source's name and click on the "Delete" button. The source will be removed from the simulation and from the MapWindow map display.

4.6.7 Clearing All Existing Sources

Existing point, line, and area sources appear in the source list on the "Sources" tab. To permanently delete all sources and corresponding source information so that they are no longer available in the current simulation, click on the "New Scenario" button. The sources will be removed from the simulation and from the MapWindow map display.

4.6.8 Saving Existing Sources as a Scenario

Sources can be saved to a scenario that allows for the sources to be added to a simulation at a later time using the "Scenario" tab. A scenario is usually created after all sources and source parameters have been entered. To save the sources as a new scenario, click the "Save Scenario" button. The scenario will also be added to the scenarios list, indicating that it is available for selection at a later time.

4.6.9 Adding Characteristic Files (Soil and Vegetation for Wind-blown Dust)

Soil and vegetation files that use the Zobler soil textures and Olson vegetation classes to characterize the underlying domain can be added to DUSTRAN for use in a "Wind-blown Dust" simulation. These files are created automatically at the time of the site's creation (see "Adding a New User Profile to DUSTRAN," Section 6.0) and must be associated with the scenario when performing a "Wind-blown Dust" simulation. Alternatively, high-resolution soil and vegetation data files can be generated from Environmental System Research Institute shape files using the "Polygon Layer Creator" discussed in Section 5.0. These detailed files can be used to more precisely define the underlying soil and vegetative surface characteristics for use in the wind-blown dust-emissions model.

To associate soil and vegetation characteristic files for use in a "Wind-blown Dust" simulation:

- Click the "Add Characteristic File" button. The "Characteristics Priorities" form, as shown in Figure 4.12, will appear. This form allows for files from the "Polygon Layer Creator" (see Section 5.0, "Polygon Layer Creator") to be associated with the current scenario. Currently, two categories—Soils and Vegetation—are predefined and contain references to the standard Zobler soil texture and Olson Ecosystem class files created by the "Add Site" wizard within DUSTRAN.
- Click the "Add," "Higher," or "Lower" buttons for a given category to add a new characteristic file or to increase or decrease the file's usage priority. Generally, high-resolution files should be ranked higher in the list. A checkbox next to a file's name means the file will be used in the current simulation.
- Click "OK" to save the characteristic file selections or "Cancel" to ignore the most recent changes.

The "Polygon Layer Creator" button is used to launch the Polygon Layer Creator. This application allows for either manual or automatic (through Environmental System Research Institute shape files) creation of characteristic files for use in DUSTRAN (see Section 5.0, "Polygon Layer Creator"). Although the existing soils and vegetation classification files are sufficient for running a "Wind-blown Dust" simulation in DUSTRAN, higher-resolution files can be associated with each category to provide better resolution. These files can be created within the "Polygon Layer Creator" tool, either manually or through user-specified shape files that use Zobler or Olson classification schemes.

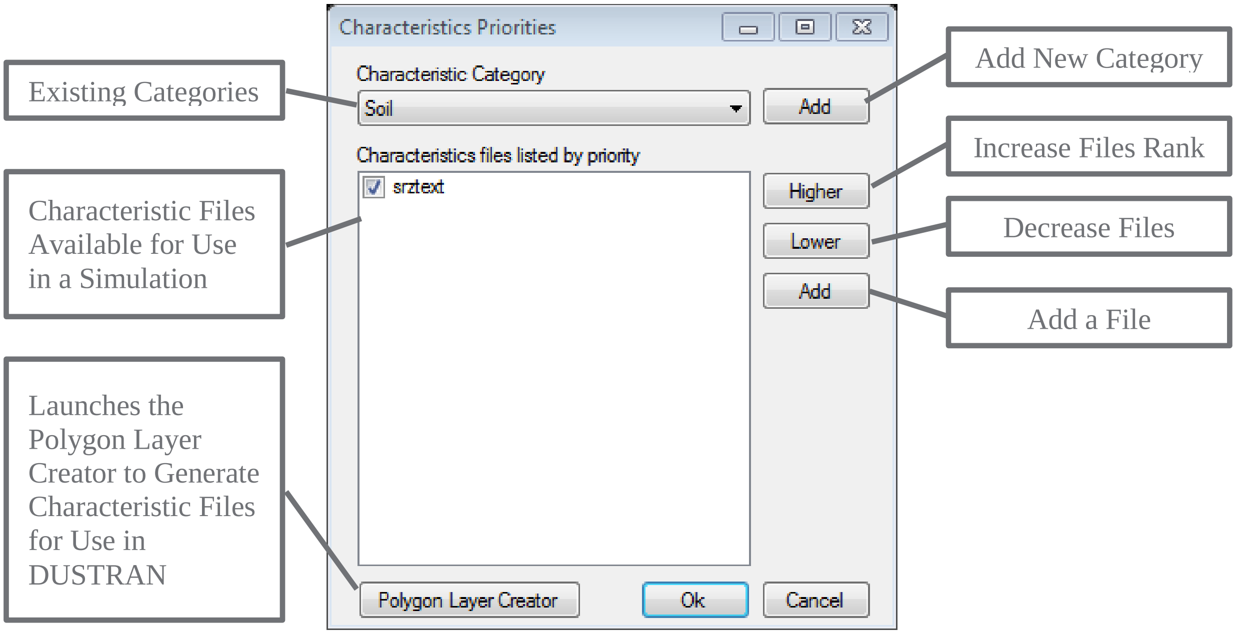


Figure 4.12. Characteristics Priorities Dialog Box for Adding Files, such as Soil and Vegetation Classes, in DUSTRAN

4.7 Scenarios Tab

The "Scenarios" tab is used to open or delete an existing scenario from the simulation. A scenario contains previously saved source information, including each source's location and parameter settings. A scenario gets created from the "Sources" tab (Note: To create a scenario, see Section 4.6, "Sources Tab"). When selected, the "Scenarios" tab displays the current scenario name as well as a list of currently available scenarios for the site. Figure 4.13 shows the "Scenarios" tab with an example scenario entered.

4.7.1 Adding an Existing Scenario

Previously saved sources (i.e., a scenario) can be opened for use in the current model simulation. To open a scenario:

- Click on the "Scenarios" tab. Highlight the scenario you wish to add by clicking on its name (Note: If no sources have been previously saved as a scenario, there will be no scenarios available in the list).
- Click the "Add Scenario"  button, and the scenario will be added to the simulation. All sources in the scenario will be listed on the "Sources" tab and displayed in MapWindow.

Multiple scenarios can be added to a simulation by following the steps above.

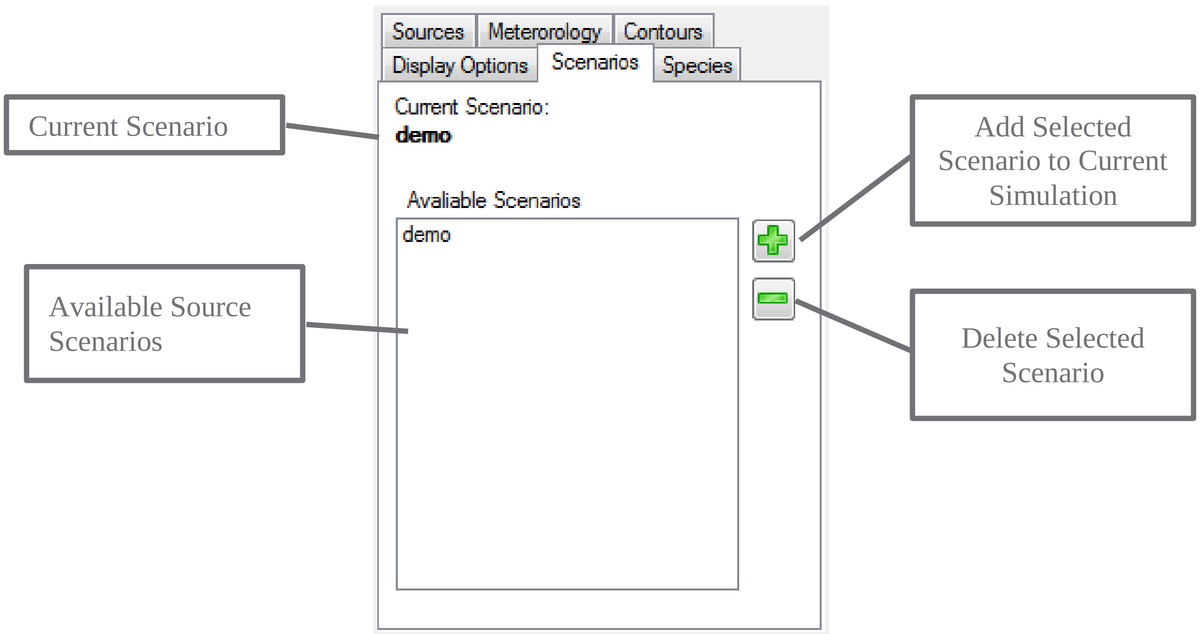


Figure 4.13. Example Scenario Tab with Controls Labeled

4.7.2 Deleting an Existing Scenario

Deleting a scenario permanently deletes all source and source-parameter information stored in that scenario from the site. Once the scenario is deleted from the site, it is no longer available for use in future simulations. To delete a scenario:

- Click on the "Scenarios" tab. Highlight the scenario you wish to delete by clicking on its name.
- Click the "Delete Scenario"  button, and the scenario will be deleted from the list.

4.8 Meteorology Tab

The "Meteorology" tab is used to select the surface and upper-air meteorological data sources for use in a simulation. Two options exist for specifying the meteorological data source in DUSTRAN:

- Single Observation—for specifying meteorological conditions on a form within the DUSTRAN interface. These observations are for a single location (center of the domain) and persist for the duration of the simulation.
- User Defined—for specifying preformatted CALMET surface and upper-air observation files for direct use in DUSTRAN.

Each of these options is discussed further in the following sections.

4.8.1 Selecting Single Observation Meteorology

Single meteorological observation data can be used for any of the available simulation types by selecting "Single Observation" from the "Use" list on the "Meteorology" tab. A form will appear, allowing for the entry of standard meteorological parameters that apply to a single point on the domain (i.e., the center of the domain). The single observation data input form displayed will allow for the entry of meteorological data for each hour of the simulation. Single observation data required by a simulation will vary depending upon the simulation type selected with AERMOD simulations requiring different inputs than the CALPUFF or CALGRID models. Figure 4.14 and Figure 4.15 provide examples of the meteorological input forms used for CALPUFF and AERMOD, respectively.

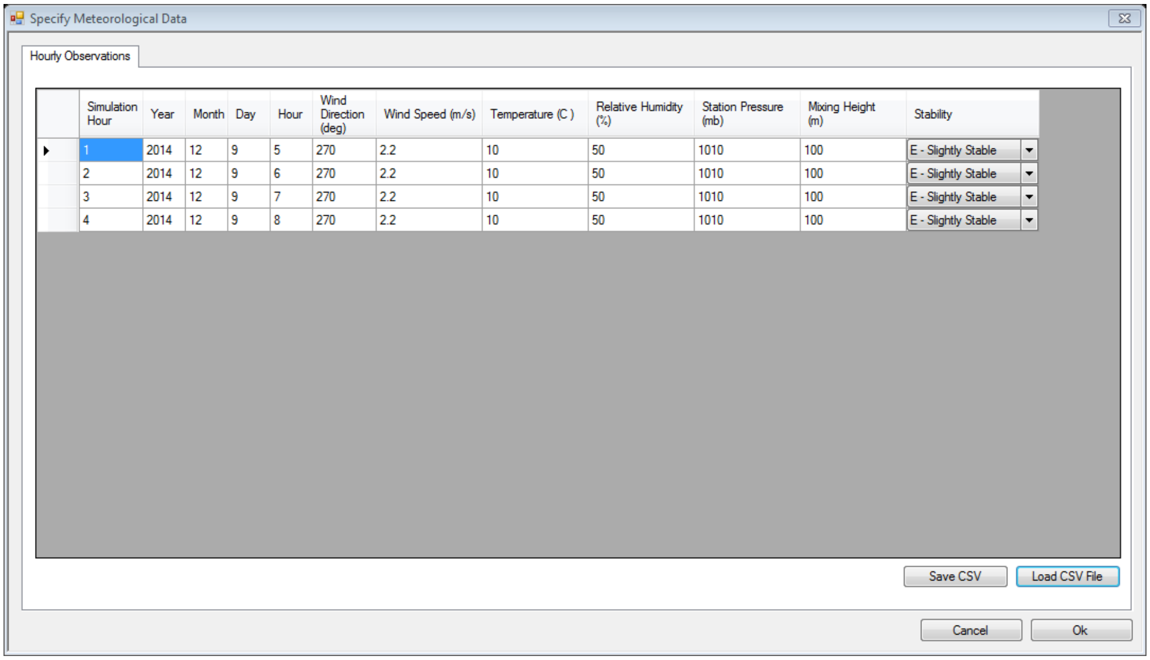


Figure 4.14. Example Meteorological Data Input Form for a CALPUFF simulation

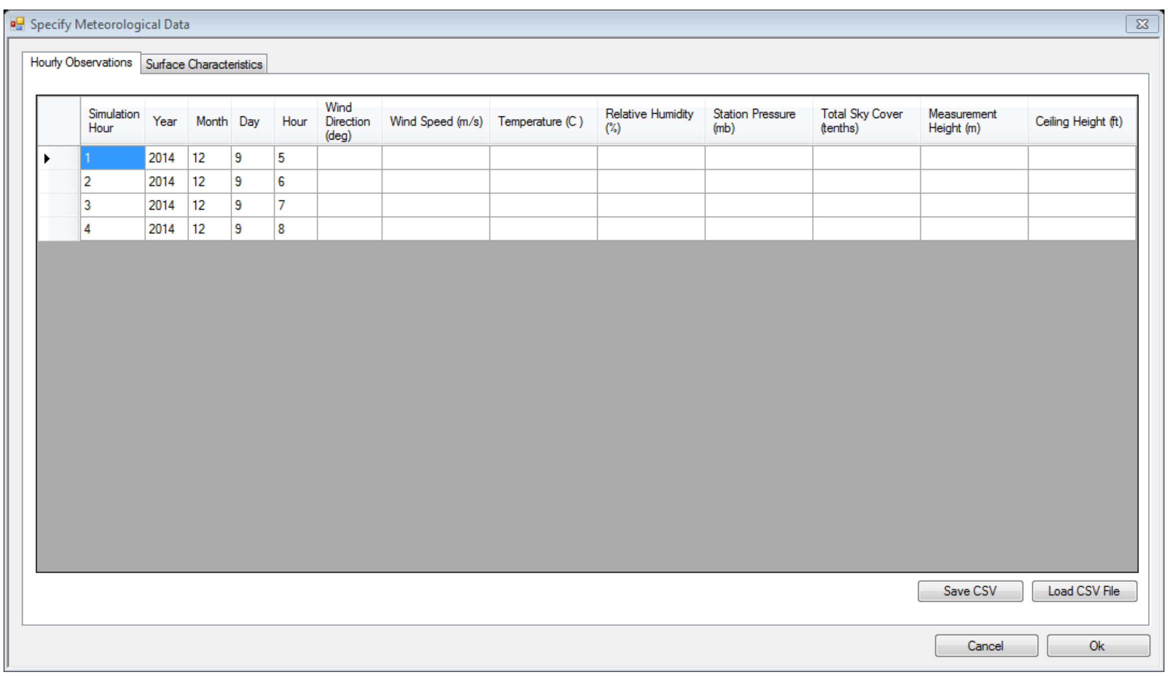


Figure 4.15. Example Meteorological Data Input Form for an AERMOD simulation

To use a single observation in a DUSTRAN simulation:

- Select “Single Observation” from the “Meteorology” tab.
- Enter the meteorological observations on the “Specify Meteorological Data” form and then click “OK” to continue.
- Data entered into the form may be saved out to a .csv file using the “Save CSV” option.
- Data saved from an earlier simulation may be loaded into the form using the “Load CSV File” option.

For each simulation using the AERMOD model, surface characteristic data must be entered along with the meteorological data. Clicking on the “Surface Characteristics” tab of the “Specify Meteorological Data” form will display the “Surface Characteristics” input panel. Figure 4.16 is an example of the “Surface Characteristics’ form with data entered. Segments within the form are used to define the different land-use surface characteristics associated with the simulation site. The number of segments that need to be filled depends on the site; the form allows for multiple segments or an individual segment that defines the entire simulation area. While the form allows any number of segments to be entered, enough segments must be entered to cover 360 degrees. The chart at the top of the form displays the segments and their degrees of coverage as they are entered by the user. Besides allowing the user to enter values directly into the form, data can also be entered by using lookup tables. The first method of accessing the lookup table is to click on the data box with the right button of the mouse, which brings up a menu version of the lookup table data. The second is to click on the “Lookup Tables” button, which will display the complete lookup tables in a separate form. Figure 4.17 shows an example lookup table. Clicking on a value in either of these lookup tables will automatically place the value in the input form.

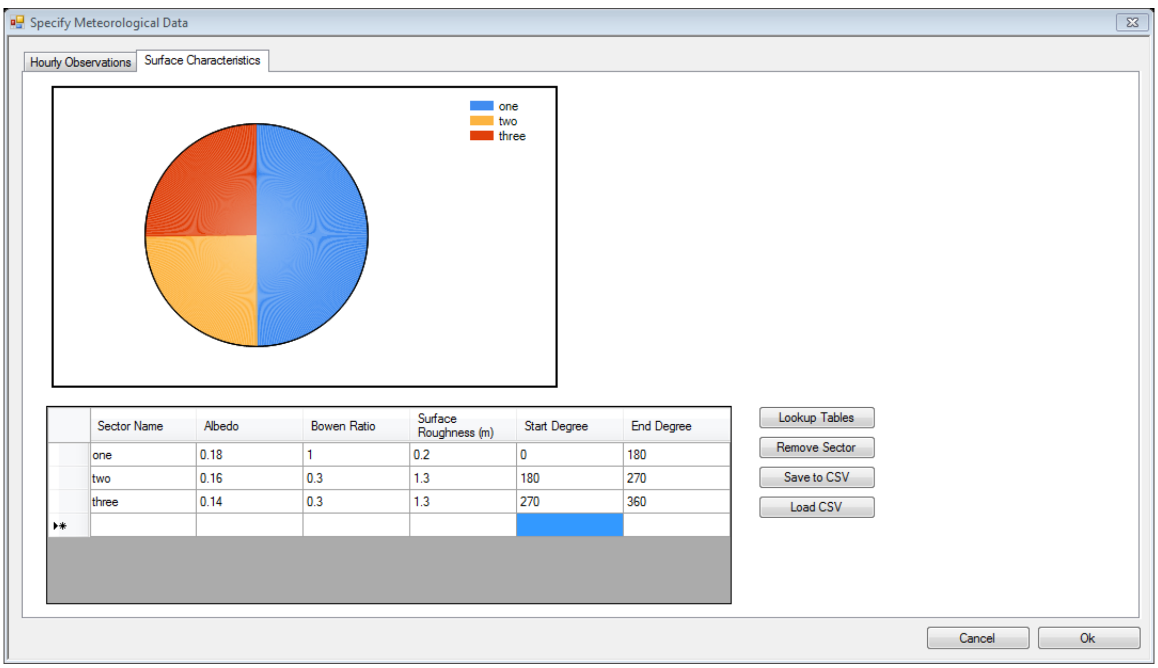
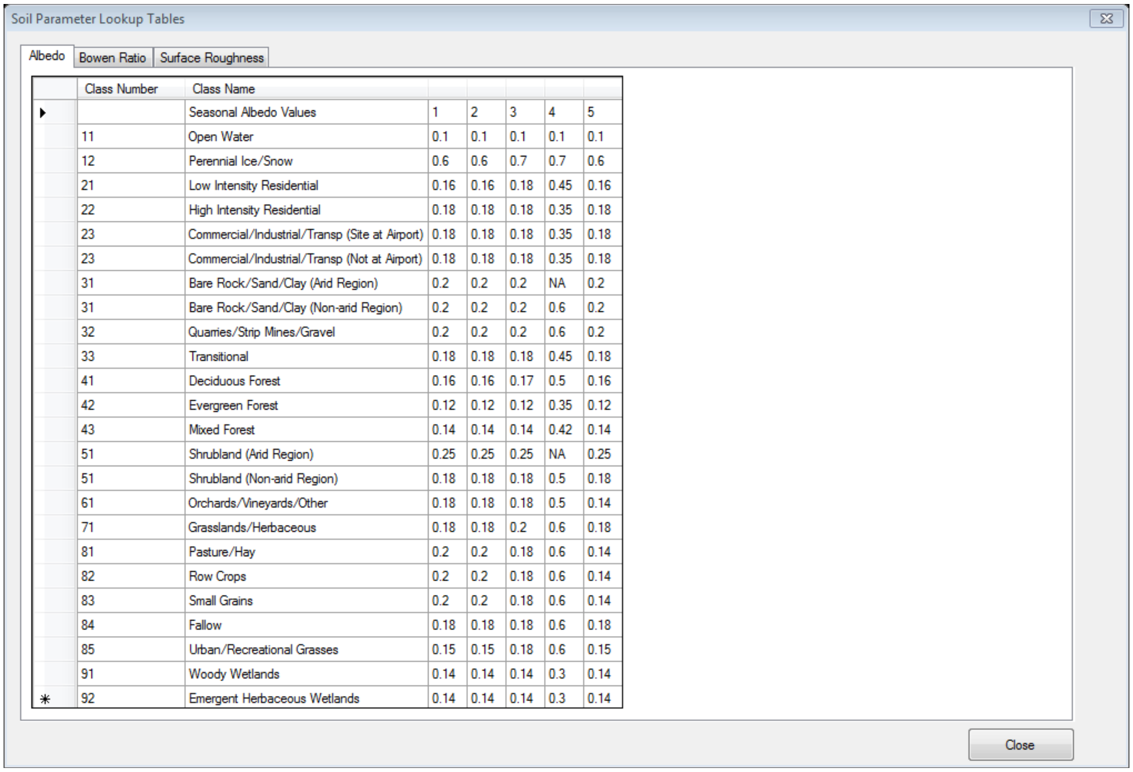


Figure 4.16. AERMOD Surface Characteristics Input Form



Class Number	Class Name					
	Seasonal Albedo Values	1	2	3	4	5
11	Open Water	0.1	0.1	0.1	0.1	0.1
12	Perennial Ice/Snow	0.6	0.6	0.7	0.7	0.6
21	Low Intensity Residential	0.16	0.16	0.18	0.45	0.16
22	High Intensity Residential	0.18	0.18	0.18	0.35	0.18
23	Commercial/Industrial/Transp (Site at Airport)	0.18	0.18	0.18	0.35	0.18
23	Commercial/Industrial/Transp (Not at Airport)	0.18	0.18	0.18	0.35	0.18
31	Bare Rock/Sand/Clay (Arid Region)	0.2	0.2	0.2	NA	0.2
31	Bare Rock/Sand/Clay (Non-arid Region)	0.2	0.2	0.2	0.6	0.2
32	Quarries/Strip Mines/Gravel	0.2	0.2	0.2	0.6	0.2
33	Transitional	0.18	0.18	0.18	0.45	0.18
41	Deciduous Forest	0.16	0.16	0.17	0.5	0.16
42	Evergreen Forest	0.12	0.12	0.12	0.35	0.12
43	Mixed Forest	0.14	0.14	0.14	0.42	0.14
51	Shrubland (Arid Region)	0.25	0.25	0.25	NA	0.25
51	Shrubland (Non-arid Region)	0.18	0.18	0.18	0.5	0.18
61	Orchards/Vineyards/Other	0.18	0.18	0.18	0.5	0.14
71	Grasslands/Herbaceous	0.18	0.18	0.2	0.6	0.18
81	Pasture/Hay	0.2	0.2	0.18	0.6	0.14
82	Row Crops	0.2	0.2	0.18	0.6	0.14
83	Small Grains	0.2	0.2	0.18	0.6	0.14
84	Fallow	0.18	0.18	0.18	0.6	0.18
85	Urban/Recreational Grasses	0.15	0.15	0.18	0.6	0.15
91	Woody Wetlands	0.14	0.14	0.14	0.3	0.14
92	Emergent Herbaceous Wetlands	0.14	0.14	0.14	0.3	0.14

Figure 4.17. Surface Characteristics Lookup Table

4.8.2 Selecting User-Defined Meteorology

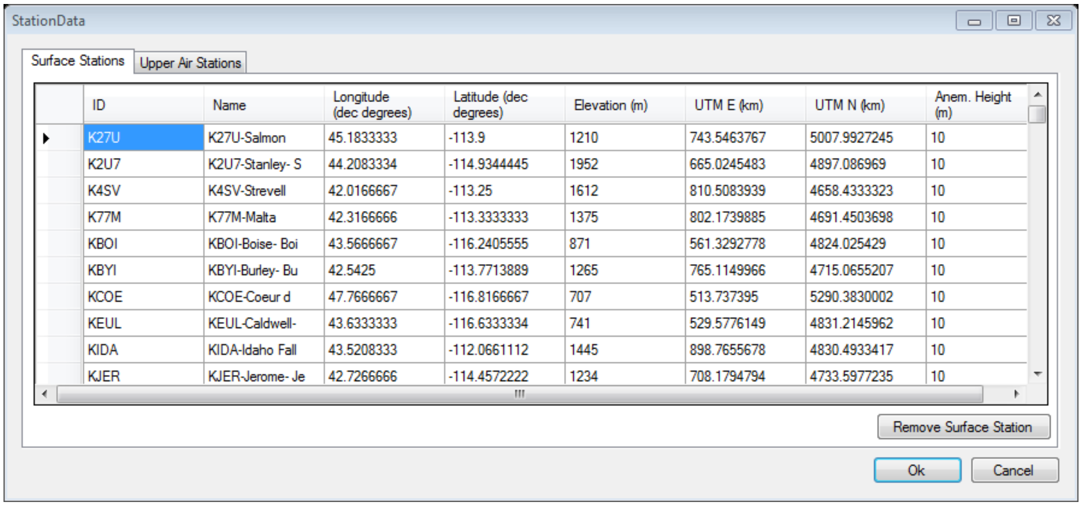
User-defined meteorological data can be used in a CALPUFF or CALGRID simulation by selecting "User Defined" from the "Use" list on the "Meteorology" tab. This option requires the surface and upper-air meteorological data to already exist and to be in the correct format for CALMET. Specifically, the user must have created the CALMET surface (SURF.dat) and upper-air (UP.dat) files using a processing utility exterior to the DUSTRAN interface. For more information on the format of the meteorological input files and processing utilities for CALMET, refer to "A User's Guide for the CALMET Meteorological Model" (Scire et al. 2000b). **Note**: this option is not available for simulations using AERMOD.

To associate existing CALMET-ready meteorological data files with DUSTRAN, two files must be created—a .snf (surface) and a .unf (upper-air) file. These are DUSTRAN-specific files and contain station metadata, such as station name, ID, and coordinate location. To create a .snf and .unf to associate with existing CALMET SURF.dat and UP.dat files:

- Select "User Defined" from the "Meteorology" tab.
- Click the "Change Met Input Directory" to specify the directory where the CALMET-ready SURF.dat and UP.dat data files reside.
- Click on the "Edit Met Files" button. Select the "Surface Stations" tab and enter each station's Name, ID, Longitude, Latitude, Elevation, and UTM Easting and Northing coordinates. Note that the station ID must agree with the station ID(s) specified in the CALMET-ready SURF.dat file.

- Select the "Upper Air Stations" tab and enter each station's Name, ID, Longitude, Latitude, Elevation, and UTM Easting and Northing coordinates. Note that the station ID must agree with the station ID(s) specified in the CALMET-ready UP.dat file.
- Click "OK" on the "Edit Met Station Data" form.

Figure 4.18 shows a sample "Surface Stations" tab that is used to specify the station information that is associated with the CALMET-ready SURF.dat file. The "Upper Air Stations" tab (not shown) is a similar form for entering upper-air station information that is associated with the CALMET-ready UP.dat files.



ID	Name	Longitude (dec degrees)	Latitude (dec degrees)	Elevation (m)	UTM E (km)	UTM N (km)	Anem. Height (m)
K27U	K27U-Salmon	45.1833333	-113.9	1210	743.5463767	5007.9927245	10
K2U7	K2U7-Stanley- S	44.2083334	-114.9344445	1952	665.0245483	4897.086969	10
K4SV	K4SV-Strevell	42.0166667	-113.25	1612	810.5083939	4658.4333323	10
K77M	K77M-Malta	42.3166666	-113.3333333	1375	802.1739885	4691.4503698	10
KBOI	KBOI-Boise- Boi	43.5666667	-116.2405555	871	561.3292778	4824.025429	10
KBYI	KBYI-Burley- Bu	42.5425	-113.7713889	1265	765.1149966	4715.0655207	10
KCOE	KCOE-Coeur d	47.7666667	-116.8166667	707	513.737395	5290.3830002	10
KEUL	KEUL-Caldwell-	43.6333333	-116.6333334	741	529.5776149	4831.2145962	10
KIDA	KIDA-Idaho Fall	43.5208333	-112.0661112	1445	898.7655678	4830.4933417	10
KJER	KJER-Jerome- Je	42.7266666	-114.4572222	1234	708.1794794	4733.5977235	10

Figure 4.18. Sample "Surface Stations" Tab for Specifying CALMET-Ready Station Locations in DUSTRAN

4.9 Contours Tab

The "Contours" tab is used to select the contours of interest to view for a given emission type and time period after completing a simulation in AERMOD, CALPUFF, or CALGRID. Various contour types can be displayed, including concentration, exposure (i.e., time-integrated concentration), deposition, and total deposition for the simulated averaging period. Contours can be animated over the simulated period to provide a dynamic perspective of the plume advection and diffusion pattern. Figure 4.19 displays an example of the "Contours" tab following a simulation run.

4.9.1 Setting the Contour Type

After completing a simulation, various contour types are available for display in the MapWindow map display. To display the contour type of interest, select the "Contours" tab and select the desired "Contour Type" from the listbox. The dispersion model that generated the contours precedes the contour type name. Contour types include:

- "Conc"—the average air concentration. If "Conc" is followed by "AVG" and an integer value, then the concentration is for the averaging period defined by the integer.
- "Dep"—the average deposition. If "Dep" is followed by "AVG" and an integer value, then the deposition is for the averaging period defined by the integer.
- "Exp"—the total exposure (i.e., time-integrated concentration).
- "TotalDep"—the total accumulated deposition.

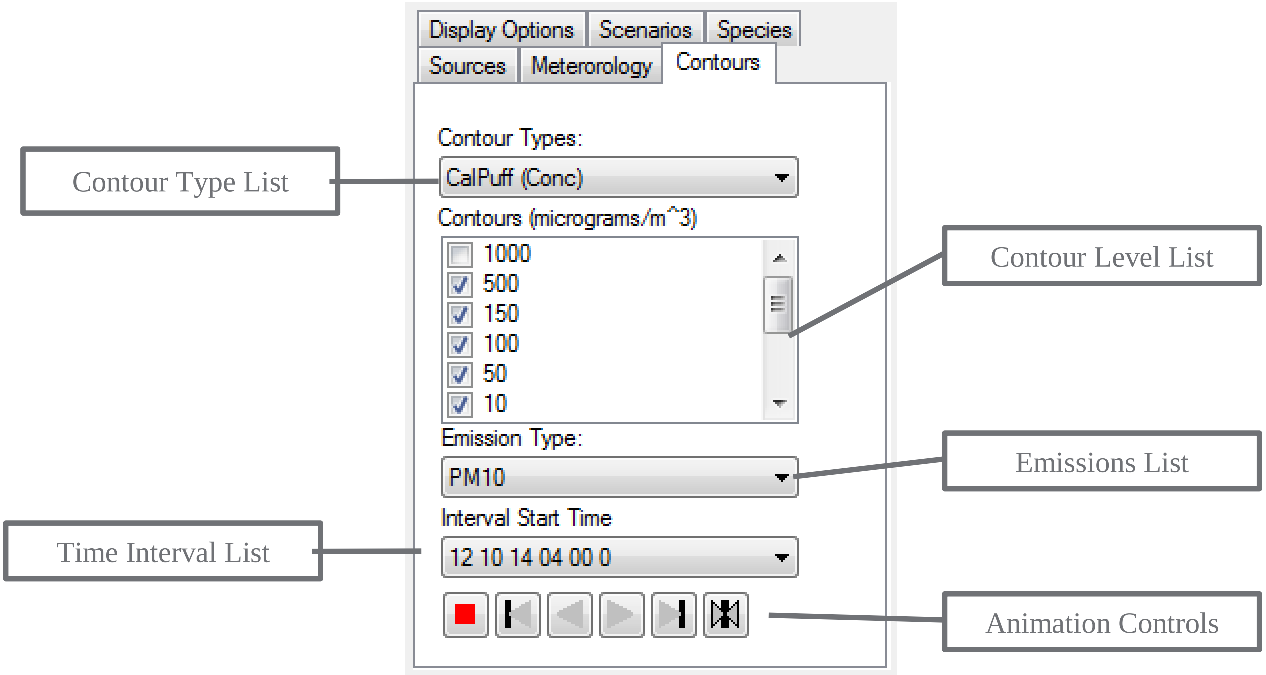


Figure 4.19. Example Contours Tab with Controls Labeled

4.9.2 Selecting Contour Levels

After completing a simulation, various contour levels are available for display. In general, these levels span many orders of magnitude to allow the user to clearly define the concentration or deposition footprint for a given scenario. To view the contour levels, select the "Contours" tab and check the desired levels within the "Contours" list to be displayed in the MapWindow map display. Uncheck any levels to remove them from display in the MapWindow map display. Typically, a plume ground-level footprint can reasonably be described over a range of 3-to-5 orders of magnitude. Displaying more than 5 orders of magnitude is primarily used to evaluate the model computations rather than having any semblance to a real plume. The number of contour levels and values of contour levels computed by DUSTRAN are specified in the Cal.par file described in Section A.2. Currently, DUSTRAN is set up for computing a concentration range spreading 12 orders of magnitude, which reasonably brackets expected concentrations over a range of emission and dispersion rates.

4.9.3 Selecting Emission Type

After completing a simulation, contours for a given emission type are available for display. The available emission type(s) correspond to the species selected on the "Species" tab before the model simulation. To set the emission type to display, click on the "Contours" tab and select a desired species from the "Emission Type" listbox. The contours for that particular emission type will be displayed in the MapWindow map display.

4.9.4 Selecting Interval Start Time

After completing a simulation, concentration and deposition contours can be displayed for any hourly time interval in the simulated period. To view the results for a particular time interval, select the "Contours" tab and choose the desired time interval from the "Interval Start Time" listbox. The listed times are in the following format: mm dd yy hh min sec, where mm is month, dd is day, yy is year, min is minutes, and sec is seconds. Averaged hourly results are referenced to the beginning of the "Interval Start Time." For example, to view the 0900 to 1000 hourly average concentration, select 0900 from the interval listbox.

4.9.5 Animating Contours

After completing a simulation, the displayed contours can be animated within the MapWindow map display to provide a dynamic perspective of the concentration or deposition pattern in time. The "Contours" tab contains a series of buttons to control the animation for the simulation. Figure 4.20 displays the animation control with each button labeled. The buttons (from left to right, in Figure 4.20) perform the following functions:

- Stop : Stops the current animation at a given interval time step.
- Reset  : Resets the contour animation to the first time step in the simulation.
- Back : Steps the contour animation back one time interval.
- Forward : Advances the contour animation one time interval.
- Run Once : Runs contour animation once, sequentially displaying each interval in the simulation once.
- Loop : Continuously loops the contour animation.

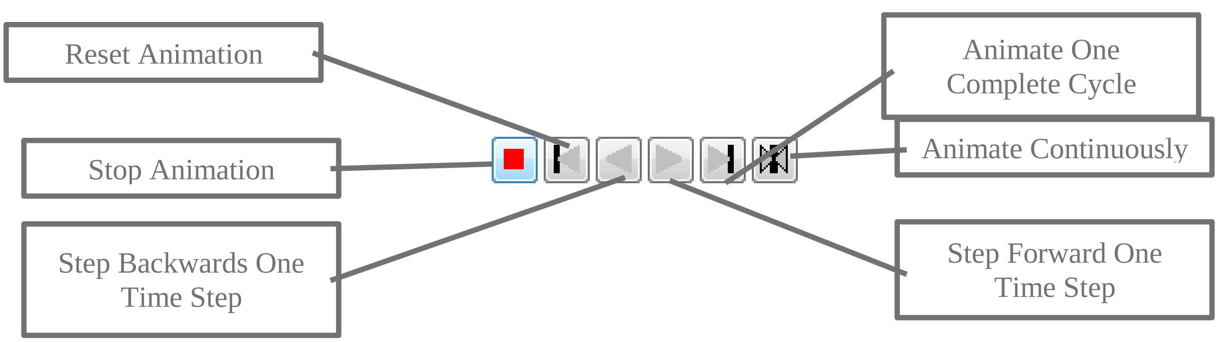


Figure 4.20. Animation Control with Buttons Labeled

4.10 Display Options Tab

The "Display Options" tab allows for controlling the rendering of certain items within the MapWindow map display, including contours, receptors, wind vectors, surface meteorological stations, and upper-air station locations. In addition, the appearance, such as the symbol size, shape, or color, for some of the items can be adjusted. This tab is usually visited after making a DUSTRAN simulation to control which objects to display in the MapWindow map display. Figure 4.21 is an example of the "Display Options" tab showing the available items for displaying in MapWindow after completing a simulation. The various "Display Options" are described in the following sections.

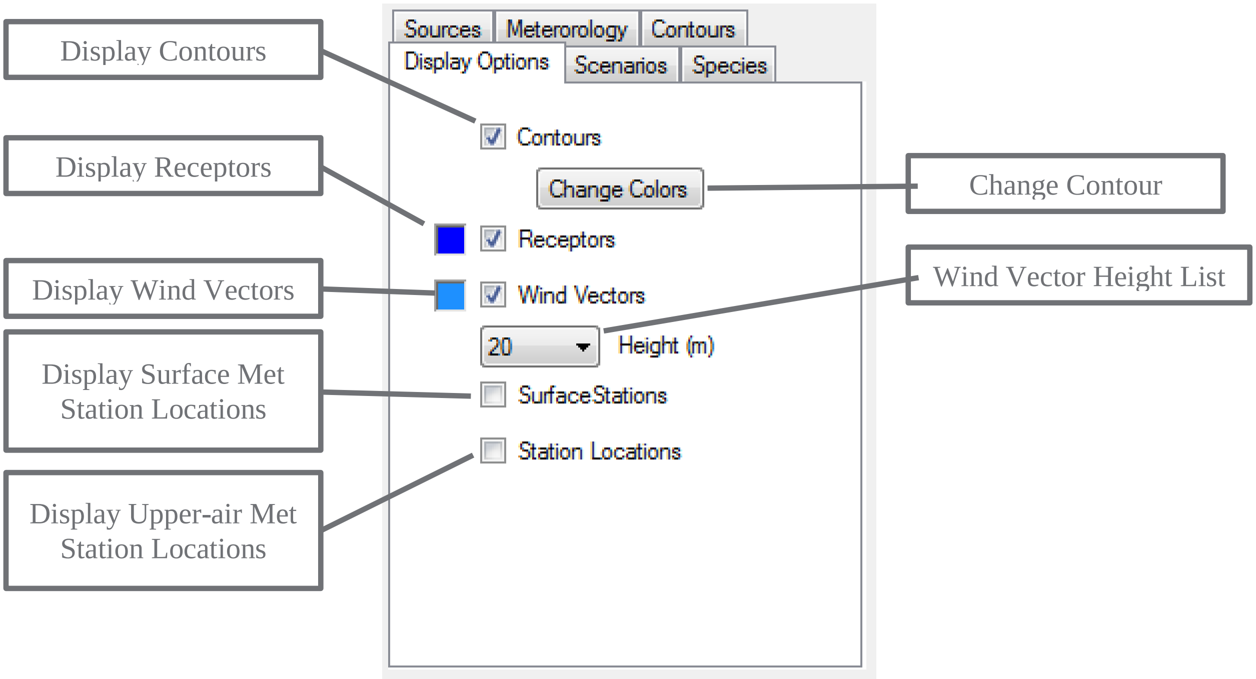


Figure 4.21. Display Options Tab

4.10.1 Displaying the Contour Results

The contours generated by the simulation can be viewed in the MapWindow map display by clicking on the checkbox labeled "Contours" on the "Display Options" tab. The colors used to display the different contouring levels can be changed by clicking on the "Change Colors" button which will bring up the "Change Contour Colors" form (See Figure 4.22). Clicking on the desired contour level will then cause a color selection form to appear which can be used to change the display color of the selected contour level.

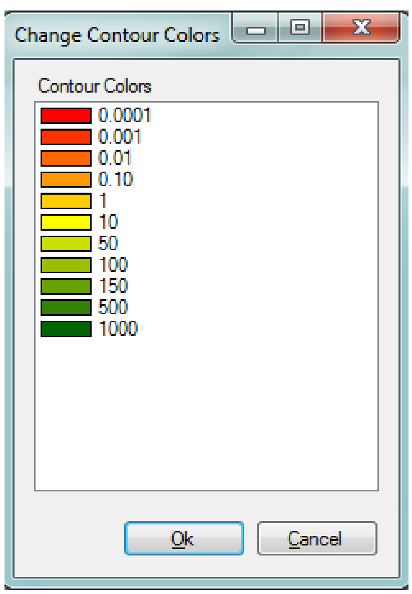


Figure 4.22. Contour Color Selection Form

4.10.2 Displaying the Receptor Network

The receptor field—where concentration and deposition values are calculated—can be viewed in the MapWindow map display by clicking on the checkbox labeled “Receptors” on the “Display Options” tab. In addition, the color, size, and other properties of the displayed receptors can be adjusted by clicking on the colored box next to the checkbox.

4.10.3 Displaying the Calculated Wind Vector Field

The gridded wind vector field created by the CALMET meteorological model and used in the dispersion calculations can be viewed in the MapWindow map display by clicking on the checkbox labeled “Wind Vectors” on the “Display Options” tab. Wind vectors can be displayed for various heights above the ground (in meters) by selecting a height from the listbox beneath the “Wind Vectors” label. In addition, the color, size, and other properties of the wind vectors can be adjusted by clicking on the colored box next to the checkbox. The number of wind vectors plotted and the size (i.e., scale) of the vectors can be adjusted within the Cal.par file. By default, wind vectors are displayed at every other receptor location on the primary grid and are scaled such that a 3 m/s wind is exactly the distance between successive receptor locations.

4.10.4 Displaying Surface and Upper-Air Meteorological Station Locations

Surface and upper-air meteorological stations that were used in the CALMET meteorological model for creating the gridded meteorological fields can be displayed in the MapWindow map display by clicking on the checkbox labeled “Surface Stations” and “Station Locations,” respectively, on the “Display Options” tab. In addition, the color, size, and other properties of the stations can be adjusted by clicking on the colored boxes next to their respective checkboxes.

5.0 **Polygon Layer Creator**

The Polygon Layer Creator is a software application that can be used to create polygon-based GIS data for use in a DUSTRAN scenario. Using the .mwprj project file for a DUSTRAN site, a user can draw areas onto the map of the site and assign characteristic codes to each of the polygon areas through simple point-and-click operations. Once new areas have been drawn or existing areas have been modified, the utility can be used to generate a new shape file containing the changes as well as a .csv file. The text file can be used directly by the DUSTRAN modeling system for retrieving certain land-based characteristics, such as soil textures and vegetation classes, for use in wind-blown dust simulations. The interface for the Polygon Layer Creator consists of three parts: a map display, a table of contents for the layers displayed in the map, and a toolbar with buttons for use in controlling the map display and editing the polygons. Figure 5.1 displays an example of the Polygon Layer Creator interface with a sample site .mwprj file loaded.

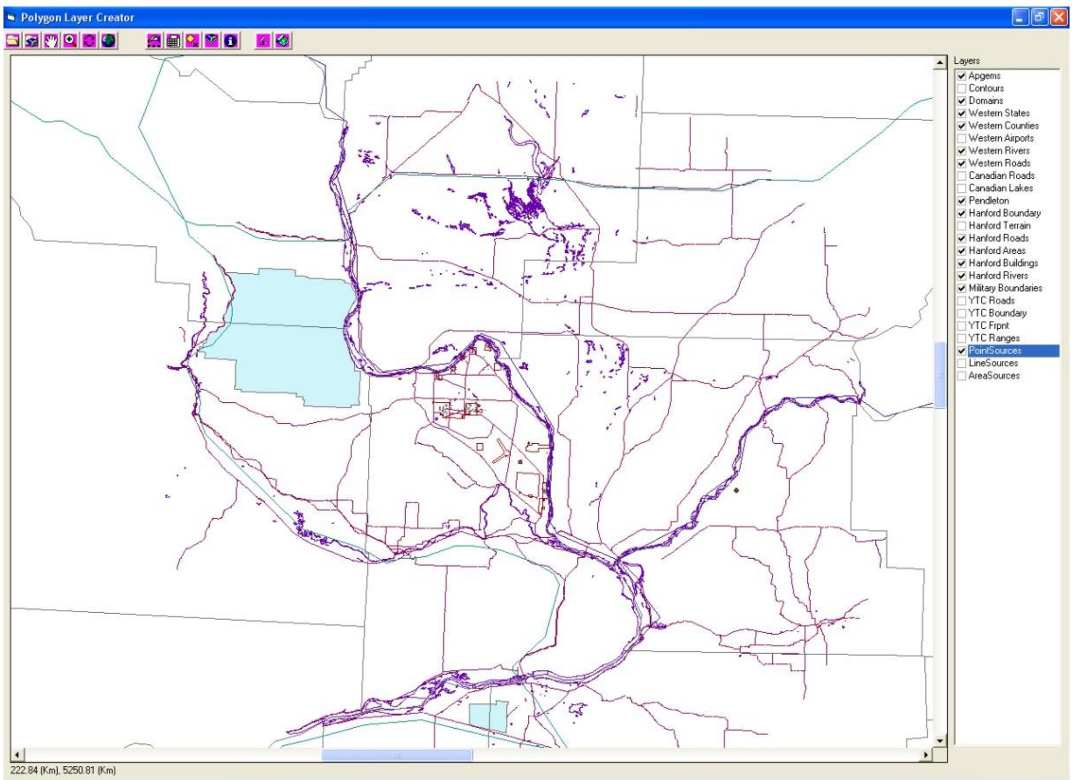


Figure 5.1. Polygon Layer Creator Interface

5.1.1 Starting the Polygon Layer Creator

The Polygon Layer Creator can be accessed from both within and outside of the DUSTRAN interface. To access the Polygon Layer Creator from within DUSTRAN:

- Select the "Sources" tab and click on the "Characteristic Files"  button. Characteristic files are .csv files created by the Polygon Layer Creator that describe certain characteristics of the domain, such as soil textures and vegetation classes. These files can be read directly by the DUSTRAN interface for calculation purposes, such as calculating gridded dust emissions for the model domain.

- The “Characteristic Priorities” (Figure 5.2) form will appear, from which the “Polygon Layer Creator” button can be clicked to launch the application. The “Characteristic Priorities” form is used to associate the files created in Layer Creator with the current scenario.

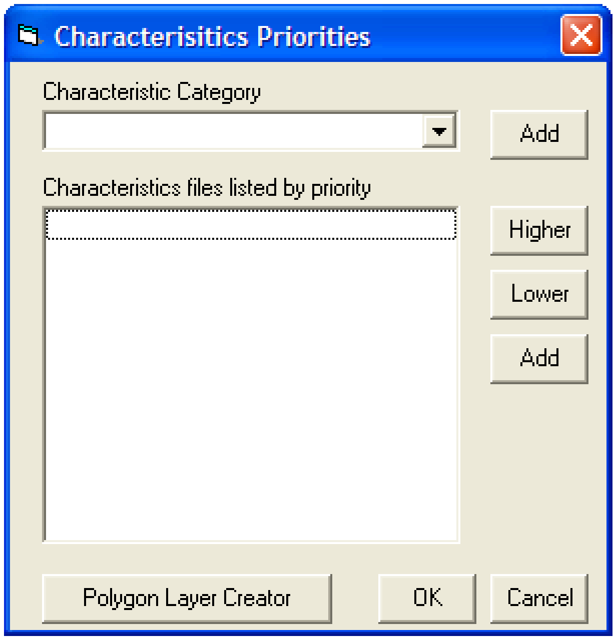


Figure 5.2. Launching Polygon Layer Creator from Within DUSTRAN

The Polygon Layer Creator can also be accessed outside of the DUSTRAN interface:

- To start the Polygon Layer Creator, double-click on the CreateLayer.exe file in the main DUSTRAN directory.

Once started, the application’s interface will be displayed with all buttons disabled except for the “Load shapefile” button.

5.1.2 Loading a Shapefile

A GIS-based shapefile (.shp) can be loaded into the layer creator interface to provide a reference background on which to either create, add, or modify polygons representing the soil characteristics of the simulation site. To load a shapefile:

- Click on the “Load Shapefile file” button.
- Using the browse window, navigate to the desired DUSTRAN site directory and select the .shp file representing the simulation site.
- Click on the “Open” button of the browse window to open the .shp file and close the browse window.
- Once the shapefile has been opened, the selected map layers will be automatically loaded into the map display and the layer names will be displayed in the “Layers” list.

5.1.3 Importing Polygons from an Existing Shape File

Shapefiles can be loaded into the “Polygon Layer Creator” utility to quickly create polygon shapes for use in DUSTRAN. These shape files typically describe characteristic features for a site, such as soil textures or vegetation classes. To import polygons from a shape file:

- Click on the “Import polygons from shape file” button.
- Using the browse window, navigate to the directory folder that contains the desired shape file.
- Click on the shape file name and then click on the “Open” button to load the shape file and close the browse window.
- Once the shape file has been selected, a window will be displayed listing the field names available in the shape file.
- Select the primary field name that corresponds to the polygon values that will be used to describe the shapes. Figure 5.3 displays an example of the field selection window.
- Once opened, the shape file will be automatically loaded into the map display and its name added to the “Layers” list.

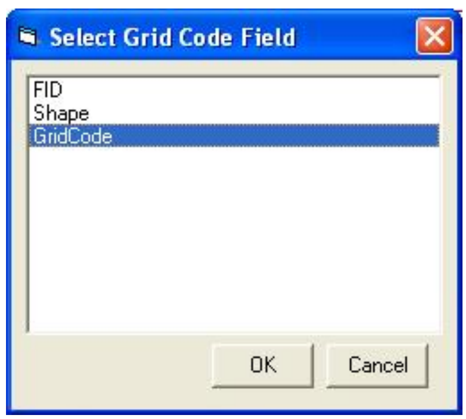


Figure 5.3. Select Grid Code Field Window

5.1.4 Starting a New Polygon Set

After a site’s .mwprj file has been loaded, polygons can be created to describe certain underlying characteristics of a site. To clear all existing polygons from a site and start a new polygon set:

- Click on the “Start new polygon set” button to clear the current set of polygons and start a new set.
- Once the “Start new polygon set” button has been clicked, the previously selected polygons will be cleared, and new polygons can be entered.

5.1.5 Site Navigation Within Map Display Window

Standard navigation buttons are included within the Polygon Layer Creator for working with a site in the map display window. These navigation features include:

- Panning

- Zooming
- Reverting to Prior Extent
- Viewing the Full Extent.

5.1.5.1 Panning the Map Display

Panning is used to move around a site without changing the zoom. To pan the view in the map display:

- Click on the “Pan map image”  button.
- Click and hold on a point in the viewable map display.
- Move the mouse to move the map view displayed in the map control.

5.1.5.2 Zooming into the Map Display

Zooming is used to focus on a particular location within a site. To zoom on the map display:

- Click on the “Zoom into map image”  button.
- Using the mouse, click on a point in the map display and then drag a rectangle around the area you would like to zoom into.
- Release the mouse button, and this will zoom the extent of the viewable map to the rectangle drawn.

5.1.5.3 Reverting the Map Display to the Last Extent

Reverting the map display to the last extent is used to undo a particular zoom level. To revert to the extent for the map display:

- Click on the “Zoom to last extent”  button. The map display will automatically revert to the last extent used.

5.1.5.4 Returning the Map Display to the Full Extent

The full extent of the map display, as defined by the layers within the .mwprj file, is an efficient way to quickly view the entire extent of the site. To view the full extent:

- Click on the “Zoom to full map extent”  button. The map display will automatically adjust to the full extent of the map and refresh the display.

5.1.6 Working with Polygons

The Polygon Layer Creator allows for the creation of polygon areas that spatially represent a certain characteristic, such as soil textures or vegetation classes, for a site. Each polygon is assigned a value, called a grid code, which identifies it from surrounding polygons. For example, soil textures may be assigned a grid code ID, which identifies it by Zobler soil category (see Table 2.7). These grid codes can be read by DUSTRAN and used in certain calculations. Currently, DUSTRAN can read gridded Zobler soil texture IDs and Olson Ecosystem vegetation codes (see Section 2.3.2), both of which are used in the wind-blown dust-emissions module. The Polygon Layer Creator creates gridded values of these codes from the polygon shapes that are displayed in the map window and writes them to a text file for use in DUSTRAN. This section provides guidance on how to work with polygon shapes within the Polygon Layer Creator.

5.1.6.1 Adding Polygons to the Layer

Polygons can be manually drawn and added to a layer to create a collection, or set, of polygons. These polygons are used to represent a certain characteristic of the site, such as a specific soil texture or vegetation class. Polygons are differentiated by their grid code, which is a value that is entered after drawing the polygon shape. To add a new polygon to a polygon set:

- Click on the "Add polygons" button.
- Draw the new polygon onto the map display using the mouse. Each corner of the polygon is created by a left-click of the mouse at the desired location.
- When finished drawing the desired polygon, double-click on the last point of the polygon. An input window will be displayed asking for the code value associated with the polygon. Figure 5.4 displays an example of the input window.
- Enter the value into the input window and click on the "OK" button.
- The new polygon will then be added to the map display.

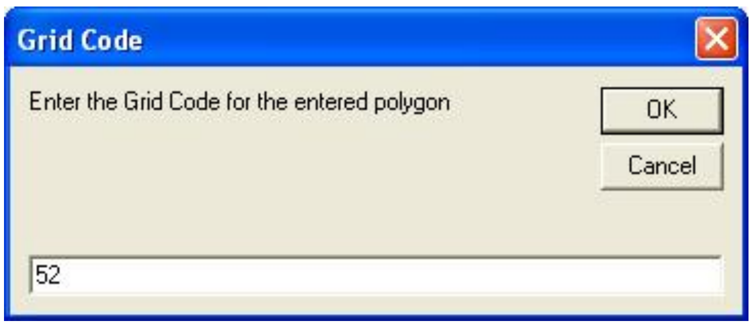


Figure 5.4. Grid Code Input Window

5.1.6.2 Editing the Value Associated with a Polygon

Each polygon has an associated value which identifies a particular attribute of the polygon. These values are assigned at the time the polygon is created, but can be edited. To edit a value associated with a particular polygon:

- Click on the "Edit layer polygon" button.
- Double-click on the polygon whose value is to be changed.
- Once clicked, the polygon will flash, and an input window will be displayed showing the original value of the polygon.
- Enter the new value for the polygon and then select the "OK" button.

5.1.6.3 Deleting a Polygon

A single polygon shape and its associated value can be deleted from the map display. To delete a polygon:

- Click on the "Delete polygon" button.

- Double-click on the polygon to be deleted.
- Once clicked, the polygon will flash, and then a message window will be displayed asking for confirmation of the deletion.
- Click on the “OK” button to delete the polygon.

5.1.6.4 Changing the Color of the Polygon Boundaries

The boundaries that define a polygon area can have different colors to distinguish it from other polygons within the map display. Normally, colors are used for display purposes and visually represent values used to define each polygon layer. To change polygon boundary colors:

- Click on the “Change the color of the polygon boundaries” button. Once the button has been clicked, the “Grid Code Color Select” window (See Figure 5.5) will be displayed.
- Within the “Grid Code Color Select” window, double-click on the code value that the color will be changed. Once the code has been clicked, a color selection window will be displayed.
- Select the desired color by clicking on the colored box and then click on the “OK” button.
- Once the colors have been selected for all polygons, click the “OK” button of the “Grid Code Color Select” window. The polygon boundary colors will be automatically updated in the map display.

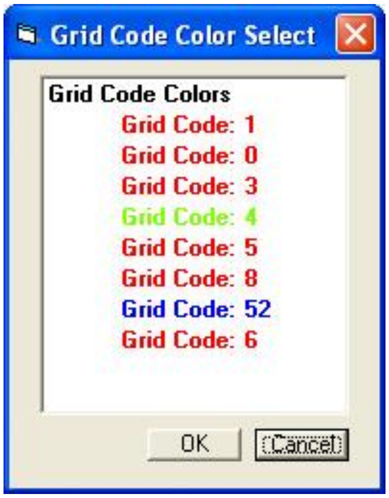


Figure 5.5. Grid Code Color Selection Window

5.1.6.5 Viewing the Value of a Polygon

The grid code value that defines a polygon can be viewed within the map display window. To view the polygon value:

- Click on the “Show polygon grid code” button and then click on the polygon of interest.
- After clicking on a polygon, the polygon will flash, and the “Polygon Grid Code Value” window (see Figure 5.6) will be displayed with the value of the polygon.
- To close the “Polygon Grid Code Value” window, click on the “OK” button.

Figure 5.6. Polygon Grid Code Value Window

5.1.7 Setting the Resolution of the Output .csv File

The polygon areas and their associated values output a .csv file that can be read by DUSTRAN and used in certain calculations. Currently, DUSTRAN can read gridded Zobler soil texture IDs and Olson Ecosystem vegetation codes, both of which are used in the wind-blown dust emissions module. To set the resolution, or grid spacing, derived from the polygon shapes:

- Click on the “Modify the resolution of the output data”  button; the “Specify Sampling Resolution” window will be displayed (Figure 5.7).
- Enter the desired “Easting” and “Northing” resolutions into the text boxes provided.
- Click on the “OK” button to accept the changes and close the window.

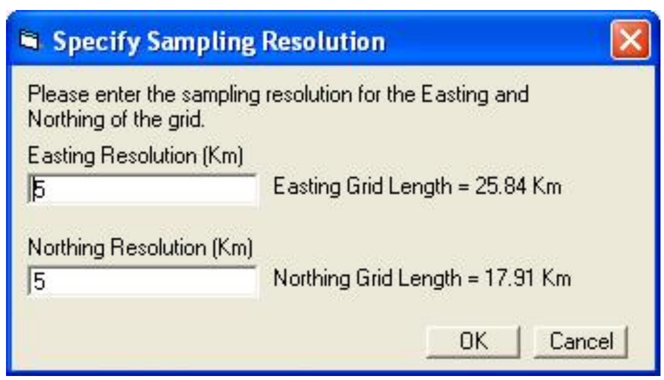


Figure 5.7. Specify Sampling Resolution Window

The output grid resolution should be commensurate with the average size of the polygons making up the data set. For example, if the various polygons making up the data set have an average dimension of roughly 10 km on a side, then the grid resolution should be on the order of 5 km to have roughly four grid points per polygon.

5.1.8 Creating the Output Shape File and .csv File

The polygons can be saved to a shape and text file for use in DUSTRAN. The shape file allows for the polygons to be displayed within the DUSTRAN map window; the text file can be interrogated by the DUSTRAN software for use in certain calculations. Currently, DUSTRAN reads gridded Zobler soil texture IDs and Olson Ecosystem vegetation codes, both of which are used in the wind-blown dust-emissions module. To create the files, click on the “Create shape file”  button; a file browse window will be displayed. Enter a name for the output files and then click on the “Open” button to save the files.

6.0 Adding a New User Profile to DUSTRAN

DUSTRAN V2.0 allows for the creation and use of multiple user profiles. These user profiles allow multiple users to use the same DUSTRAN software to run individual simulations using their own separate and distinct simulation sites. User profiles created using DUSTRAN V2.0 are no longer limited to being located within the DUSTRAN directory and can be created at any location the user desires as long as the machine on which DUSTRAN is installed has read/write access to that location. This allows users using the same machine to run their own instance of DUSTRAN without overwriting another user's site data. This section provides the step-by-step instructions for adding a new user profile to DUSTRAN.

1. To add a new user profile to DUSTRAN, both MapWindow and DUSTRAN must already be installed. Open MapWindow and click on the "D" button on the MapWindow toolbar. The DUSTRAN user interface will open and be displayed along with the MapWindow interface.
2. Within the DUSTRAN interface click on the "Add Site" button (see Figure 6.1).

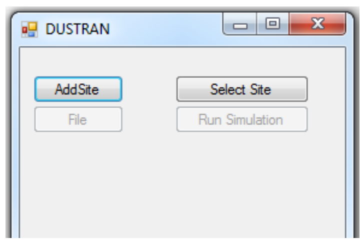


Figure 6.1. DUSTRAN's "Add Site" Button

3. The "Select Simulation Profile To Use" window will be displayed.

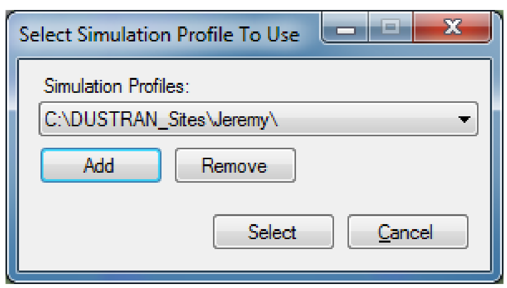


Figure 6.2. "Select Simulation Profile To Use" Window

4. Click on the "Add" button to add a new profile.
5. Within the "Browse For Folder" window, either select an existing folder to locate the new profile or create a new folder.
6. In the "Simulation Profile Name" window, enter a name for the new user profile.
7. The profile name and path should now appear in the profile selection window. Click on the "Select" button to select the new profile. This profile will be used to run an instance of DUSTRAN without overwriting another user's site data.

7.0 Adding a New Site to DUSTRAN

A new site can be added to DUSTRAN using the "Add Site" wizard within the DUSTRAN interface. A new site contains all the geophysical data, such as terrain and land use/land cover, that is needed to run the AERMOD, CALPUFF, and CALGRID dispersion models. In addition, a new site contains base geospatial data that are used for displaying and analyzing the site within MapWindow. After a new site has been created, the user can add additional geospatial data to customize the map for a particular scenario or application.

This section provides step-by-step instructions for adding a new site to DUSTRAN. Specifically, the "Add Site" wizard will guide the user through a series of forms that automate the following tasks:

- Specify the spatial range, or extent, of the new site.
- Designate a name and UTM zone for the site.
- Create a new site directory that contains all the necessary template files for performing a simulation in DUSTRAN.
- Build DUSTRAN's meteorological surface (.snf) and upper-air (.unf) station information files.
- Build and save new GIS map files for display in MapWindow.
- Register the site so that it can be opened within the DUSTRAN interface.

1. To add a new site, both MapWindow and DUSTRAN must already be installed. Open MapWindow and click on the "DUSTRAN" button on the MapWindow toolbar. DUSTRAN's interface will open alongside the MapWindow application interface.
2. Within DUSTRAN, click on the "Add Site" button (see Figure 7.1):

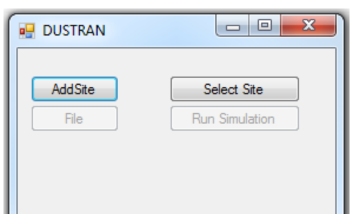


Figure 7.1. The DUSTRAN "Add Site" Button

3. After clicking "Add Site," a set of instructions will appear that provides information on how to select the site location within MapWindow (see Figure 7.2). After reading the instructions, click "OK" to continue.
4. A map of the continental United States will be automatically loaded into the display of the MapWindow interface.
5. Use the MapWindow navigation tools (e.g., zoom and pan) to navigate to a desired location on the base map where the new site is to be located. Right-click on the general location of where the site is to be located. This sets a central point from which all geospatial data will be selected and clipped.
6. After right-clicking on the map, a dialog box will appear prompting for the geospatial site dimensions (see Figure 7.3). This affects the overall extent of the geospatial data and sets the clipping envelope for the data. The default values for the height and width are 600 km to verify that the spatial envelope is large enough to provide flexibility in siting the model domain in DUSTRAN. After setting the height and width for the geospatial envelope, click "OK" to continue.

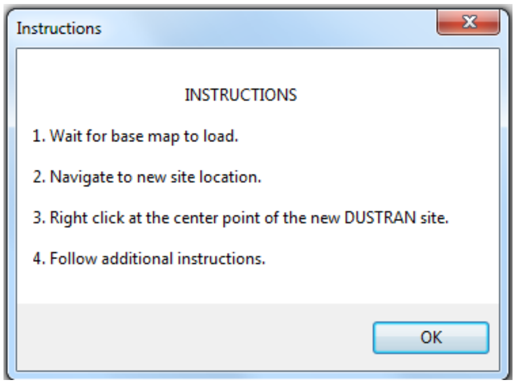


Figure 7.2. DUSTRAN's Instructions for Selection of Site Location

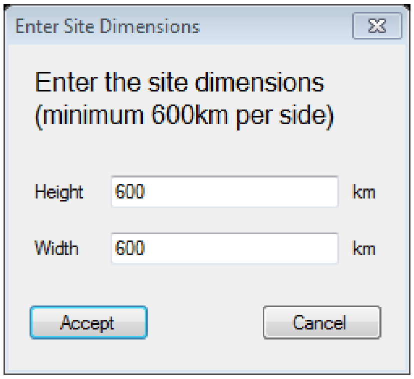


Figure 7.3. Prompt for Geospatial Site Dimensions

7. The base map will be displayed with a red box that represents the clipping envelope that will be applied to the geospatial data (see Figure 7.4). The box is used to intersect the "US_States" layer. The envelope of the intersecting states serves as the actual clipping boundary for the geospatial data. Click "Yes" to accept or "No" to start again.
8. After accepting the clipping envelope, a dialog box will appear prompting for the site name and UTM zone (see Figure 7.5). The site name must be unique and cannot be the name of an existing site. The UTM zones are labeled on the base map (see image above for an example) and are used as the coordinate system in DUSTRAN. If the center of the site is in a UTM zone that is different than the UTM zone selected by the user, the selection must be confirmed before continuing.
9. Click "Finish" to finalize the site creation process. A progress bar and task list display the progress of the site creation. Because the utility must clip all of the base layers, loading the geodatabase may take several minutes, especially for sites located on the eastern coast of the United States.
10. A window will appear asking for the sampling resolution for the vegetation and soil texture files that are used in wind-blown dust simulations (see Figure 7.6). A default resolution of 4 km has been set.
11. Click the "OK" button, and the wizard will create four files: srztext.shp, srztext.csv, owe14d.shp, and owe14d.csv. The srztext files define the soil texture, and the owe14d files define the vegetation cover. The files with the extension .shp are shape files that are viewable within MapWindow and are stored within the site's geodatabase; the .csv files are comma-delimited ASCII files that are interrogated by DUSTRAN when performing a wind-blown dust simulation and are stored within the TerData directory of the new site.

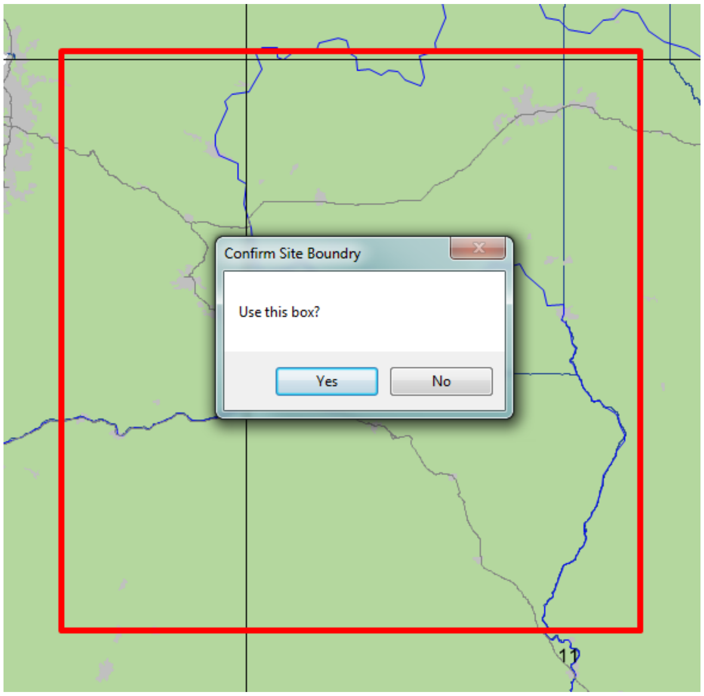


Figure 7.4. Red Box Enclosing Clipping Envelope

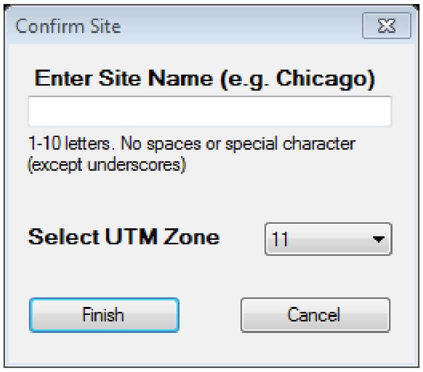


Figure 7.5. Prompt for Site Name and UTM Zone

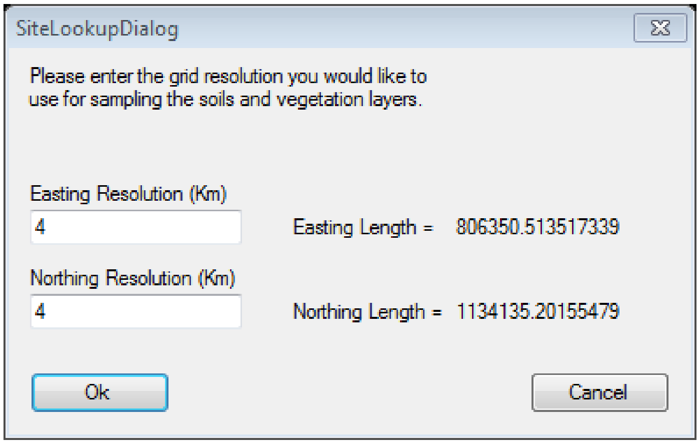


Figure 7.6. Prompt for Sampling Resolution

12. Following the site's registration, the site may be opened by selecting the "Select Site" button on the main DUSTRAN interface.

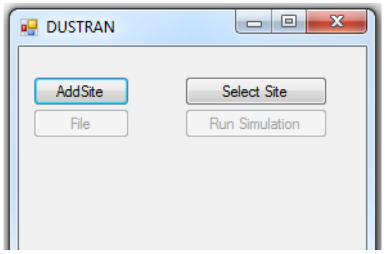


Figure 7.7. Notification for Accessing the New Site

8.0 DUSTRAN Example Tutorials

This section provides example tutorials for creating and running DUSTRAN simulations. Three tutorials are provided, all based on the sample Yakima site. The first tutorial simulates dust emissions from a point and area source using the CALPUFF model. The second tutorial simulates dust emissions from the same point and area source using the AERMOD model. The point-source emissions are prescribed explicitly; the area-source emissions are from vehicular activity and use the vehicular dust-emissions module intrinsic to DUSTRAN to generate the particulate emission rate. The third tutorial simulates a wind-blown dust event using the wind-blown dust-emissions module within DUSTRAN.

To run the examples detailed below, MapWindow and DUSTRAN must already be installed and at least one user profile created. Refer to the section entitled “DUSTRAN Installation Instructions, Section 3.0” in this user’s guide for detailed instructions on installing DUSTRAN and Section 6.0 for instructions on how to add a user profile.

8.1 Creating the Yakima Tutorial Site

Each of the following DUSTRAN tutorials requires a simulation site named “Yakima”; Yakima is located in central Washington State. The following instructions provide the necessary steps to create a Yakima site for use in the DUSTRAN tutorials.

1. Start the MapWindow application and then click on the DUSTRAN “D” button located on its toolbar to start the DUSTRAN plugin.
2. Select the “Add Site” button on the DUSTRAN main interface.
3. Select one of the existing profiles using the “Select Simulation Profile To Use” form and click “Select”.
4. Once the add site instructions are displayed click the “Ok” button and wait for the map layers to be loaded into the MapWindow map display (Figure 8.1).

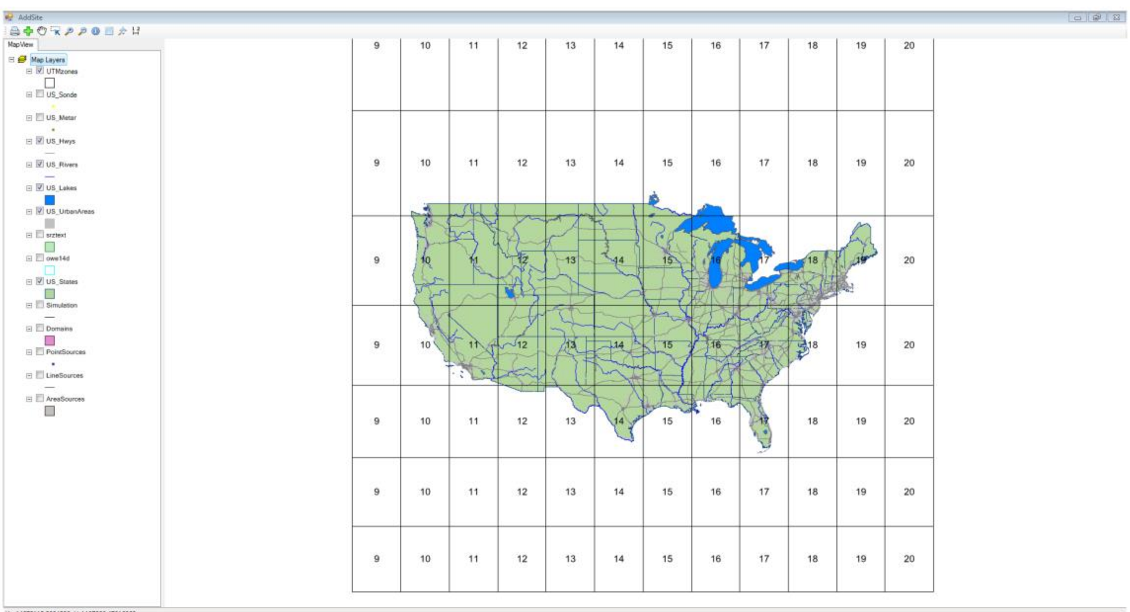


Figure 8.1. Add Site Map

5. Use the “Add Data” button () on the “AddSite” toolbar to navigate and select the “Boundp.shp” shapefile located in the “Tutorial-Layers\WGS84-World-Mercator” folder in the DUSTRAN directory.
6. If a prompt is displayed asking to re-project the files to match the map projection select the “No” option.
7. Once the Boundp.shp shapefile is loaded, the map will automatically zoom to the Hanford boundary
8. Use the “Add Data” button () on the “AddSite” toolbar to navigate and select the DPoints.shp shapefile located in the “Tutorial-Layers\WGS84-World-Mercator” folder in the DUSTRAN directory.
9. If a prompt is displayed asking to re-project the files to match the map projection select the “No” option.
10. At this time the Hanford boundary should be displayed on the map as well as a domain center point (Figure 8.2).

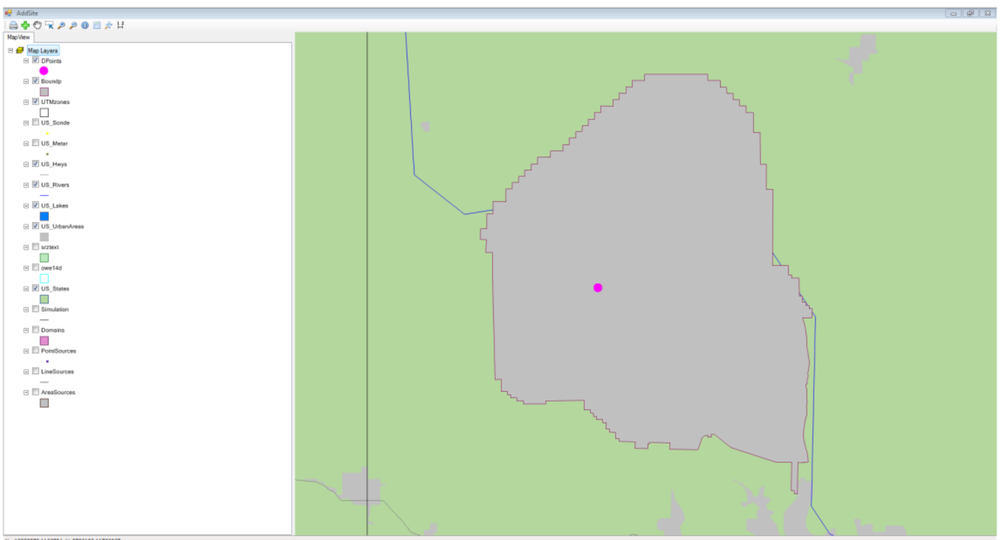


Figure 8.2. Add Site Map with Boundary and Point Data Loaded

11. *Right-click* on the domain point displayed on the map; the application will then walk through the “Add-Site” process detailed in Section 7.0.
12. When prompted to enter a site name, enter “Yakima” and leave the UTM Zone value at the default value of “11.”

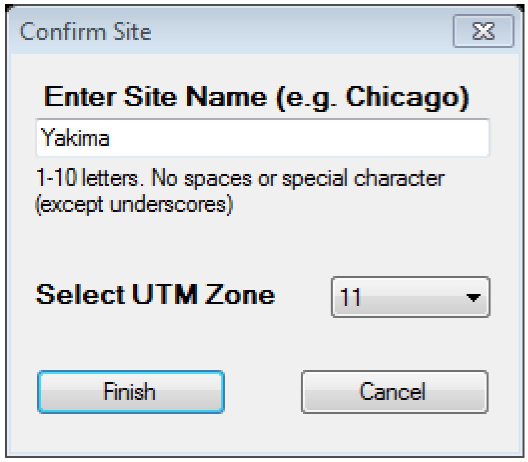


Figure 8.3. Site Name and Zone Input Form

13. For each additional prompt accept the default value provided.
14. Once the site has been created, MapWindow should return to a blank map; a new "Site" folder named Yakima should now be listed inside the selected profile directory.
15. Replace the "owe14d.csv" and "srztext.csv" files located in the "Yakima\TerData" folder with the files located in the "Tutorial-Layers\TerData" folder; these files will be used in the windblown dust tutorial.
16. Click the "Select Site" button on the DUSTRAN interface and then select "Yakima" from the "Current Site" list; click the "Open" button to load the new "Yakima" site within DUSTRAN.
17. Once the "Yakima" site has loaded, use the "Add Map Layer" button ( ) within MapWindow to select and add the "Points.shp" shapefile in the "\Tutorial-Layers\WGS84-UTM-11" folder (Figure 8.4). Note: "If a point is not displayed in the center of the map window, the "Points" layer may need to be moved to the top of the "Data Layers" list within MapWindow.

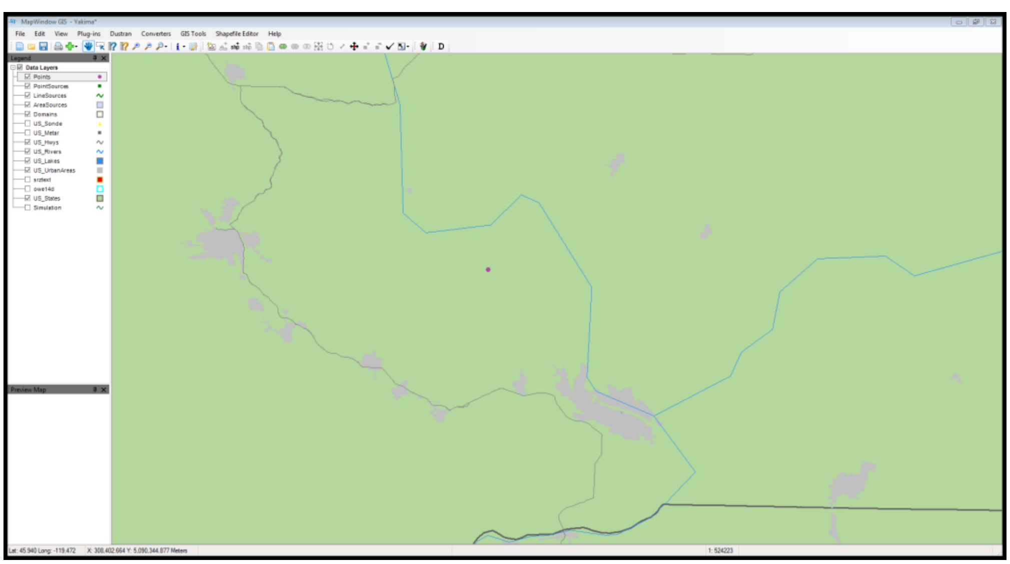


Figure 8.4. Points Layer Displayed

18. Once the “Points.shp” shapefile has been loaded, click on the “Add New Domain Center” button () from within the main DUSTRAN interface.
19. When prompted, click on the point displayed by the “Points.shp” shapefile.
20. When prompted, enter “Yakima” for the domain name, and click “Ok” to add a new model domain.
21. With the new model domain added, the “Yakima” site is now ready to be used with the DUSTRAN tutorials (Figure 8.5).

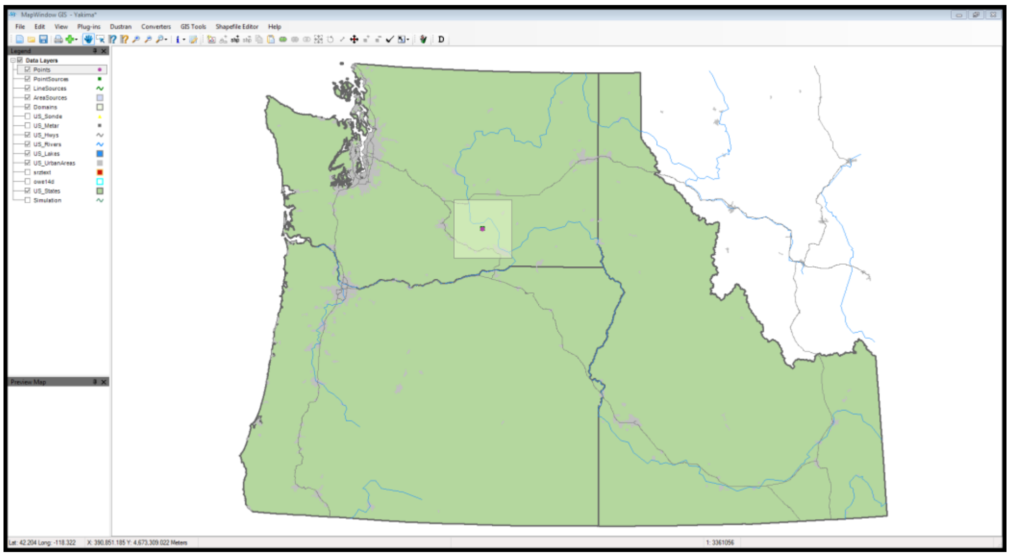


Figure 8.5. Completed Yakima Site

8.2 Simulating Dust Dispersion from Active Source Emissions Using CALPUFF

This tutorial steps through a dust-dispersion scenario from active source emissions for the “Yakima” site using the CALPUFF dispersion model. To perform this tutorial, the Yakima site must already exist before completing this tutorial; see Section 8.1 for instructions on creating the Yakima site.

In this example, a point source will be used to represent particle emissions from a stack; the emissions will be defined explicitly. An area source will also be created and it will represent a region of dust emissions from vehicular activity; these emissions will be calculated automatically by the DUSTRAN vehicular dust-emissions module. Both sources will be set to run for the same duration, and the downwind air-concentration and ground deposition will be simulated.

8.2.1 Starting DUSTRAN

DUSTRAN is an integrated dispersion modeling application integrated with the MapWindow GIS application. To begin a DUSTRAN simulation, open MapWindow. On the MapWindow toolbar, click on the “D” button (Figure 8.6):

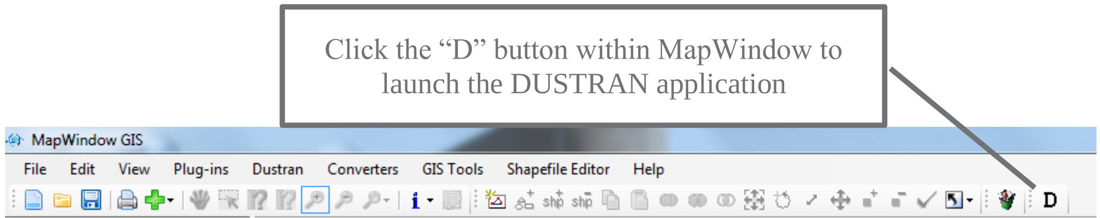


Figure 8.6. MapWindow Toolbar

8.2.2 Selecting a Site

The user interface to the DUSTRAN model will appear alongside the MapWindow GIS application (Figure 8.7). Click the "Select Site…" button to open a dialog box that allows the user to select an existing site:

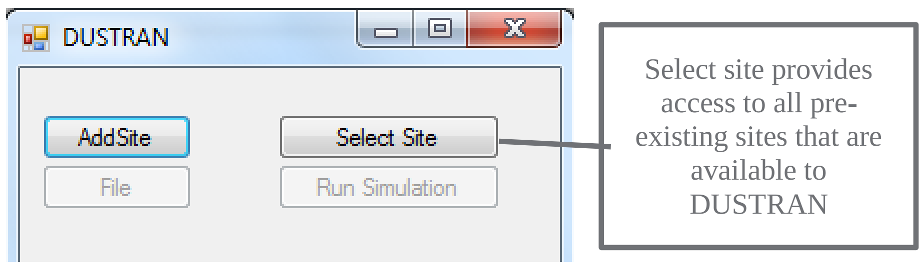


Figure 8.7. DUSTRAN User Interface

Select "Yakima" from the list of available model sites (Figure 8.8) and then click "Open" (Note: Section 8.1 provides instructions on how to create the "Yakima" site.):

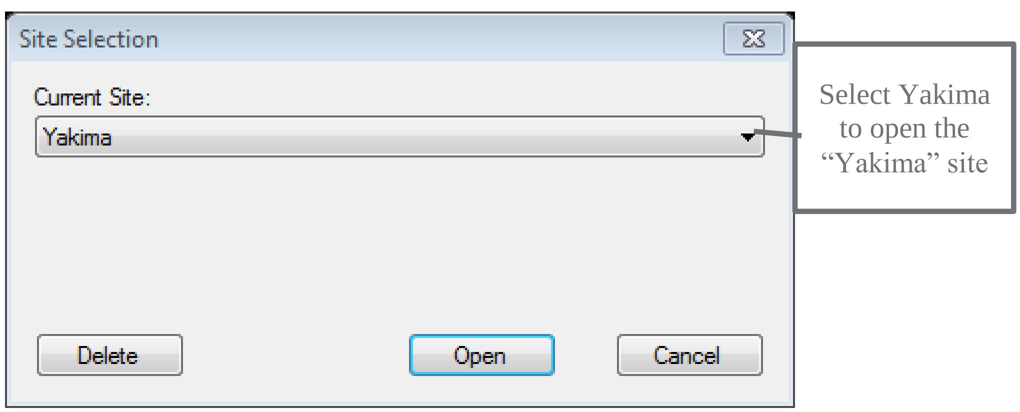


Figure 8.8. Available DUSTRAN Sites

The "Yakima" site will open within MapWindow. A list of available GIS data layers will appear in the left frame, and DUSTRAN-specific input parameters will appear in the DUSTRAN frame (Figure 8.9).

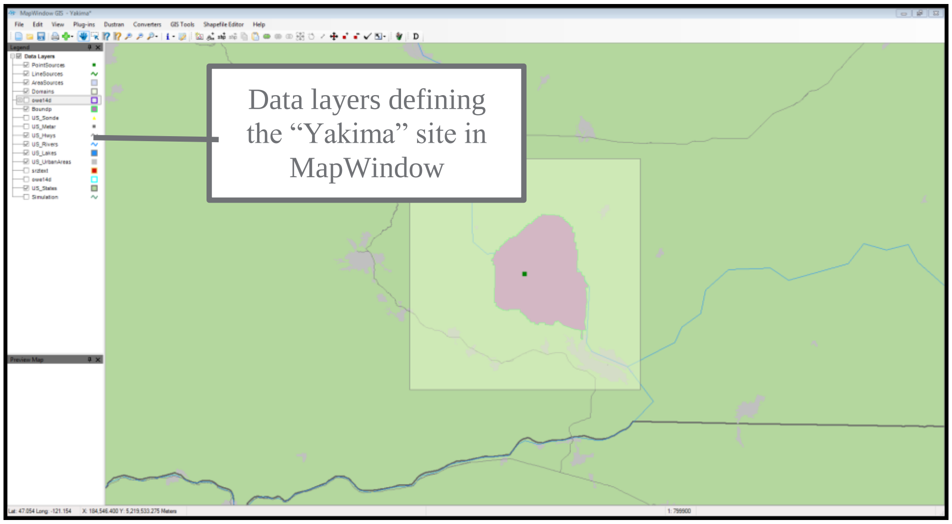


Figure 8.9. Yakima Site Displayed in DUSTRAN

8.2.3 Creating a Domain

The first step in setting up a scenario in DUSTRAN is to select a domain and set the domain size. A domain is a user-specified area where both meteorological and dispersion model calculations are performed.

Select the "Yakima" domain from the domain list in the "Domain" panel of the DUSTRAN interface.

By default, the domain size is set to 100 km. Since CALPUFF is a puff model, it can be used to model long-range dispersion out to several hundred kilometers. However, in this example, a domain size of 80 km will be used. Set the size from the "Size" listbox in the "Domain" panel to 80 km. The domain should appear as a shaded, rectangular region within MapWindows.

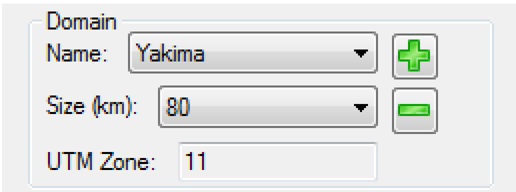


Figure 8.10. Domain Panel

8.2.4 Setting the Release Period

This example will simulate and early-morning release. Set the "Release Period" "Start Time" to 7 a.m. and the "Release Duration" to 3 hours.

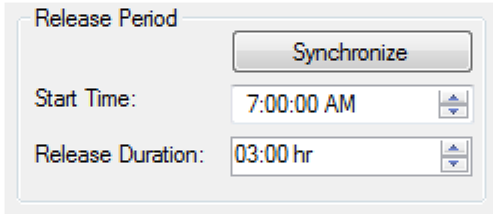


Figure 8.11. Release Period Panel

8.2.5 Setting the Simulation Scenario

This example will utilize the “CALPUFF” dispersion model to perform the atmospheric dispersion. Therefore, select the simulation “Type” as “Source Emission – CALPUFF”. Since the “Yakima” domain is being used, set the “Time Zone” to PST. Set the “Start Date Time” to the current date and the start time to 4 a.m. (Note: the simulation start time for a CALPUFF or CALGRID simulation must always begin before sunrise, as CALMET, the meteorological model used for these simulations, needs to calculate the hourly surface heat flux from sunrise until the end of scenario or sunset, whichever occurs first.) Finally, set the model “Run Duration” to 8 hours.

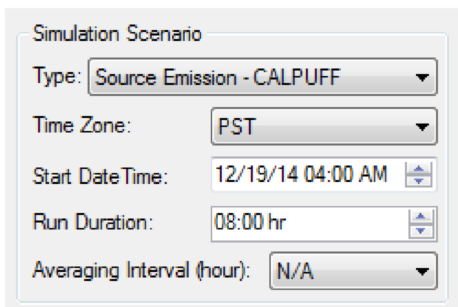


Figure 8.12. Simulation Scenario Panel

8.2.6 Setting the Model Species

By default, there are four particulate matter (PM) species available to model in DUSTRAN (Figure 8.13). This tutorial will model the 10 micron (PM_{10}) species; therefore, uncheck $PM_{2.5}$, PM_{15}, and PM_{30}.

Additional species (particles and gases) can be added using the “Add Species”  button on the “Species” tab, but for this exercise, we will use the existing PM_{10} species in the list.

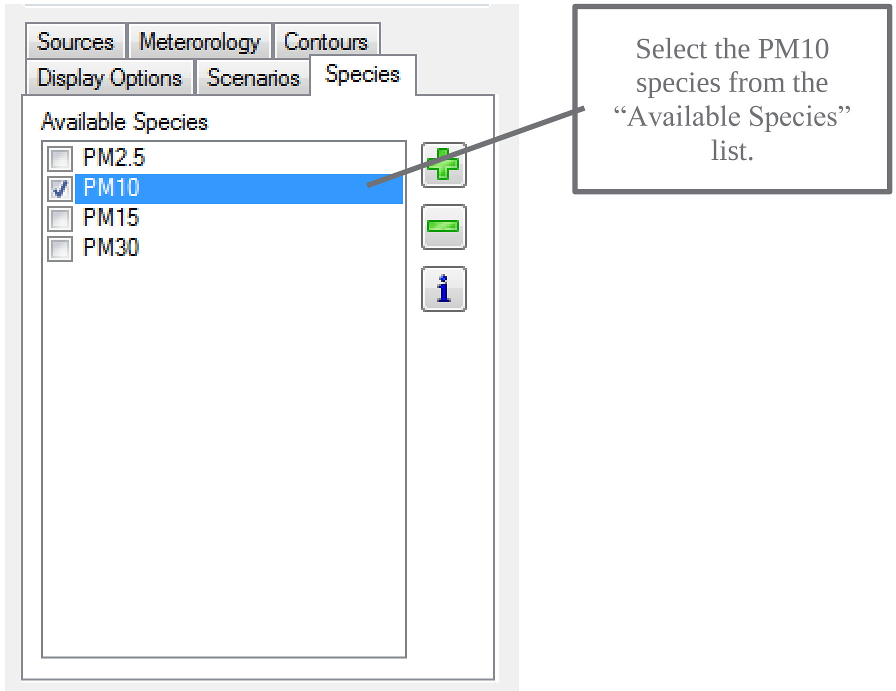


Figure 8.13. Species Tab – "Available Species" List

8.2.7 Creating a Point Source

Click on the "Sources" tab. Note that there is already a point source called "Yakima" listed under "Point Sources"; this corresponds to the point that is used by DUSTRAN to mark the center of the model domain. Uncheck the name, and the point will disappear from the center of the domain in the MapWindow map display.

To create a new point source, click on the "Point Source" button on the "Sources" tab. Click on a location within the domain to place the point source. Call the source "Stack" and click "OK."

The "Source Input" form for the point source will appear (Figure 8.14). Enter the "Release Parameters" as in Figure 8.14 and click "OK" to continue:

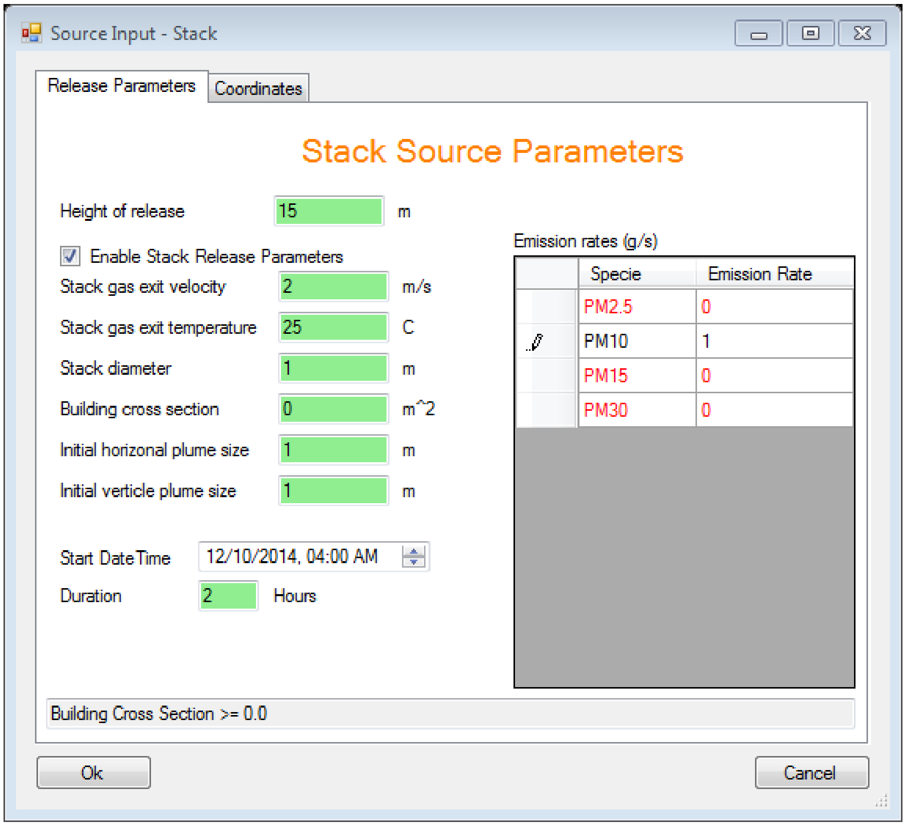


Figure 8.14. Point Source Input Form – "Release Parameters" Tab

Notice that the new source shows up as a point in the MapWindow display and also appears in the "Point Source" list on the "Sources" tab.

8.2.8 Creating an Area Source

Next, create an area source by clicking on the "Area Source"  button on the "Sources" tab. An area source can be a triangle or four-sided polygon; it is created by clicking on three or four locations in the MapWindow map display. Note: the final corner should be a "right-clicked" to complete the polygon. Create the area source by drawing ≈1 km square polygon near the center of the domain, enter the source name "Field," and click "OK."

The "Source Input" form for the area source will appear (Figure 8.15):

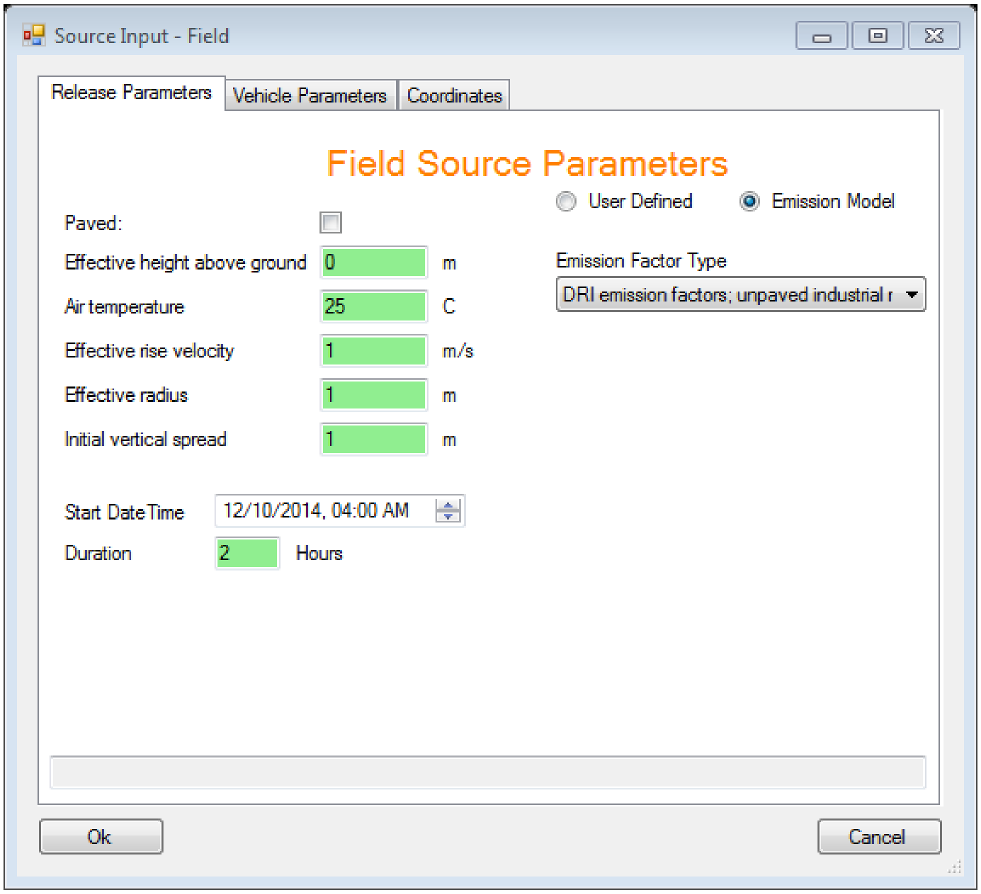


Figure 8.15. Area Source Input Form – "Release Parameters" Tab

For the area source, we will use the "Emission Model" to calculate dust emissions created by a single vehicle using the "DRI Factors" (Desert Research Institute). Vehicular dust emissions are a function of vehicle parameters, such as weight and speed. Click on the "Vehicle Parameters" tab (Figure 8.16) and specify the following vehicular information:

- The "Distance Traveled" is the total distance traveled by the vehicle within the area; enter a distance traveled of "5 km" (Figure 8.16). The emissions are assumed to be uniformly distributed over the area and constant for the duration of the release.
- Click "OK," and the area source will appear in the MapWindow map display and also under the "Area Sources" list on the "Sources" tab (Figure 8.17).

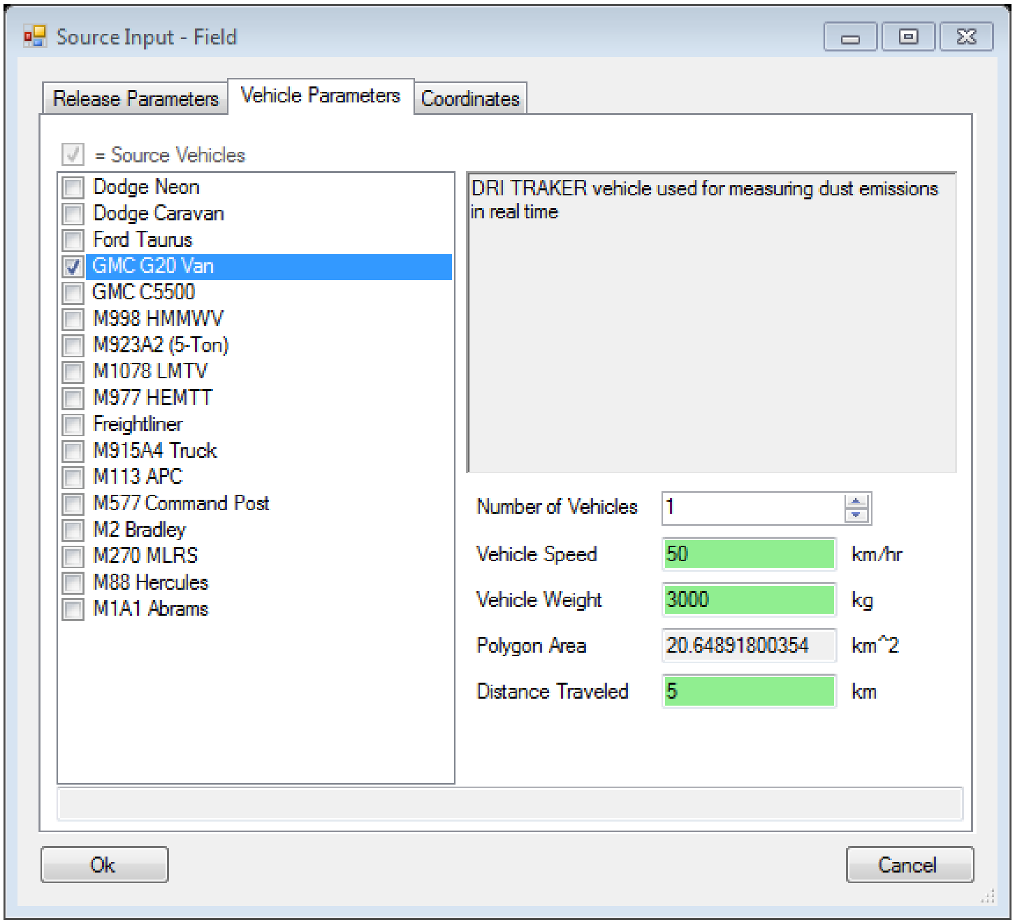


Figure 8.16. Area Source Input Form – "Vehicle Parameters" Tab

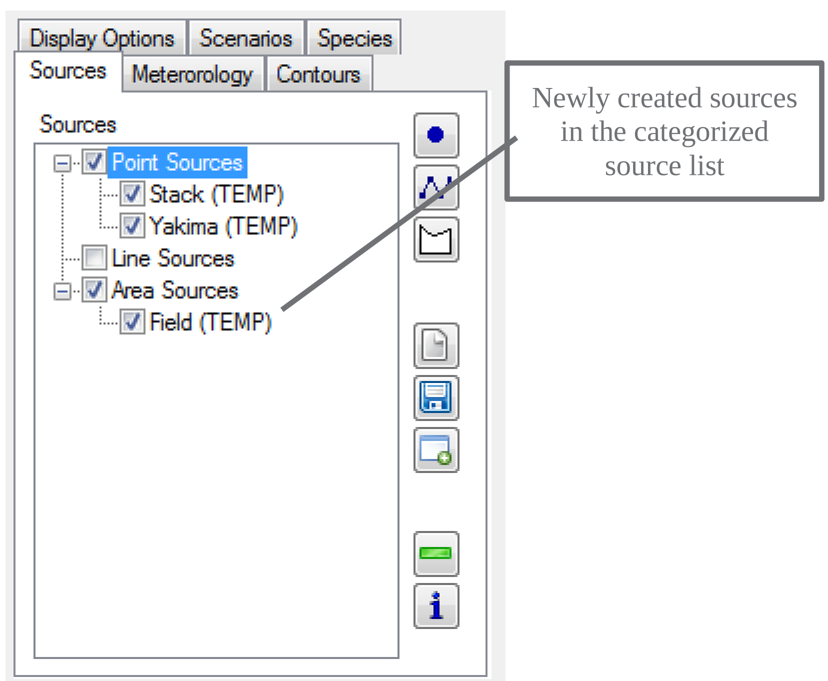


Figure 8.17. Sources Tab – "Sources" List

8.2.9 Entering Meteorological Data

Next, click on the “Meteorology” tab within DUSTRAN and select “Single Observation” from the listbox. The “Specify Meteorological Data” form appears. Enter the meteorological observations as shown in the form below (Figure 8.18) shows the completed form.

Simulation Hour	Year	Month	Day	Hour	Wind Direction (deg)	Wind Speed (m/s)	Temperature (C)	Relative Humidity (%)	Station Pressure (mb)	Mixing Height (m)	Stability
1	2014	12	30	5	240	2.2	20	20	1005	200	E - Slightly Stable
2	2014	12	30	6	260	3	19	20	1007	200	E - Slightly Stable
3	2014	12	30	7	255	2.3	20	20	1003	200	E - Slightly Stable
4	2014	12	30	8	180	4	19	15	1000	200	E - Slightly Stable
5	2014	12	30	9	160	4.5	22	45	1005	300	D - Neutral
6	2014	12	30	10	157	4	23	55	1010	320	C - Slightly Unstable
7	2014	12	30	11	155	3.8	22	62	1011	500	C - Slightly Unstable
8	2014	12	30	12	160	2	23	60	1012	550	C - Slightly Unstable

Figure 8.18. Specify Meteorological Data Form – “Hourly Observations” Tab

8.2.10 Running DUSTRAN

After the sources, meteorology, and release duration information have been entered, a DUSTRAN simulation can be made. To run DUSTRAN, click on the “Run Simulation” button (Figure 8.19).

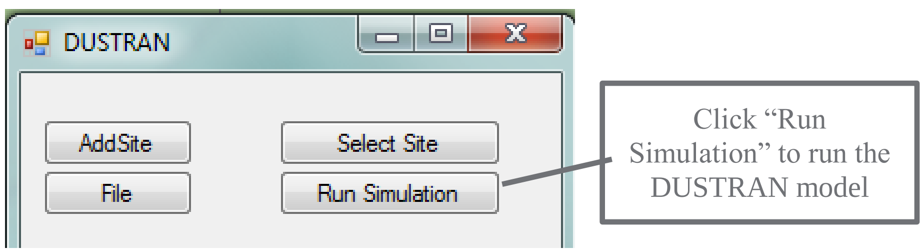


Figure 8.19. “Run Simulation” Button for Running DUSTRAN Simulation

8.2.11 Displaying Model Output

After the models finish running, click on the “Display Options” tab. Check “Contours” and “Wind Vectors” so that the plume contours and wind field will be displayed in the MapWindow map display (Figure 8.20). Wind vectors can be displayed for various layers throughout the domain; 20 meters is the top of the first layer, with mid-cell corresponding to 10 meters above ground level:

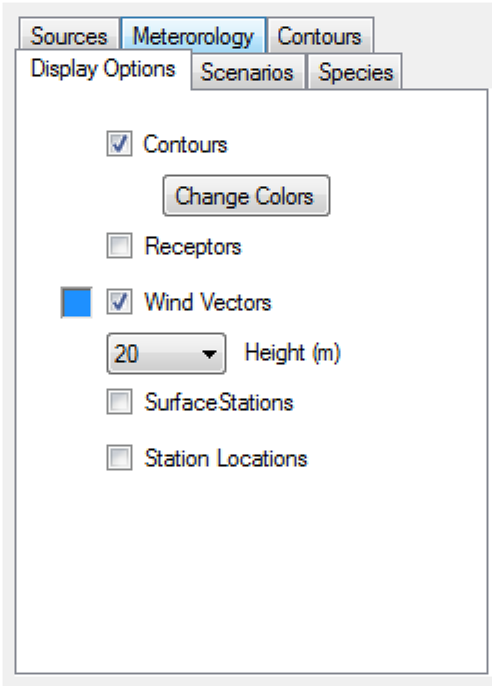


Figure 8.20. Display Options Tab – "Contours" and "Wind Vectors" options

8.2.12 Viewing Model Results

For each model time step, DUSTRAN calculates plume concentration and exposure as well as deposition and total deposition. To view a particular contour, click on the "Contours" tab and choose from the "Contour Types" listbox (Figure 8.21):

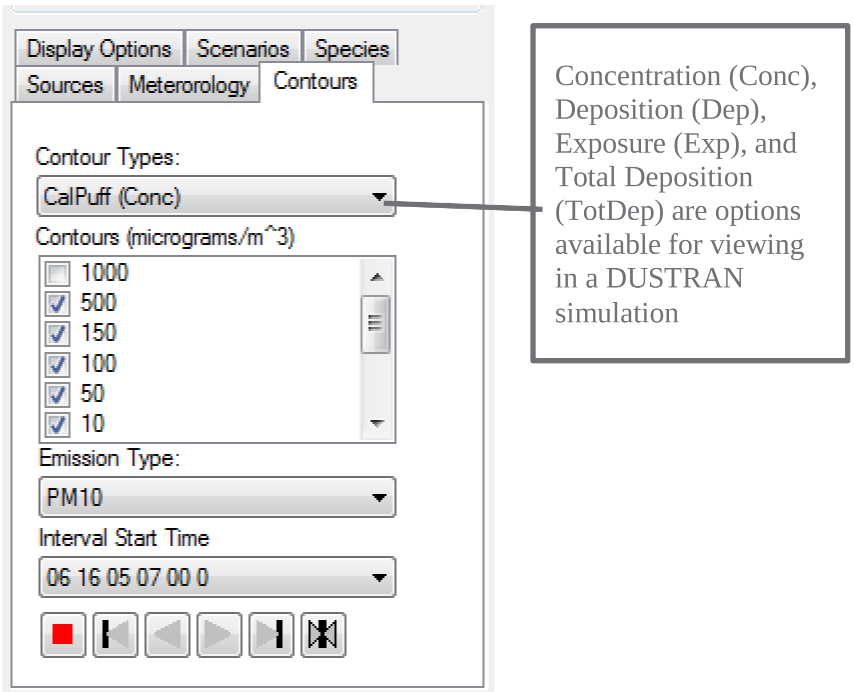


Figure 8.21. Contours Tab - "Contour Types" Listbox

In this example, select "Conc" to display hourly concentrations within MapWindow.

For a given contour type, numerous "Contours" are available for displaying. To display a particular contour interval, check the box next to the contour value. The default selection is normally adequate for displaying the maximum extent of the plume envelope.

To view a particular time step, select an interval from the "Interval Start Time" listbox. In this example, hourly time steps are available from the start of the release (7:00 a.m. local time) till the end of the run duration. Choose the 9:00 a.m. time step, which corresponds to the one hour (9:00 a.m. till 10:00 a.m.) average concentration (Figure 8.22):

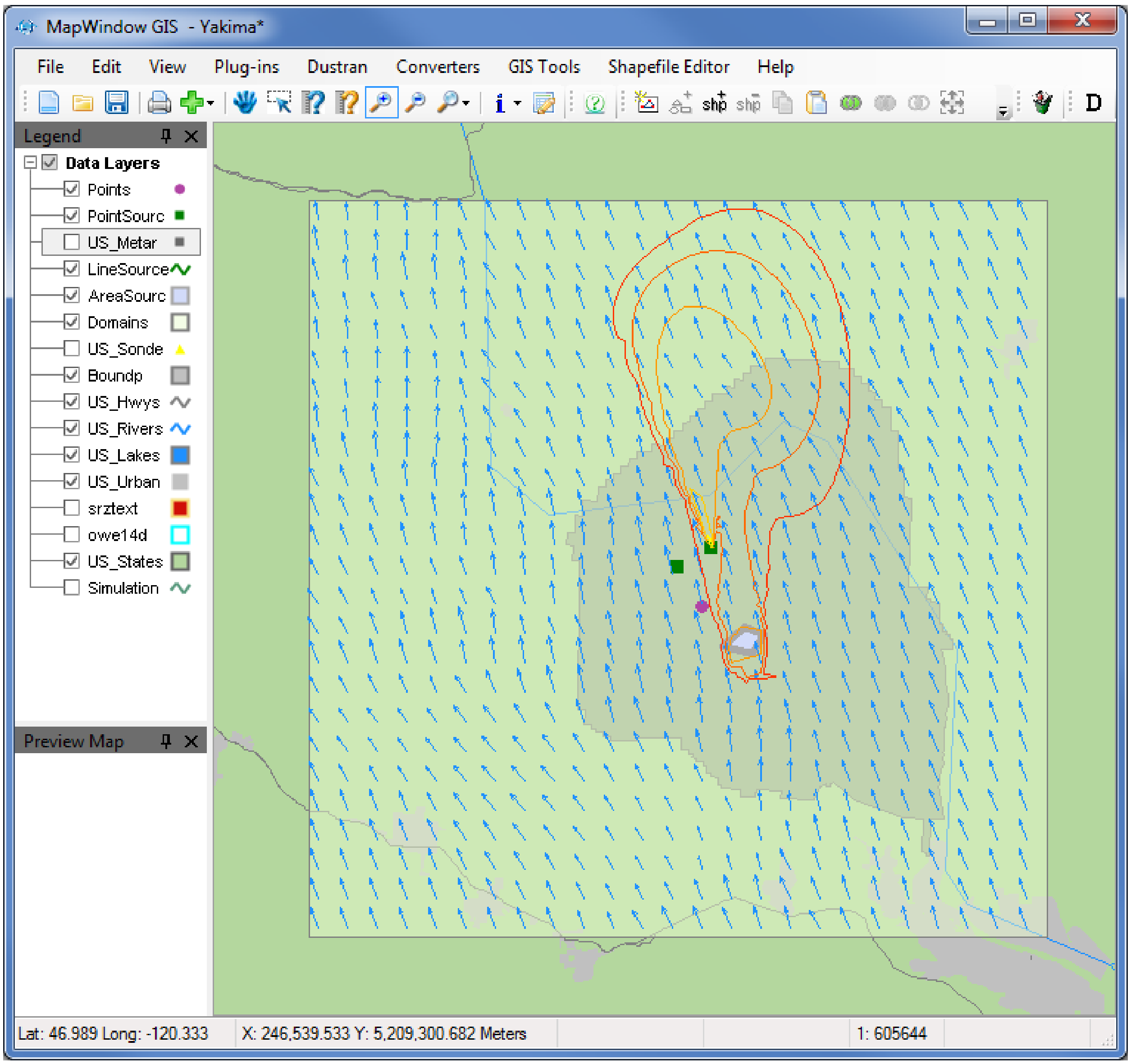


Figure 8.22. Display of Concentration Contours and Wind Vectors for the Hour from 9-10 AM

Notice in the above image that the two plumes (point and area-source plumes) have merged. In addition, note the distortion to the wind field due to the local topography. The distortion is caused by terrain blocking because of the early-morning stability and slope flow due to cold-air drainage.

8.3 Simulating Dust Dispersion from Active Source Emissions Using AERMOD

This tutorial steps through a dust-dispersion scenario from active source emissions for the "Yakima" site using the AERMOD dispersion model. To perform this tutorial, the Yakima site must already exist before completing this tutorial; see Section 8.1 for instructions on creating the Yakima site.

In this example, a point source will be used to represent particle emissions from a stack; the emissions will be defined explicitly. An area source will also be created and it will represent a region of dust emissions from vehicular activity; these emissions will be calculated automatically by the DUSTRAN vehicular dust-emissions module. Both sources will be set to run for the same duration, and the downwind air-concentration and ground deposition will be simulated.

8.3.1 Starting DUSTRAN

DUSTRAN is an integrated dispersion modeling application integrated with the MapWindow GIS application. To begin a DUSTRAN simulation, open MapWindow. On the MapWindow toolbar, click on the "D" button (Figure 8.23):

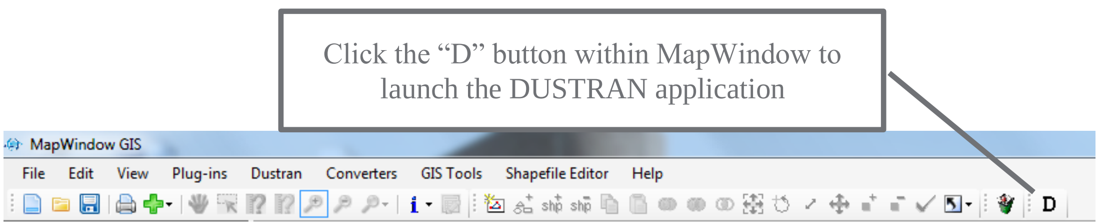


Figure 8.23. MapWindow Toolbar

8.3.2 Selecting a Site

The user interface to the DUSTRAN model will appear alongside the MapWindow GIS application (Figure 8.24). Click the "Select Site…" button to open a dialog box that allows the user to select an existing site:

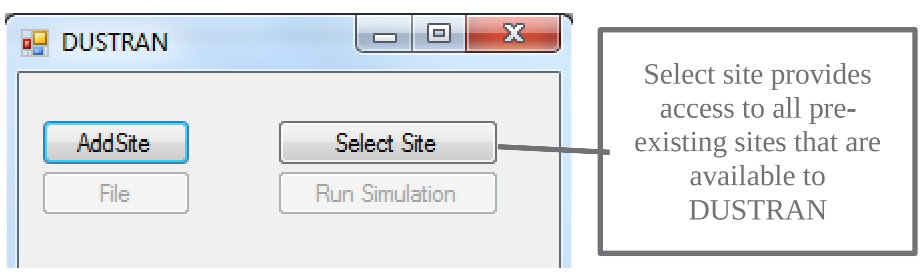


Figure 8.24. DUSTRAN User Interface

Select "Yakima" from the list of available model sites (Figure 8.25) and then click "Open" (Note: Section 8.1 provides instructions on how to create the "Yakima" site.):

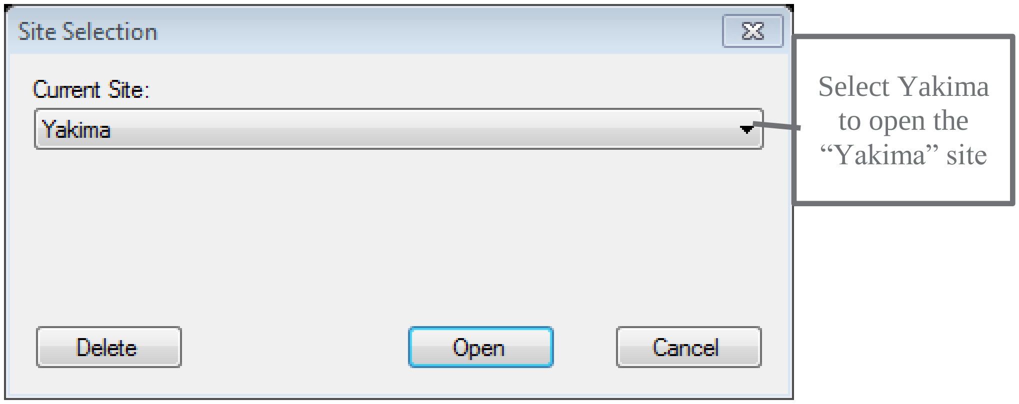


Figure 8.25. Available DUSTRAN Sites

The "Yakima" site will open within MapWindow. A list of available GIS data layers will appear in the left frame, and DUSTRAN-specific input parameters will appear in the DUSTRAN frame (Figure 8.26).

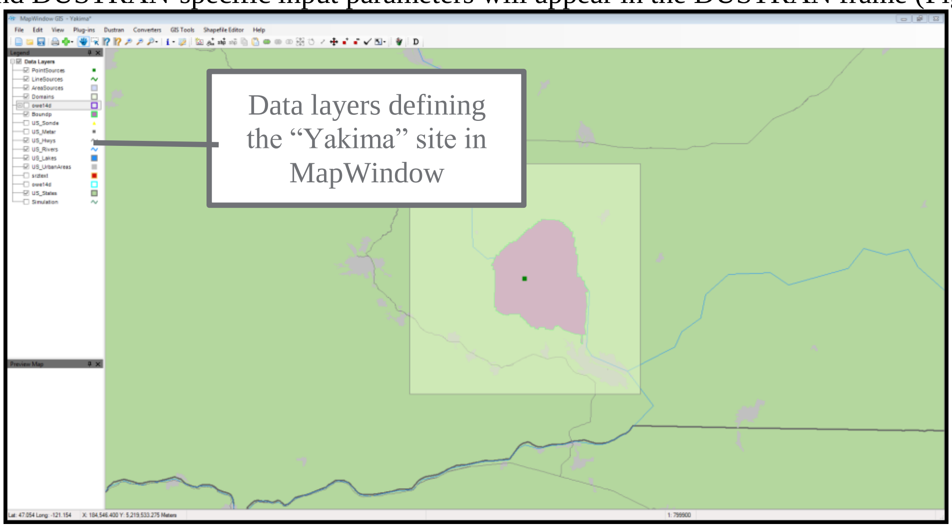


Figure 8.26. Yakima Site Displayed in DUSTRAN

8.3.3 Creating a Domain

The first step in setting up a scenario in DUSTRAN is to select a domain and set the domain size. A domain is a user-specified area where both meteorological and dispersion model calculations are performed.

Select the "Yakima" domain from the domain list in the "Domain" panel of the DUSTRAN interface.

Since AERMOD is a plume model, it should only be used to model short-range dispersion (i.e., out to approximately 50 km). In this example, a domain size of 50 km will be used. Set the size from the "Size" listbox in the "Domain" panel to 50 km (Figure 8.27). The domain should appear as a shaded, rectangular region within MapWindows.

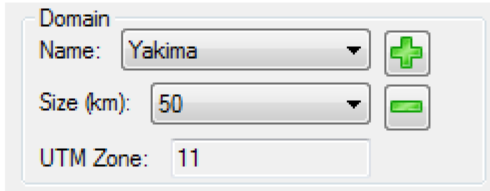


Figure 8.27. Domain Panel

8.3.4 Setting the Release Period

This example will simulate and early-morning release. Set the “Release Period” “Start Time” to 7 a.m. and the “Release Duration” to 3 hours (Figure 8.28).

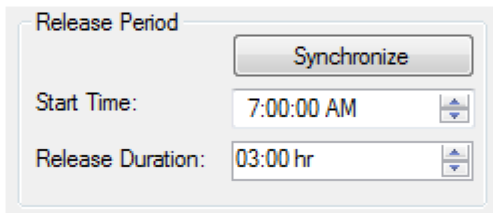


Figure 8.28. Release Period Panel

8.3.5 Setting the Simulation Scenario

This example will utilize the “AERMOD” dispersion model to perform the atmospheric dispersion. Therefore, select the simulation “Type” as “Source Emission – AERMOD”. Since the “Yakima” domain is being used, set the “Time Zone” to PST. Set the “Start Date Time” to the current date and the start time to 7 a.m. Finally, set the model “Run Duration” to 3 hours (Figure 8.29).

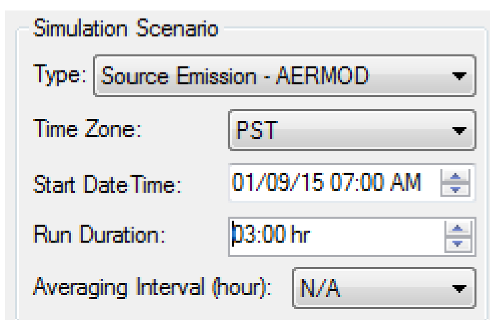


Figure 8.29. Simulation Scenario Panel

8.3.6 Setting the Model Species

By default, there are four particulate matter (PM) species available to model in DUSTRAN (Figure 8.30). This tutorial will model the 10 micron (PM_{10}) species; therefore, uncheck $PM_{2.5}$, PM_{15}, and PM_{30}.

Additional species (particles and gases) can be added using the “Add Species”  button on the “Species” tab, but for this exercise, we will use the existing PM_{10} species in the list.

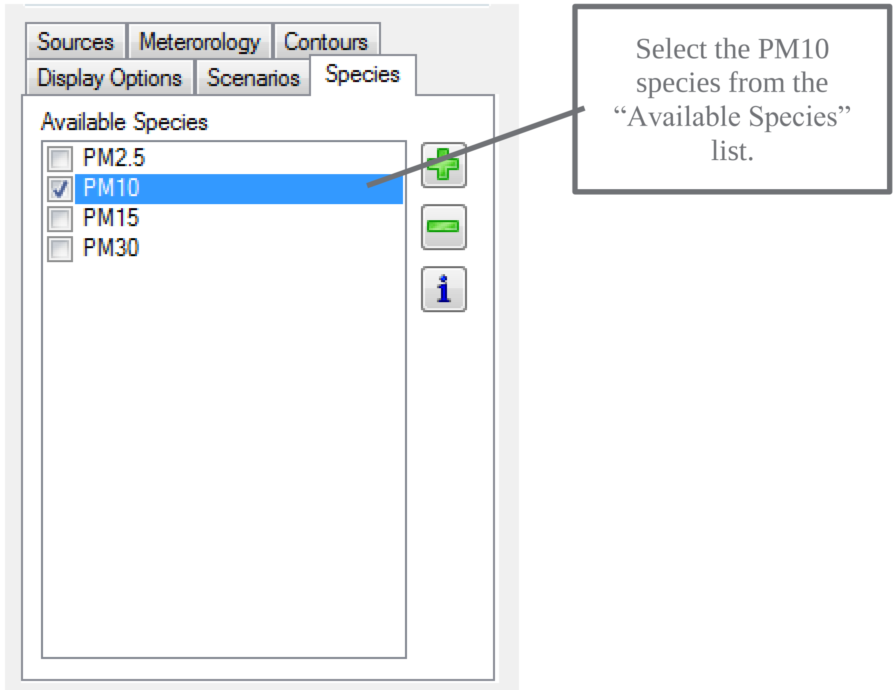


Figure 8.30. Species Tab – "Available Species" List

8.3.7 Creating a Point Source

Click on the "Sources" tab. Note that there is already a point source called "Yakima" listed under "Point Sources"; this corresponds to the point that is used by DUSTRAN to mark the center of the model domain. Uncheck the name, and the point will disappear from the center of the domain in the MapWindow map display.

To create a new point source, click on the "Point Source"  button on the "Sources" tab. Click on a location within the domain to place the point source. Call the source "Stack" and click "OK."

The "Source Input" form for the point source will appear (Figure 8.31). Enter the "Release Parameters" as in Figure 8.31 and click "OK" to continue:

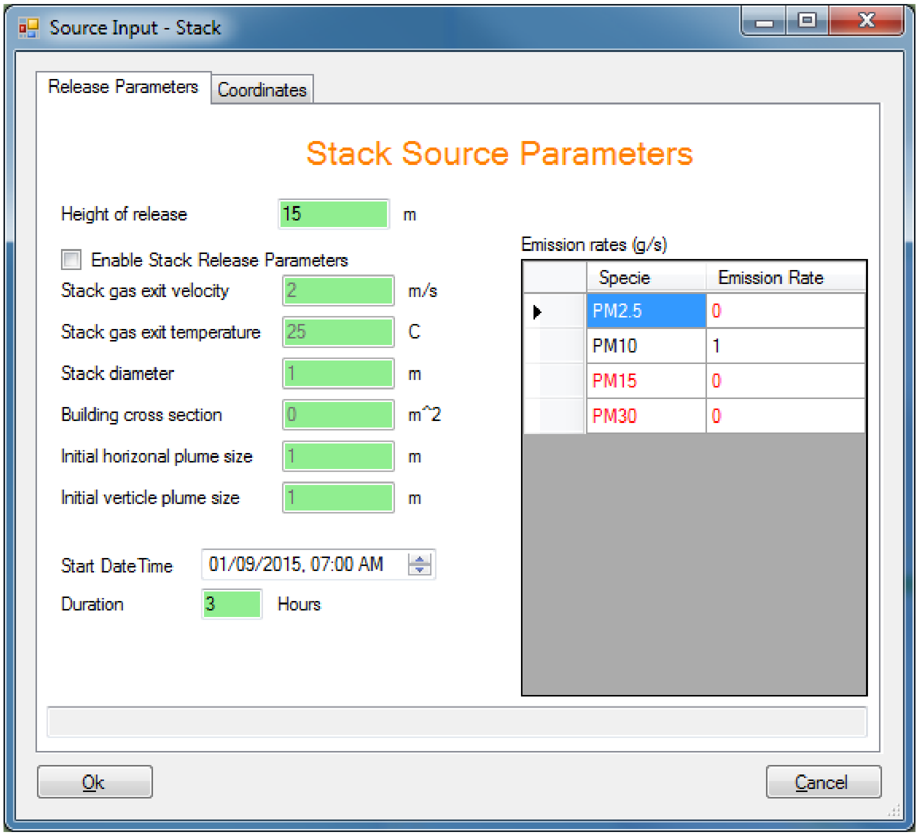


Figure 8.31. Point Source Input Form – “Release Parameters” Tab

Notice that the new source shows up as a point in the MapWindow display and also appears in the “Point Source” list on the “Sources” tab.

8.3.8 Creating an Area Source

Next, create an area source by clicking on the “Area Source”  button on the “Sources” tab. An area source can be a triangle or four-sided polygon; it is created by clicking on three or four locations in the MapWindow map display. Note: the final corner should be a “right-clicked” to complete the polygon. Create the area source by drawing ≈1 km square polygon near the center of the domain, enter the source name “Field,” and click “OK.”

The “Source Input” form for the area source will appear (Figure 8.32):

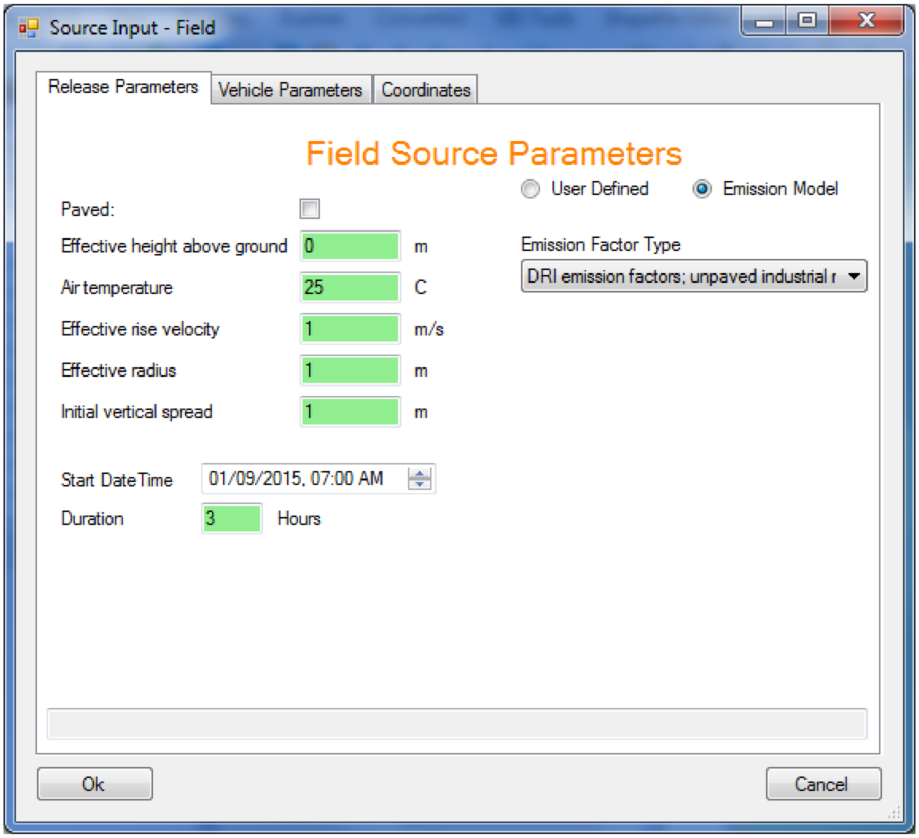


Figure 8.32. Area Source Input Form – "Release Parameters" Tab

For the area source, we will use the "Emission Model" to calculate dust emissions created by a single vehicle using the "DRI Factors" (Desert Research Institute). Vehicular dust emissions are a function of vehicle parameters, such as weight and speed. Click on the "Vehicle Parameters" tab (Figure 8.33) and specify the following vehicular information:

- The "Distance Traveled" is the total distance traveled by the vehicle within the area; enter a distance traveled of "5 km" (Figure 8.33). The emissions are assumed to be uniformly distributed over the area and constant for the duration of the release.
- Click "OK," and the area source will appear in the MapWindow map display and also under the "Area Sources" list on the "Sources" tab (Figure 8.34).

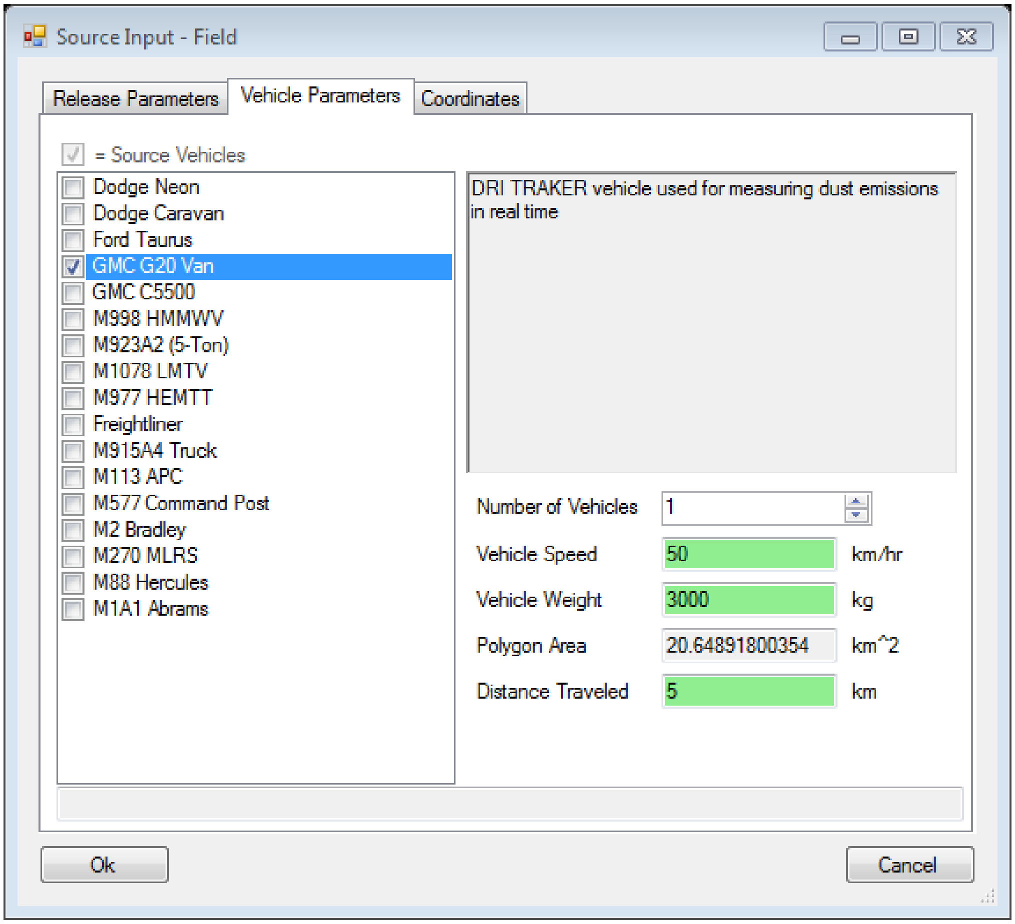


Figure 8.33. Area Source Input Form – "Vehicle Parameters" Tab

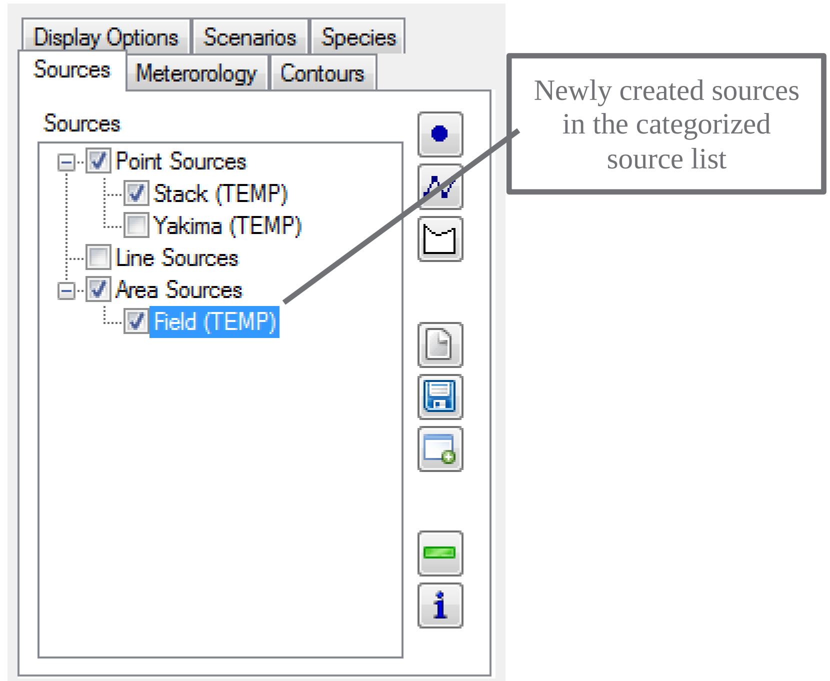
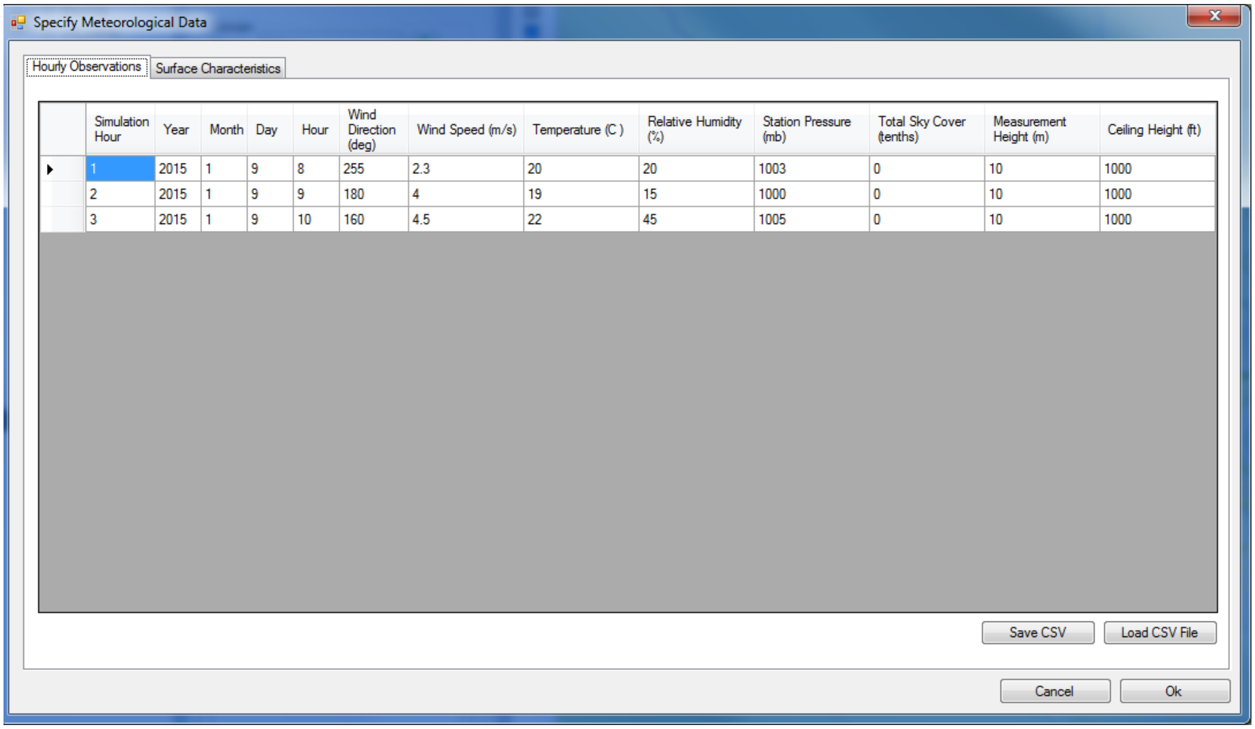


Figure 8.34. Sources Tab – “Sources” List

8.3.9 Entering Meteorological Data and Surface Characteristics

Next, click on the “Meteorology” tab within DUSTRAN and select “Single Observation” from the listbox. The “Specify Meteorological Data” form appears. Enter the meteorological observations as shown in the form below (Figure 8.35):



Simulation Hour	Year	Month	Day	Hour	Wind Direction (deg)	Wind Speed (m/s)	Temperature (C)	Relative Humidity (%)	Station Pressure (mb)	Total Sky Cover (tenths)	Measurement Height (m)	Ceiling Height (ft)
1	2015	1	9	8	255	2.3	20	20	1003	0	10	1000
2	2015	1	9	9	180	4	19	15	1000	0	10	1000
3	2015	1	9	10	160	4.5	22	45	1005	0	10	1000

Figure 8.35. Specify Meteorological Data Form – "Hourly Observations" Tab

AERMOD's meteorological preprocessor, AERMET, also requires information about the underlying model domain surface characteristics. Click on the "Surface Characteristics" tab to specify unique sectors and associated surface characteristics (i.e., albedo, Bowen ratio, surface roughness). Multiple sectors can be specified, and the resulting sectors should cover the entire model domain (i.e., 360 degrees). In this example, a single "Shrubland" sector is specified (0 to 360 degrees). Values for "Albedo", "Bowen Ratio", and "Surface Roughness" can be selected by right-clicking in a given cell and selecting a value or by using the "Lookup" button to list tables of common values. Enter the surface characteristics as shown in Figure 8.36:

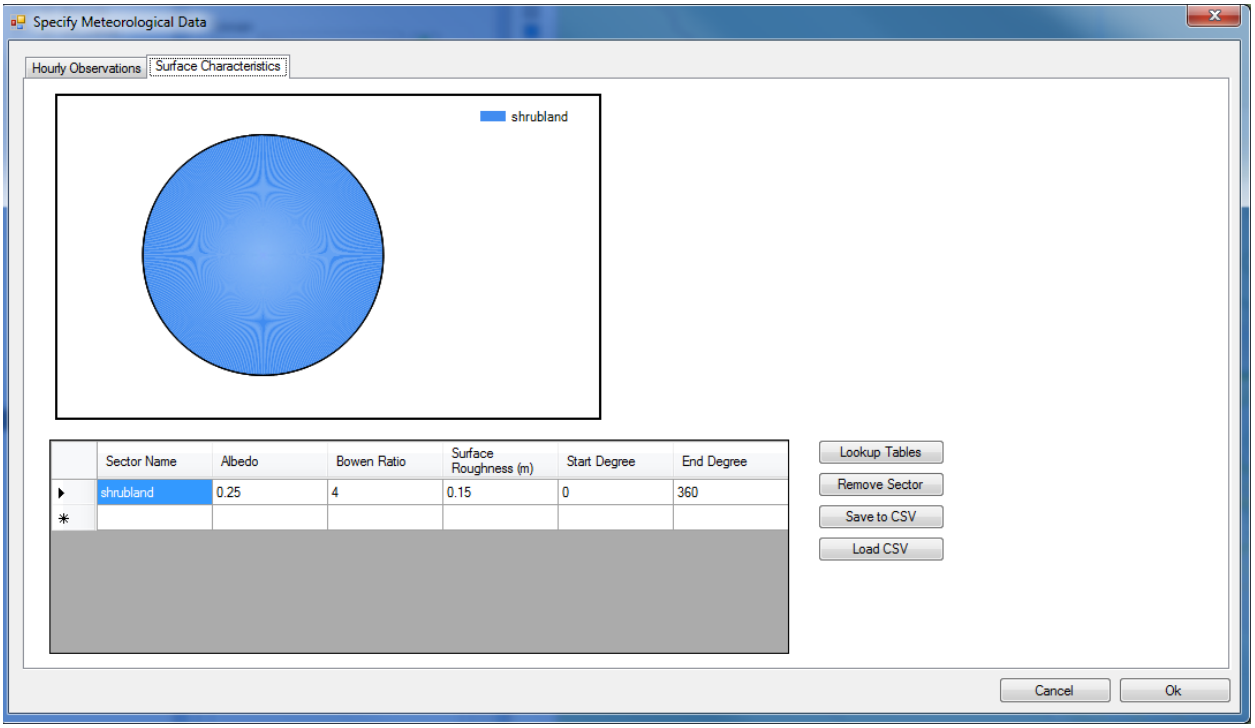


Figure 8.36. Specify Meteorological Data Form – "Surface Characteristics" Tab

8.3.10 Running DUSTRAN

After the sources, meteorology, and release duration information have been entered, a DUSTRAN simulation can be made. To run DUSTRAN, click on the "Run Simulation" button (Figure 8.37).

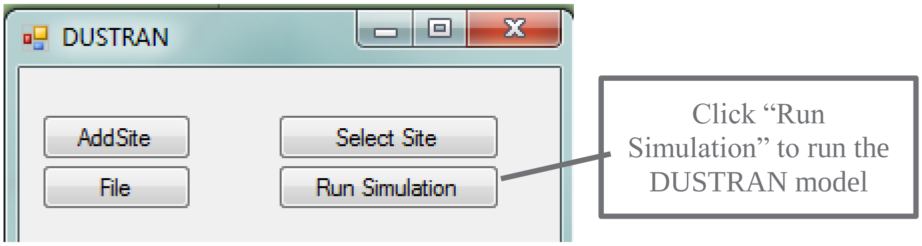


Figure 8.37. "Run Simulation" Button for Running DUSTRAN Simulation

8.3.11 Displaying Model Output

After the models finish running, click on the "Display Options" tab. Check "Contours" so that the plume contours will be displayed in the MapWindow map display (Figure 8.38). Note: AERMOD is a plume model and uses a single, hourly wind direction to transport the plume. As a result, there is no wind vector field to display within AERMOD.

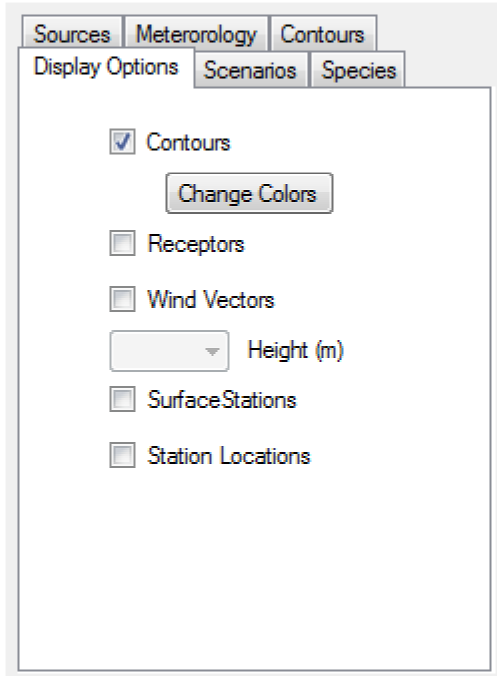


Figure 8.38. Display Options Tab – "Contours" options

8.3.12 Viewing Model Results

For each model time step, DUSTRAN calculates plume concentration and exposure as well as deposition and total deposition. To view a particular contour, click on the "Contours" tab and choose from the "Contour Types" listbox (Figure 8.39):

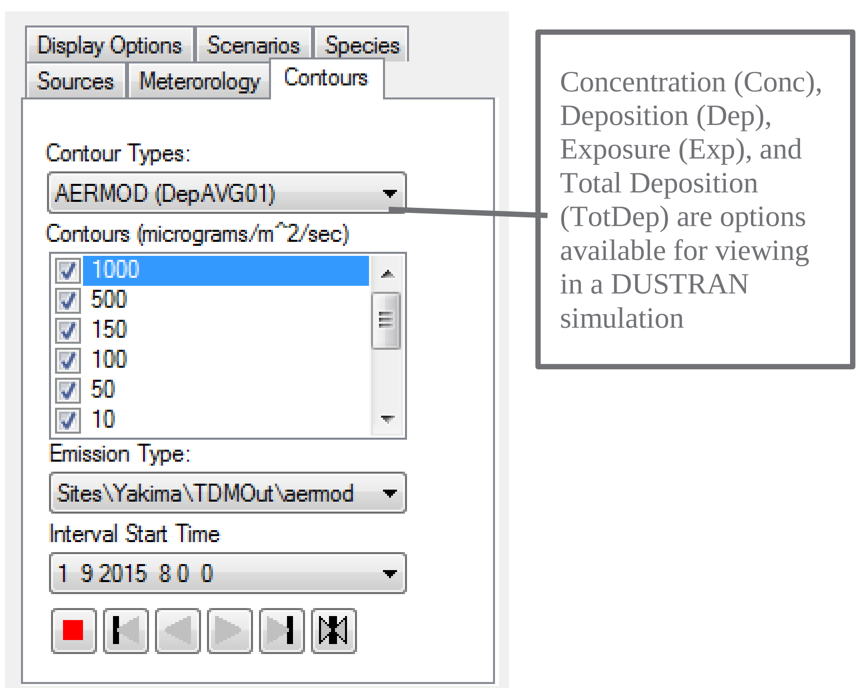


Figure 8.39. Contours Tab - "Contour Types" Listbox

In this example, select “Dep” to display hourly surface deposition values within MapWindow.

For a given contour type, numerous “Contours” are available for displaying. To display a particular contour interval, check the box next to the contour value. The default selection is normally adequate for displaying the maximum extent of the plume envelope.

To view a particular time step, select an interval from the “Interval Start Time” listbox. In this example, hourly time steps are available from the start of the release (7:00 a.m. local time) till the end of the run duration. Choose the 9:00 a.m. time step, which corresponds to the one hour (9:00 a.m. till 10:00 a.m.) average concentration and surface deposition (Figure 8.40):

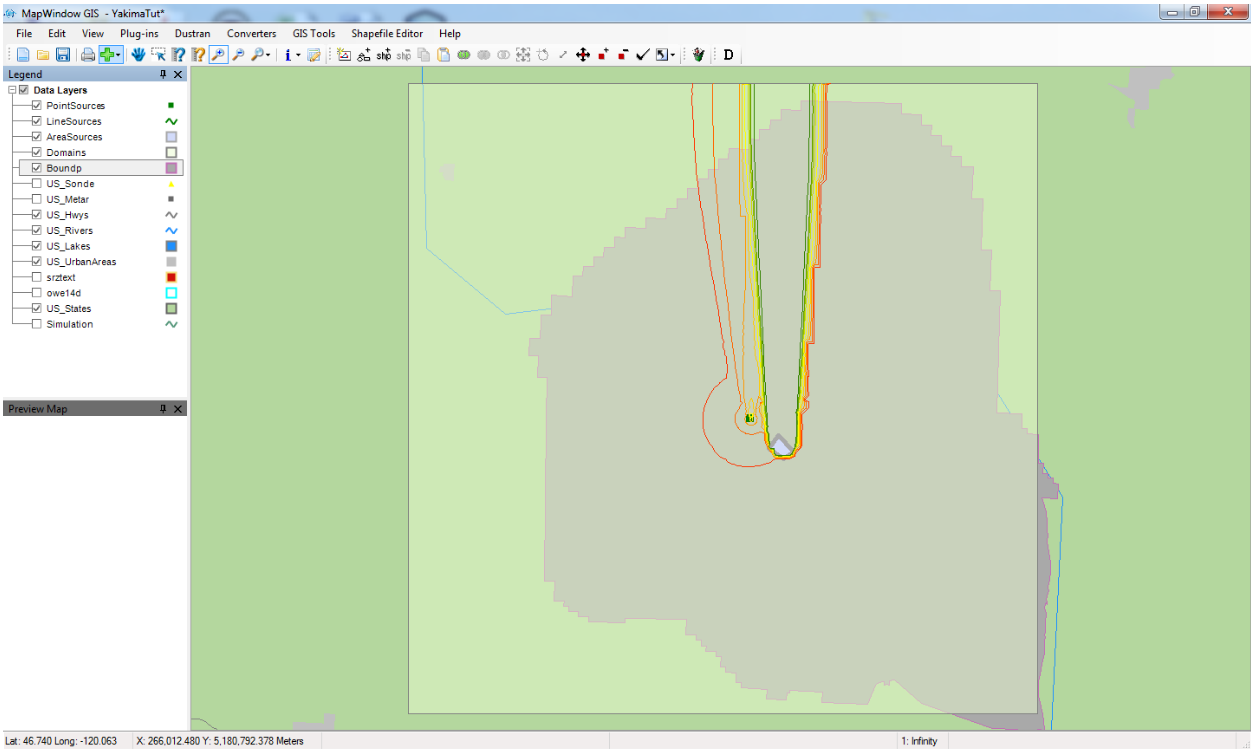


Figure 8.40. Display of Deposition Contours for the Hour from 9-10 AM

Notice in the above image that the deposition pattern from the two plumes (point and area-sources) are merged.

8.4 Simulating Wind-blown Dust Dispersion

This tutorial steps through the process of simulating a wind-blown dust scenario. Gridded dust emissions are created automatically by the emissions module and are a function of wind speed, soil texture, and vegetation class. In this scenario, a high wind event is assumed to occur over a 3-hour period, and the resulting wind-blown dust transport and diffusion is simulated.

8.4.1 Starting DUSTRAN

DUSTRAN is an integrated dispersion modeling application within the MapWindows GIS interface. To begin a DUSTRAN simulation, open MapWindows. On the MapWindow toolbar (Figure 8.41), click on the "D" button:

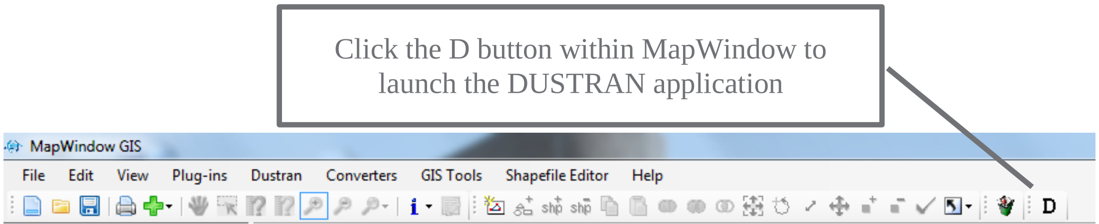


Figure 8.41. MapWindow Toolbar

8.4.2 Selecting a Site

The user interface to the DUSTRAN model will appear alongside the interface of the MapWindow application (Figure 8.42). Click the "Select Site…" button to open a dialog box which allows the user to select an existing site:

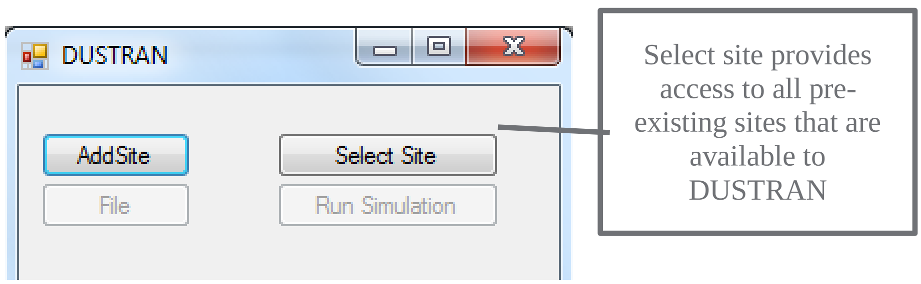


Figure 8.42. Portion of User Interface to the DUSTRAN Model

Select "Yakima" from the list of available model (Figure 8.43) sites and then click "Open":

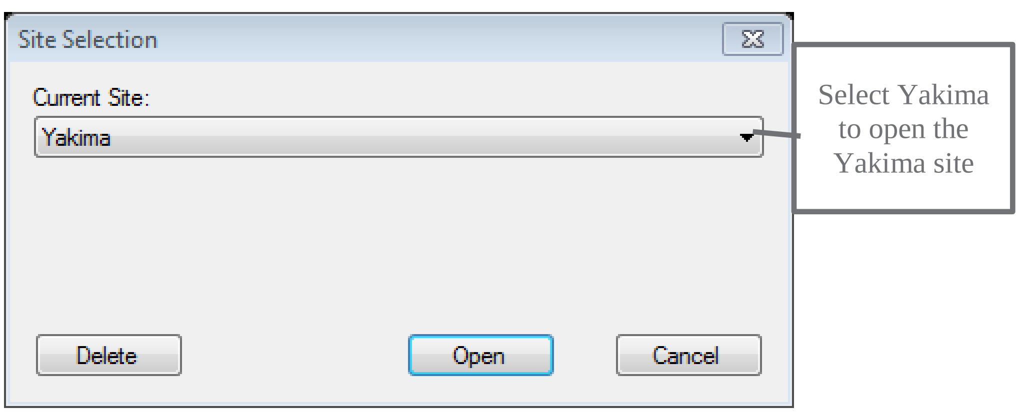


Figure 8.43. List of Available Model Sites

The Yakima site will open within the MapWindow map display. A list of available GIS data layers will appear in the layer list of the MapWindow application and DUSTRAN-specific input parameters will appear in the DUSTRAN user interface (Figure 8.44):

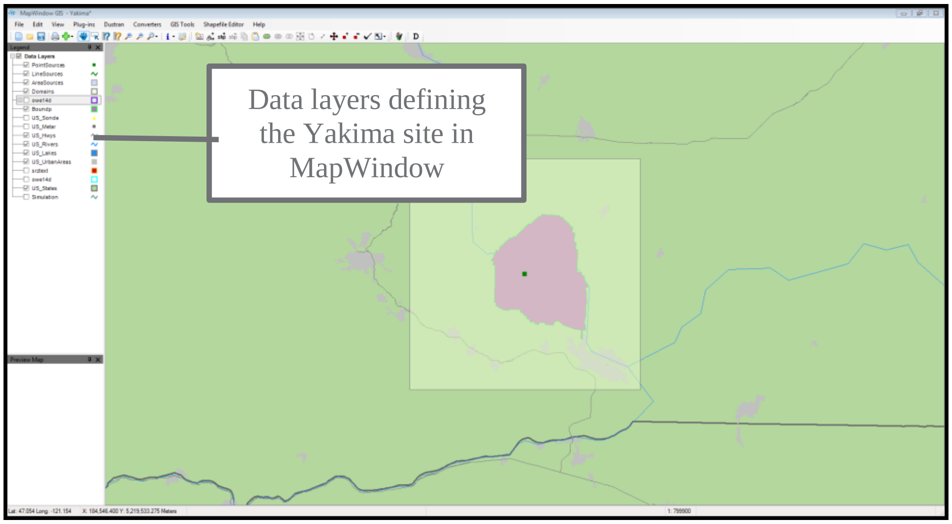


Figure 8.44. Display of Yakima Site

8.4.3 Defining the Domain

In a wind-blown dust simulation, the domain represents the area from which dust emissions will be generated. In addition, the domain is the area where both meteorological and dispersion calculations are performed.

In this example, an existing domain, called "Yakima," will be used. From the "Domain" panel within DUSTRAN, select the domain "Named" Yakima and set the domain "Size" to 200 km (Figure 8.45). The Yakima site should appear as follows within the MapWindow map display(Figure 8.46):

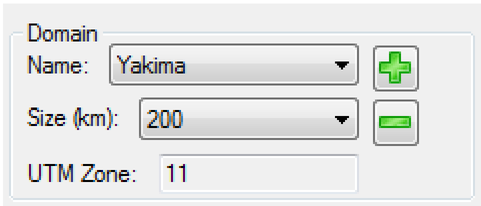


Figure 8.45. DUSTRAN Domain Panel Showing the Yakima Domain Selected with a 200 km Size

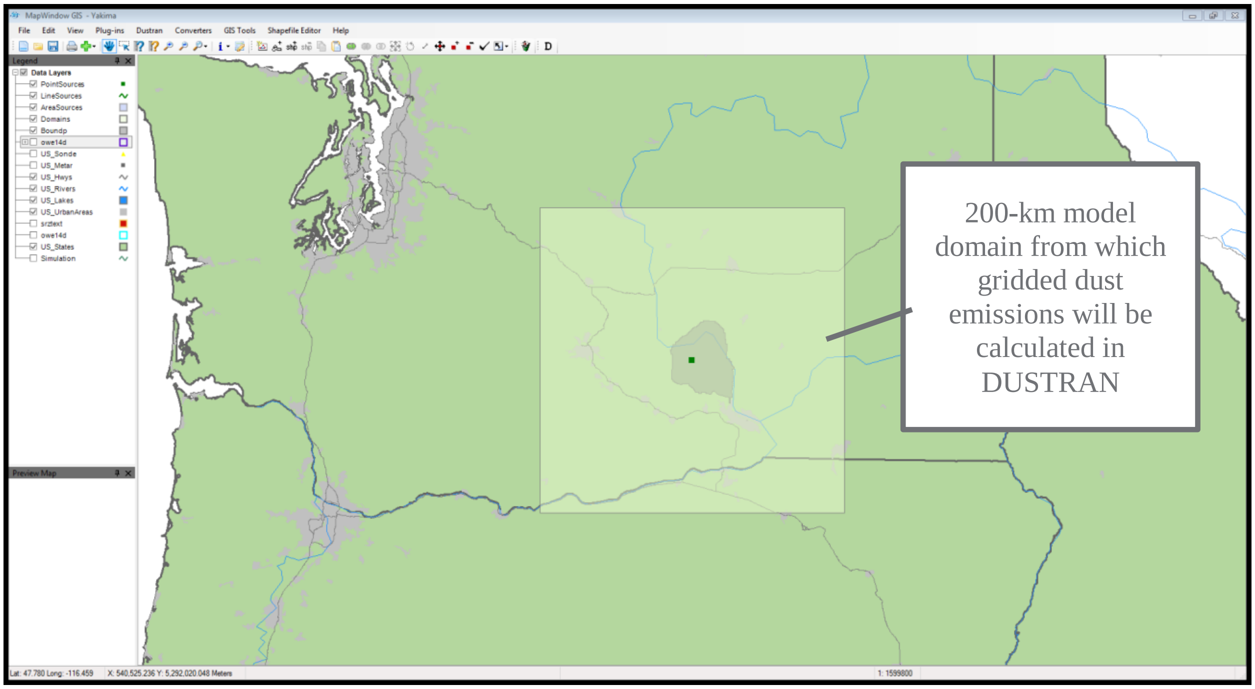


Figure 8.46. Display of 200-km-Square Yakima Domain within Yakima Site

8.4.4 Setting the Simulation Scenario

This scenario will simulate wind-blown dust during a 3-hour wind event that occurs in the early morning hours of April 15, 2005. In the "Simulation Scenario" panel, set the "Simulation Type" to "Wind-blown Dust" and the "Time Zone," "Start Date," "Start Time," and "Run Duration" as shown in Figure 8.47:

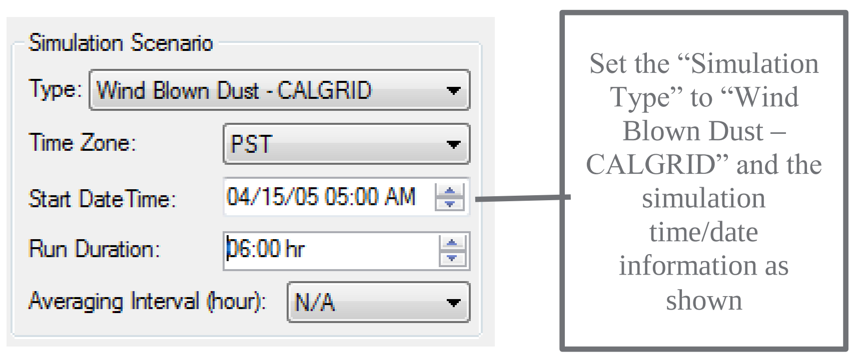


Figure 8.47. "Simulation Scenario" Panel

8.4.5 Setting the Soil and Vegetation Characteristic Files

Zobler soil texture and Olson Ecosystem vegetation class files are generated whenever a site is created using the "Add Site" wizard within the DUSTRAN interface; these files are required to calculate dust emissions for the domain. In addition, the "Polygon Layer Creator" utility can be used to create high-resolution characteristic files for direct use in DUSTRAN (see Polygon Layer Creator, Section 5.0).

In this example, the characteristic files that are generated with the site will be us To add soil and vegetation files, select the "Sources" tab and click on the "Characteristic Files" button (Figure 8.48):

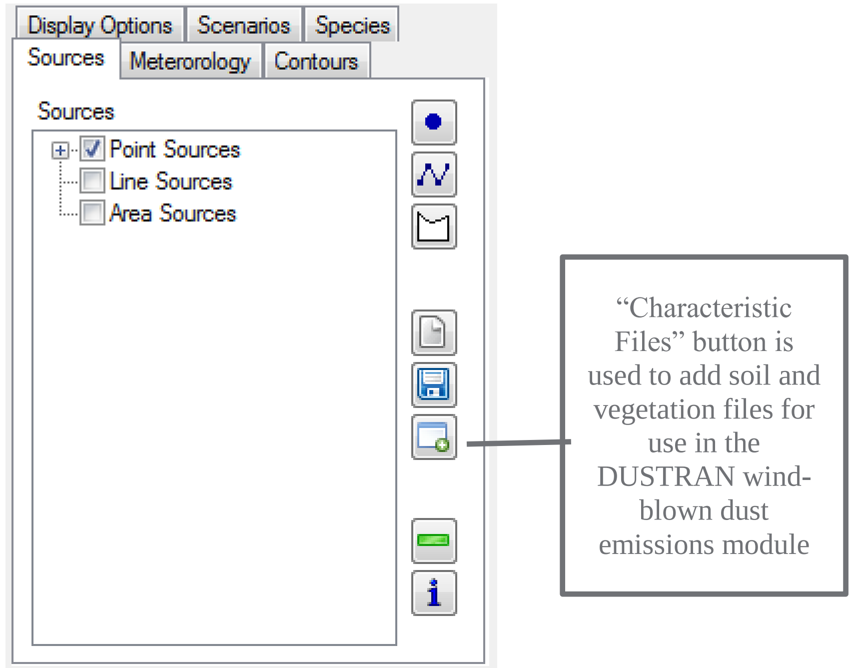


Figure 8.48. Adding Soil and Vegetation Files from DUSTRAN User Interface

After clicking the "Characteristics File" button, a form will appear that allows for the selection of both the "Soils" and "Vegetation" categories (Figure 8.49).

The standard soils file created for the site is called "srztext" and is selected by default. The file contains gridded values of Zobler soil textures for the site. Similarly, an Olson World Ecosystem gridded file, called "owe14d," exists for the vegetation category and is selected by default. Site-specific soils and vegetation files can be added for use in the emissions calculations; however, they must use the Zobler or Olson identification system (see Windblown Dust, Section 2.3.2 for more information on these classifications).

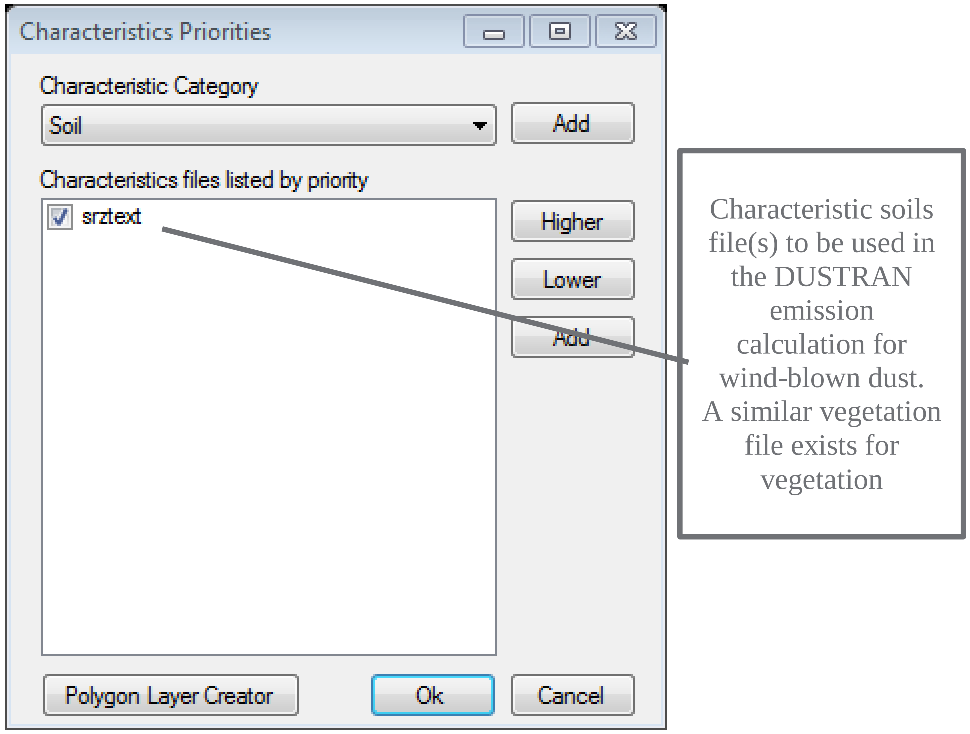


Figure 8.49. "Characteristics Priorities" Form

8.4.6 Viewing the Soil and Vegetation Characteristic Files

Shape files, which represent the various soil and vegetation categories, can be displayed within MapWindow so that potential dust-emission regions within the domain can be readily identified. Default shape files are automatically created with the site and stored in the "TerData" directory for the site. These files can be added to MapWindow as "Layers" for display.

These Zobler soil and Olson World Ecosystem shape files should have already been added to the Yakima site during the creation of the tutorial site. The layers are turned off by default, but can be activated by "checking" the box next to each layer's respective name.

To turn on the soil's layer, "check" the layer called "srztxt" (Figure 8.50). Similarly, to turn on the vegetation layer, "check" the layer called "owe14d":

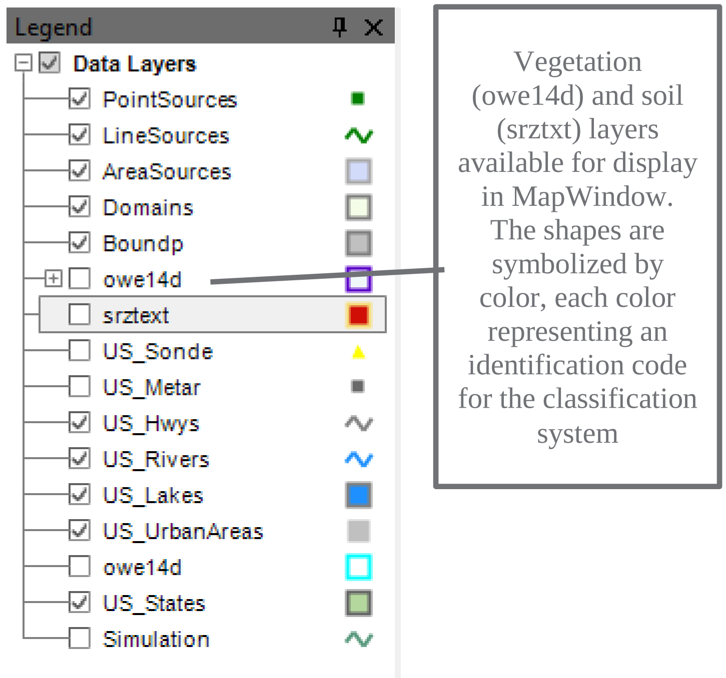


Figure 8.50. Display Showing Choices for Soil and Vegetation Layers

Figure 8.51 displays the Olson vegetation layer for the Yakima site. Each colored polygon represents an Olson World Ecosystem category ID, the values of which are displayed when the "Layer" is expanded. Of all the vegetation codes available, only four have been identified as allowing for effective wind-blown dust emissions. These categories, discussed in Section 2.3.2.4, include 8, 50, 51, and 52. For the Yakima site, only category 52—Cool/cold shrub, semi-desert/steppe exists within the domain; these are the red polygon regions and are the only potential source locations for wind-blown dust.

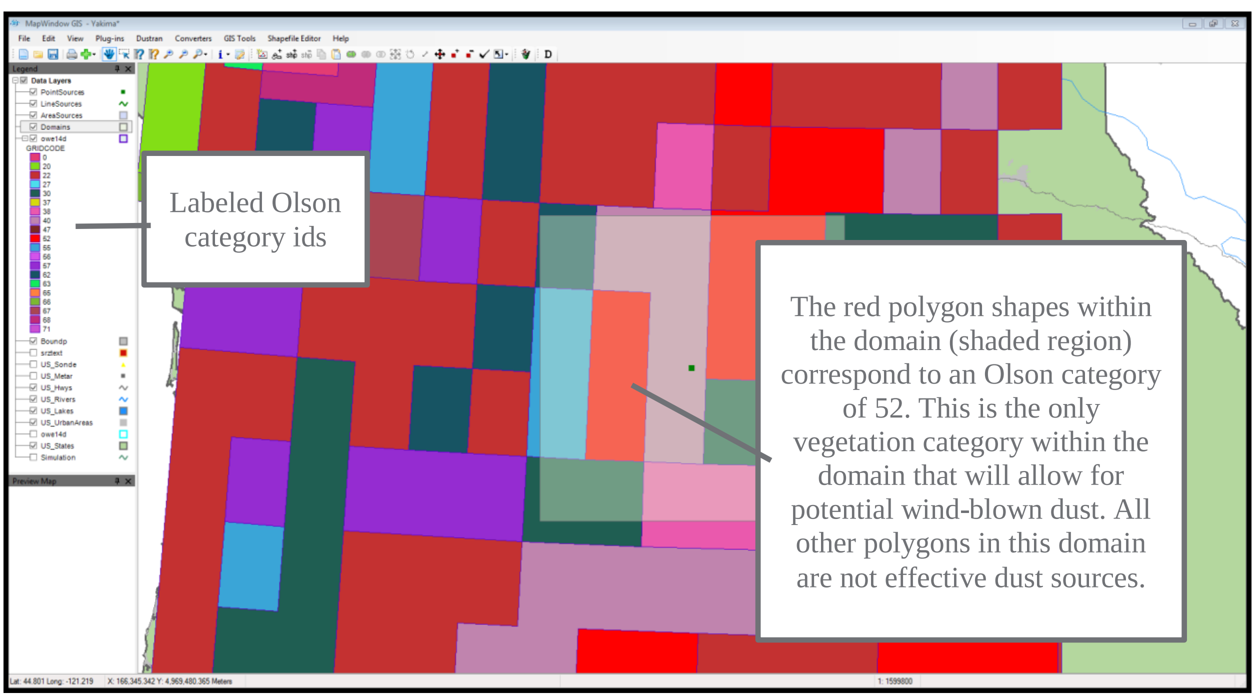


Figure 8.51. Display of Olson Vegetation Layer for the Yakima Site

8.4.7 Entering Meteorological Data

The final step before running the simulation is to set the meteorology. Click on the “Meteorology” tab within DUSTRAN and select “Single Observation” from the listbox. The “Specify Meteorological Data” form appears. Enter the meteorological parameters as shown in the form below. Select the “Atmospheric Stability” as “D – Neutral,” a wind speed of 10 m/s, and a wind direction from the southwest (240 degrees). The completed form will appear as shown in Figure 8.52:

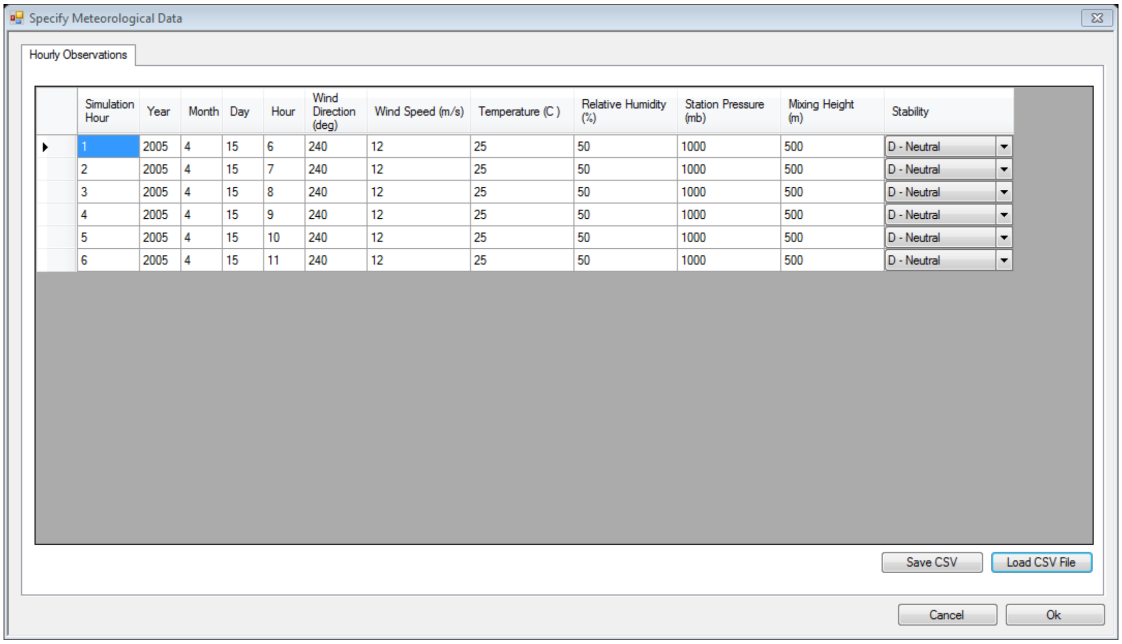


Figure 8.52. Completed “Specify Meteorological Data” Form

8.4.8 Running DUSTRAN

After the domain, simulation details, soil/vegetation categories, and meteorology have been entered, a wind-blown dust simulation in DUSTRAN can be made. To run DUSTRAN, click on the “Run Simulation” button (Figure 8.53).

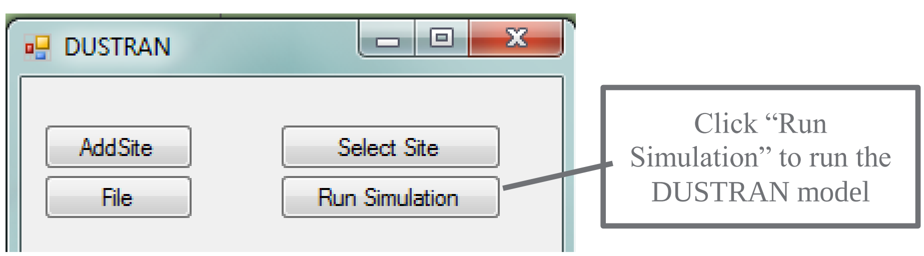


Figure 8.53. “Run Simulation” Button in DUSTRAN User Interface

8.4.9 Viewing Model Results

For each model time step, DUSTRAN calculates plume concentration and exposure as well as deposition and total deposition. To view a particular contour, click on the "Contours" tab and choose from the "Contour Types" listbox (Figure 8.54):

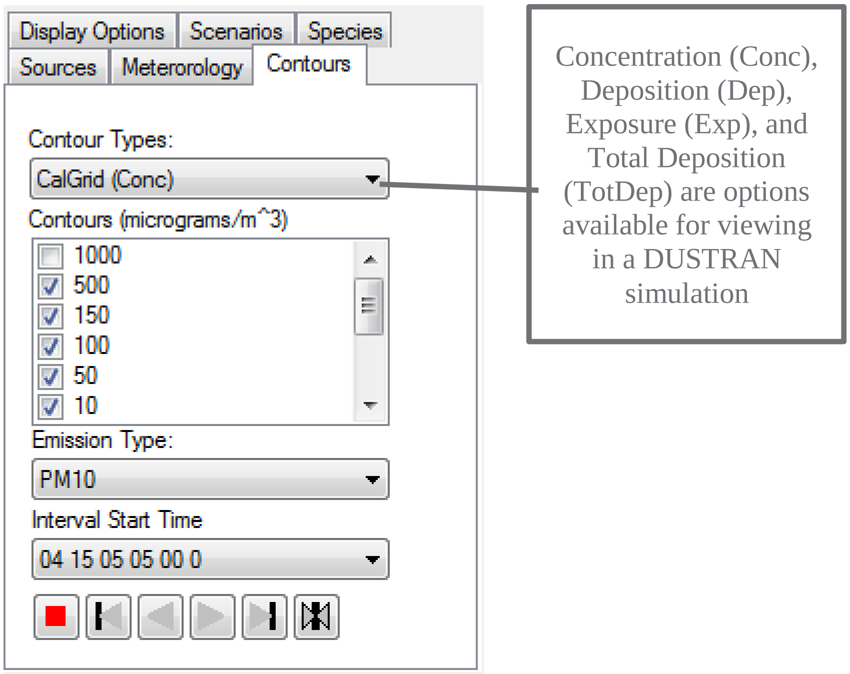


Figure 8.54. "Contours Types" Listbox

In this example, select "Conc" to display hourly concentrations within MapWindow.

For a given contour type, numerous "Contours" are available for display. To display a particular contour interval, check the box next to the contour value. The default selection is normally adequate for displaying the maximum extent of the plume envelope.

To view a particular time step, select an interval from the "Interval Start Time" listbox (Figure 8.55). In this example, hourly time steps are available from the start of the release (5:00 a.m. local time) till the end of the simulation. Choose the 7:00 a.m. time step, which corresponds to the one hour (7:00 a.m. till 8:00 a.m.) average concentration.

In the above image, the Olson vegetation coverage (colored polygons) and plume contours are displayed. Only the red polygons (Olson category 52—Cool/cold shrub, semi-desert/steppe) are potential source regions (as discussed previously) and are the only regions within the domain with plume contours.

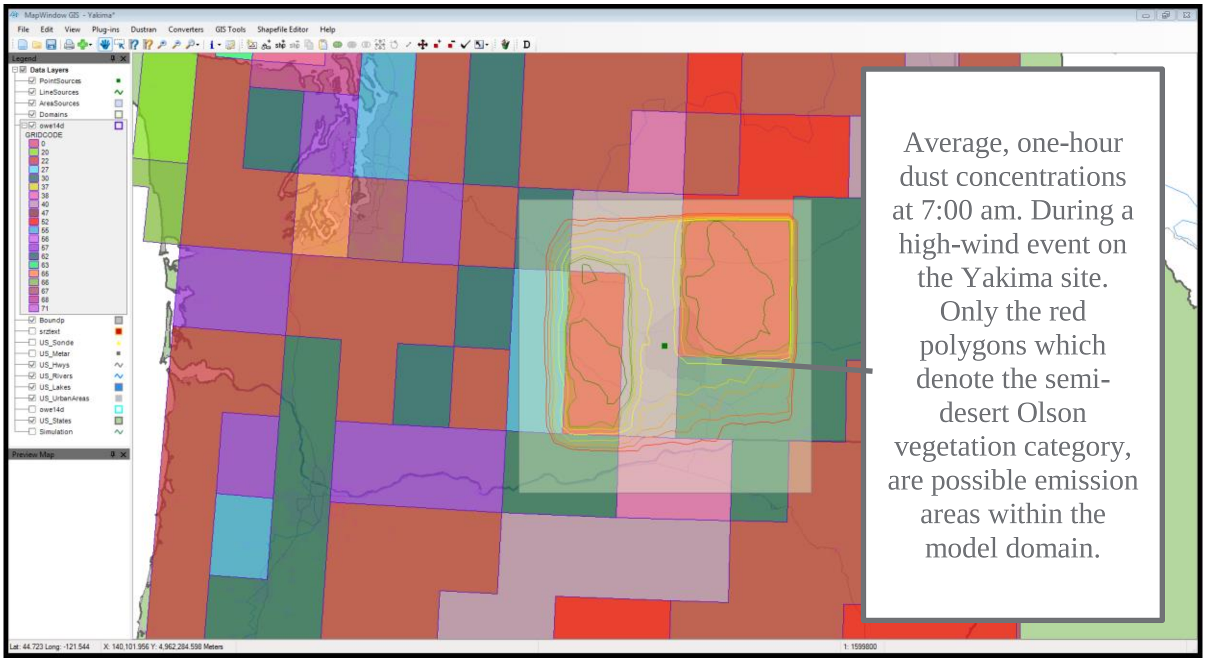


Figure 8.55. Display of Concentration Contours for the Hour from 7-8 AM

9.0 References

Allwine KJ Jr., FC Rutz, WJ Shaw, JP Rishel, BG Fritz, EG Chapman, BL Hoopes, and TE Seiple. 2006. “DUSTRAN 1.0 User's Guide: A GIS-Based Atmospheric Dust Dispersion Modeling System.” PNNL-16055, Pacific Northwest National Laboratory, Richland, WA.

Fecan F, B Marticorena, and G Bergametti. 1999. “Parameterization of the Increase of the Aeolian Erosion Threshold Wind Friction Velocity Due to Soil Moisture for Arid and Semi-arid Areas.” *Annales Geophys.* 17:194-157.

Gillette DA, and R Passi. 1988. “Modeling dust emission caused by wind erosion.” *J. Geophys. Res.* 93:14233-14242.

Gillies JA, V Etyemezian, H Kuhns, D Nikolic, and DA Gillette. 2005a. “Effect of Vehicle Characteristics on Unpaved Road Dust Emissions.” *Atmos. Environ.* 39:2341-2347.

Gillies JA, WP Arnott, V Etyemezian, H Kuhns, H Moosmuller, D DuBois, M Abu-Allaban, G Schwemmer, DA Gillette, WG Nickling, R Varma, T Wilkerson, and R Varma. 2005b. *Characterizing and Quantifying Local and Regional Particulate Matter Emissions from Department of Defense Installations*. Final report prepared for the Strategic Environmental Research and Development Program under Project CP-1191, March 2005.

Gillies JA, V Etyemezian, H Kuhns, H Moosmuller, J Engelbrecht, J King, S Uppapalli, G Nikolich, JD McAlpine, D Zhu, M Skiba, DA Gillette, WJ Shaw, R Hashmonay. 2010. Particulate Matter Emissions Factors for Dust from Unique Military Activities. Report prepared for the Strategic Environmental Research and Development Program under Project SI-1399, June 2010.

Irwin JS. 1998. *Interagency Workgroup on Air Quality Modeling (IWAQM) Phase 2 Summary Report and Recommendations for Modeling Long Range Transport Impacts.* EPA-454/R-98-019, Office of Air Quality Planning and Standards, Research Triangle Park, NC, 151 pp. (NTIS Accession Number PB 99-121089).

Kuhns H, J Gillies, V Etyemezian, G Nikolich, J King, D Zhu, S Uppapalli, J Engelbrecht, and S Kohl. 2010. “Effect of Soil Type and Momentum on Unpaved Road Particulate Matter Emissions from Wheeled and Tracked Vehicles.” *Aerosol Sci. Tech*. 44, 192-202.

Nickovic S, G Kallos, A Papadopoulos, and O Kakaliagou. 2001. “A Model for Prediction of Desert Dust Cycle in the Atmosphere.” *J. Geophys. Res.* 106, 18113-18129.

Olson JS. 1992. “World Ecosystems (WE1.4). Digital Raster Data on a 10-minute Cartesian Orthonormal Geodetic 1080 × 2160 grid.” In: *Global Ecosystems Database, Version 2.0*. Boulder, CO. Available at: http://www.ngdc.noaa.gov/seg/cdroms/ged_iia/datasets/a05/ow.htm (accessed 08-15-2006).

Scire JS, DG Strimaitis, and RJ Yamartino. 2000a. *A User's Guide for the CALPUFF Dispersion Model (Version 5)*. Earth Tech, Inc., Concord, MA.

Scire JS, FR Robe, ME Fernau, and RJ Yamartino. 2000b. *A User's Guide for the CALMET Meteorological Model (Version 5).* Earth Tech, Inc., Concord, MA.

Scire JS, RJ Yamartino, GR Carmichael, and YS Chang. 1989. *CALGRID: A Mesoscale Photochemical Grid Model Volume II – User's Guide.* California Resources Board, Sacramento, CA.

Shaw WJ, KJ Allwine, BG Fritz, FC Rutz, JP Rishel, and EG Chapman. 2008. "An Evaluation of the Wind Erosion Module in DUSTRAN." *Atmospheric Environment* 42(8):1907-1921. doi:10.1016/j.atmosenv.2007.11.022

Staub B, and C Rosenzweig. 1992. "Global Zobler Soil Type, Soil Texture, Surface Slope, and Other Properties. Digital Raster Data on a 1-degree Geographic (lat/long) 180 × 360 grid." In: *Global Ecosystems Database Version 2.0*. Boulder, CO, NOAA National Geophysical Data Center. Available at: http://www.ngdc.noaa.gov/seg/cdroms/ged_iia/datasets/a11/sr.htm (accessed 08-15-2006).

Tegen I, and I Fung. 1994. "Modeling of mineral dust in the atmosphere: sources, transport, and optical thickness." *J. Geophys. Res.* 99:22987-22914.

Turner DB. 1994. *Workbook of Atmospheric Dispersion Estimates: An Introduction to Dispersion Modeling*. Second Edition. ISBN 1-56670-023-X. CRC Press, Inc. Boca Raton, FL, 33431.

U.S. Environmental Protection Agency (EPA). 2004a. "AERMOD: Description of Model Formulation." EPA-454/R-03-004.

U.S. Environmental Protection Agency (EPA). 2004b. "User's Guide for the AMS/EPA Regulatory Model - AERMOD." EPA-454/B-03-001.

U.S. Environmental Protection Agency (EPA). 2004c. "User's Guide for the AERMOD Meteorological Preprocessor (AERMET)." EPA-454/B-03-002.

U.S. Environmental Protection Agency (EPA). 2004d. "AERMOD Deposition Algorithms – Science Document." Available at: http://www.epa.gov/scram001/7thconf/aermod/aer_scid.pdf.

U.S. Environmental Protection Agency (EPA). 2005. "Compilation of Air Pollutant Emission Factors (AP-42) Fifth Edition, Volume 1: Stationary Point and Area Sources." U. S. Environmental Protection Agency, Research Triangle Park, NC. Available at http://www.epa.gov/ttn/chief/ap42/.

U.S. Environmental Protection Agency (EPA). 2008. "AERSURFACE User's Guide." EPA-454/B-08-001.

U.S. Environmental Protection Agency (EPA). 2011. "User's Guide for the AERMOD Terrain Preprocessor (AERMAP)." EPA-454/B-03-003.

U.S. Environmental Protection Agency (EPA). 2014a. "ADDENDUM: User's Guide for the AMS/EPA Regulatory Model - AERMOD." EPA-454/B-03-001.

U.S. Environmental Protection Agency (EPA). 2014b. "ADDENDUM: User's Guide for the AERMOD Meteorological Preprocessor (AERMET)." EPA-454/B-03-002.

Appendix A

DUSTRAN Directory and File Documentation

Appendix A

DUSTRAN Directory and File Documentation

This appendix provides detailed information about the DUSTRAN directory structure and the supporting files used within the modeling system. The section begins with an overview of the folders and files that are within the root DUSTRAN directory. Input files that are stored within each site's "StaticData" directory are then defined. These files may be edited by an advanced user who wishes to control certain options not available within the DUSTRAN interface or control the behavior of the CALMET, CALPUFF, and CALGRID models. An overview of DUSTRAN's geodatabase is then given and is provided as a reference for users who wish to interact with the model data within MapWindow. Lastly, detailed information about the structure of the dust emission input and output files (Ldustinp.txt and Ldustout.txt) are provided.

A.1 DUSTRAN Directory

The DUSTRAN directory is the main directory used by the modeling system. This directory is normally a root directory on the machine's primary hard drive (e.g., c:\DUSTRAN). The main directory contains the executables used by the modeling system as well as the dynamic link library (DUSTRAN.dll) that integrates the modeling system with the MapWindow application. This directory also contains the individual site directories that are used within DUSTRAN simulations. The following table outlines the directories and files that are present in the DUSTRAN directory.

Name	**Description**
Aermap	Directory which contains the AERMAP executable.
Aermet	Directory which contain the AERMET executable.
Aemod	Directory which contains the AERMOD executable (EPA version [AERMOD.exe] and DRI version [AERMOD_DRI.exe]) and all files associated with a run including, the source emissions file, the AERMET generated surface (.sfc) and profile (.pfl) meteorological files, the AERMAP generated terrain files for receptors (.rec) and source (.src), and the AERMOD generated output file of concentration and deposition.
TerrFiles	Directory in which the land-use and terrain files used in the simulations are stored.
ModelFiles	Template model and processor input files are stored in this directory. The Add Site wizard uses these templates during the creation of a new site.
Site Directories	For each site present in the modeling system, a directory with the name of the site will be present. This directory is used to store all data that are specific to the site.
Areadust.exe	Wind-blown dust emission model executable.
CalConc.exe	Executable that processes the output of the CALPUFF and CALGRID models into the contours that are displayed in the map display of MapWindow.
Calgrid.exe	CALGRID dispersion model executable.
Calmet.exe	CALMET meteorological model executable.
Calpuff.exe	CALPUFF dispersion model executable.
Ctgproc.exe	Ctgproc land-use file processor executable.
LineDust.exe	Line and area dust-emission model executable.
Makegeo.exe	Makegeo land-use and terrain file processor executable.
Read62.exe	Upper air station meteorological data processor executable. This processor works with the Forecast System Laboratory (FSL) file format taken from the NOAA website.

Name	Description
Readmeviewer.exe	Executable that displays the readme.txt file before the first startup of DUSTRAN.
Terrel.exe	Terrain file processor executable.
Dustran.dll	DUSTRAN dynamic link library.
Calmet.inp	Input file used by the CALMET model. This file is generated by the modeling system before the execution of the model.
Calpuff.inp	Input file used by the CALPUFF model. This file is generated by the modeling system before the execution of the model.
Calgrid.inp	Input file used by the CALGRID model. This file is generated by the modeling system before the execution of the model.
Ctgproc.inp	Input file used by the Ctgproc processor. This file is generated by the modeling system before the execution of the processor.
Makegeo.inp	Input file used by the Makegeo processor. This file is generated by the modeling system before the execution of the processor.
Read62.inp	Input file used by the Read62 processor. This file is generated by the modeling system before the execution of the processor.
Terrel.inp	Input file used by the Terrel processor. This file is generated by the modeling system before the execution of the processor.
Calmet.lst	Output file generated by the CALMET model, which details the success or failure of the model run.
Calpuff.lst	Output file generated by the CALPUFF model, which details the success or failure of the model run.
Calgrid.lst	Output file generated by the CALGRID model, which details the success or failure of the model run.
Ctgproc.lst	Output file generated by the Ctgproc processor, which details the success or failure of the processor run.
Makegeo.lst	Output file generated by the Makegeo processor, which details the success or failure of the processor run.
Read62.lst	Output file generated by the Read62 processor, which details the success or failure of the processor run.
Terrel.lst	Output file generated by the Terrel processor, which details the success or failure of the processor run.
Convert.csv	Comma-separated text file containing the unit conversion factors used by the modeling system.
Apgemshelp.rtf	File that is displayed by the system when the Help option is selected.
Fileinstructions.rtf	File that is displayed by the system when the File Instructions option is selected.
Readme.rtf	File that is displayed by the "read me" viewer.
Baemarb.dat	Area-source input file used by the CALPUFF model. This file is generated by the modeling system based upon user inputs.
Lnemarb.dat	Line-source input file used by the CALPUFF model. This file is generated by the modeling system based upon user inputs.
Ptemarb.dat	Point-source input file used by the CALPUFF model. This file is generated by the modeling system based upon user inputs.
Ldustinp.txt	Line and area-source input file used by the line dust emission model. This file is generated by the modeling system based upon user input.
Ldustout.txt	Output file generated by the line dust emission model.

A.1.1 TerrFiles Directory

The TerrFiles directory within the root DUSTRAN directory is used as a storage location for a default set of elevation and land-use files. These files are used by the TERREL and CTGPROC processors to generate the terrain and land-use input data required by the CALMET model. The default set of files present in the directory represents data sets for North America. The following table details the default set of files that are present in this directory.

File Name	File Description
Nausgs2.img	U.S. Geological Survey (USGS) Land Use/Land Cover Scheme; used as input to the CTGPROC land-use processor for CALMET.
W100N40.dem	Global topographic digital elevation models with a horizontal grid spacing of 30 arc seconds (approximately 1 kilometer). Also known as GTOPO30 files, the files are tiled for ease-of-distribution. The four files listed in this directory spatially cover all of North America.
W100N90.dem	
W140N40.dem	
W140N90.dem	

A.1.2 ModelFiles Directory

The ModelFiles directory within the root DUSTRAN directory is used to store default template input files that are used by the preprocessors, CALMET, CALPUFF, and CALGRID models. When a new site is added to DUSTRAN using the "Add Site" wizard, these files are automatically copied into the new site's StaticData directory. The following table lists and describes the files that are stored in the ModelFiles directory.

File Name	File Description
Cal.par	Template copy of the default Cal.par file. The Cal.par file contains various model input controls that are not available from within the DUSTRAN interface. This template file is copied by the "Add Site" wizard and placed into the StaticData directory of the new site.
CalGeo.inp	CalGeo.inp is a template copy of the default input file for the TERREL, CTGPROC, and MAKEGO terrain and land-use preprocessing programs in DUSTRAN. The file is copied by the "Add Site" wizard and placed into the StaticData directory of the new site.
Calgrid.inp	Calgrid.inp is a template copy of the default input file for the CALGRID dispersion model in DUSTRAN. The file is copied by the "Add Site" wizard and placed into the StaticData directory of the new site.
Calmet.inp	Calmet.inp is a template copy of the default input file for the CALMET meteorological processor in DUSTRAN. The file is copied by the "Add Site" wizard and placed into the StaticData directory of the new site.
Calpuff.inp	Calpuff.inp is a template copy of the default input file for the CALPUFF dispersion model in DUSTRAN. The file is copied by the "Add Site" wizard and placed into the StaticData directory of the new site.
Calpost.inp	Calpost.inp is a template copy of the default input file for the CALPOST post-processor in DUSTRAN. The file is copied by the "Add Site" wizard and placed in the StaticData directory of the new site.
Pgems3.par	Template copy of the default Pgems3.par file used by the individual simulation sites. The file is copied by the "Add Site" wizard and placed into the StaticData directory of the new site.

A.1.3 Site Directories

When a new site is added through the "Add Site" wizard in DUSTRAN, a new directory is created for the site and is located within the DUSTRAN root directory. The site directory is used to store site-specific model input and output files. In addition, the site directory stores site-specific data that are used by the models. Normally, the data are stored in sub-directories based upon the type of preprocessor or model that uses or generates the data. For example, data associated with the meteorological models and processors are stored in the MetData, MetOut, and MetRaw directories. The following table details the directories and files that are found within each site directory present in the modeling system.

Name	Description

MetData	Directory used to store the meteorological data that have been processed for use in a simulation.
MetOut	Directory used to store the output files generated by the CALMET meteorological model. The files contain the gridded meteorological fields used by the CALPUFF and CALGRID dispersion models.
MetRaw	Default directory used to store historic meteorological data for a site that may be used in a simulation.
RsData	Directory used to store the run-specification files that are generated by DUSTRAN for each simulation.
StaticData	Directory used to store the model-input template files as well as the data associated with the sources and domains entered into the modeling system.
TDMOut	Directory used to store the data generated by the dispersion models during a simulation run.
TerData	Directory that can be used to store terrain and land-use files that are specific to the site. These files can be used instead of the default files found in the root TerrFiles directory, providing the static Calgeo.inp file for the site is modified to reflect the type and path of the new terrain files.
TerOut	Directory used to store the data generated by the land-use and terrain processors.
*.mdb	Access database that stores information related to the site, such as source information.
*.mwprj	MapWindow project file that stores the settings of the map display for the site.
PGEMS3.ini	Text file that stores the physical path of all the site-specific subdirectories list above.

A.1.3.1 MetData

The MetData directory within each site directory is used to store the data that are used as input by the meteorological model and processors. The data found in this directory include upper-air observations, surface observations, and files containing station characteristics.

File Name	**File Description**
Upn.dat	Upper air station meteorological data file used as input by the CALMET model. There will be one Upn.dat file present for each upper-air station selected for use during the simulation run.
Surf.dat	Surface-station meteorological data file used as input by the CALMET model.
*.upr	Upper-air-station meteorological data file generated from FSL data retrieved from the NOAA website.
*.unf	This text file contains a listing of the upper-air stations available for a site as well as descriptive information about each station. The station information found in this file includes station name, station ID, UTM coordinate location, and longitude and latitude of the station as well as the elevation.
*.snf	This text file contains a listing of the surface stations available for a site as well as descriptive information about each station. The station information found within this file includes station name, station ID, UTM coordinate location, and longitude and latitude of the station as well as the elevation of the station. The .snf file also contains the anemometer height used for the readings at the station.

A.1.3.2 MetOut

The MetOut directory within each site directory is used to store the different output files generated from the results of the meteorological model.

File Name	File Description
*.vec	Comma-separated text file containing wind-vector data used to generate the wind fields that are displayed within the MapWindow map display.
WindBlownData.txt	Comma-separated text file containing meteorological grid data pulled from the output of CALMET. These data are required as input by the wind-blown dust-emission model.
*.dat	Binary output data file generated by the CALMET meteorological model.

A.1.3.3 MetRaw

The MetRaw directory within each site directory is used to store the historical meteorological data that are available to the modeling system for a specific site. Currently, the modeling system is set up to process historical data for the Yakima and Fort Irwin simulation sites. While this directory is provided by default for each site, users are given the ability to store historical meteorological data in the directory of their choice whether they are located in the site or DUSTRAN directories. Historical data found in this directory are normally site specific and will not transfer across sites.

A.1.3.4 RsData

The RsData directory within each site directory stores the run-specification file that is generated by DUSTRAN each time a site simulation is executed. Through this directory, each of the models is given access to the specification file to retrieve inputs entered by the user before the simulation starts.

File Name	File Description
*.rs	The run specification file is a comma-separated text file that contains the data entered by the user through the user interface.

A.1.3.5 StaticData

The StaticData directory within each site directory contains the template input files that are used by the modeling system to create the actual input files used by the models and processors. The directory also contains the static parameter file Cal.par, which is used to hold model-run parameter data that are not accessible through the user interface. The Domain.dat and Sources.dat files are also stored within this directory and are used by the modeling system to track the characteristics of the domains and sources that have been entered for the site. The following table provides the details of the files that are stored within the StaticData directory.

File Name	File Description
Aermap.inp	Text file for the AERMAP model preprocessor. This file is used to retrieve the terrain and hill height scale factors for AERMOD model receptors and terrain heights for AERMOD sources. Terrain and hill height information is determined from a raster image, which is generated during site creation.
CalGeo.inp	Text file containing input sections for the Terrel, Ctgproc, and Makegeo processors. These input sections contain the default values for the individual processors and are used in the creation of the actual input file required for running the processors.
CALGRID.inp	Text file containing the default input parameters for the CALGRID model. This file is used in the creation of the actual input file for executing the CALGRID model for a simulation.
CALMET.inp	Text file containing the default input parameters for the CALMET model. This file is used in the creation of the actual input file for executing the CALMET model for a simulation.

File Name	File Description
CALPUFF.inp	Text file containing the default input parameters for the CALPUFF model. This file is used in the creation of the actual input file for executing the CALPUFF model for a simulation.
Cal.par	A static parameter file that contains inputs that are normally not changed (i.e., are considered static) for a modeling run, but can be modified for testing or fine tuning of the modeling system.
StationAlias.txt	Text file containing alias information for the National Training Center (NTC) surface station data. Currently, this file is only used for the Fort Irwin site.

A.1.3.6 TDMOut

The TDMOut directory within each site directory is used to store the output data generated by the CALGRID and CALPUFF dispersion models. The directory also contains the contouring text files that are generated by the CalConc processor based on the results generated by the dispersion models. Data contained within the .ccn and .crd files are used by the modeling system to create the contours that are shown on the DUSTRAN map display following the successful execution of a simulation. The file types that are found within this directory are listed in the following table.

File Type	File Type Description
*.ccn	Text file containing the concentration contour data generated from the output files of the CALGRID or CALPUFF models.
*.CON	Binary output file generated by the CALPUFF model.
*.crd	Text file containing the contouring grid coordinates.
*.dat	Binary output file generated by the CALGRID model.
*.dep	Text file containing the deposition contour data generated from the output files of the CALGRID or CALPUFF models.
*.exp	Text file containing the exposure contour data generated from the output files of the CALGRID or CALPUFF models.
*.pst	Text file containing the AERMOD deposition and concentration results.
*.tdp	Text file containing the total deposition contour data generated from the output files of the CALGRID or CALPUFF models.

A.1.3.7 TerOut

The TerOut directory within each site directory is used to store the output generated by the three terrain processors used by the modeling system. Output from the TERREL, CTGPROC, and MAKEGEO processors is stored within this directory following each successful simulation executed by the modeling system. The following table details the files that are stored within this directory.

File Type	File Type Description
*.out	Output file generated by the TERREL processor.
*.dat	Output files generated by the CTGPROC and MAKEGEO processors.

A.2 Site Directory Static Data Files

A.2.1 Cal.par File

Several parameters used by the DUSTRAN modeling system are not directly accessible through the DUSTRAN interface. Instead, these parameters are stored in a text file called “cal.par.” These parameters change infrequently and were intentionally omitted from the interface to minimize screen clutter and accidental user revisions.

The cal.par file is an ASCII, comma-delimited text file and can be opened in any standard text editor. The file is stored in the "StaticData" directory for each site, and the values in the file apply to that site. For the most part, the file is self-describing; variable-specific comments are provided before or on the line of the actual variable.

A sample cal.par file is provided in the following section. For clarity, each line (excluding lines that wrap) is numbered within the {} brackets. The line numbers are used to reference the table in Section A.2.1.2, which provides a detailed description of each line in the sample file. The {} brackets and line numbers do not exist in an actual cal.par file. Items in bold are comments and are not used by the code.

A.2.1.1 Example Cal.par File

{1} **Meterological parameters to construct profile from single observation and construct surface met file**
{2} **[WS, WD, Stability specified through single observation input window within DUSTRAN interface]**
{3} **A B C D E F G**
{4} 0.07,0.07,0.10,0.15,0.35,0.35,0.35,**Prural [Power Law exponents for wind profile; currently used]**
{5} 0.15,0.15,0.20,0.25,0.30,0.30,0.30,**Purban [Power Law exponents for wind profile]**
{6} -0.025,-0.020,-0.015,-0.010,0.010,0.025,0.040,**DtDz [Temperature lapse rate vs. stability]**
{7} 0.,20.,50.,100.,500.,2000.,9000., **zface heights used in the Calmet, Calpuff, and Calgrid models**
{8} 10,0 **Z0, ZTOP [Height of bottom and top sounding values (m agl)]**
{9} 11,**nHts [Number of heights in sounding]**
{10} 100,**ICEIL [Ceiling height (hundreds of feet)]**
{11} 0,**ICC [Opaque sky cover (tenths)]**
{12} 25,**TEMPK [Surface air temperature (Deg C)]**
{13} 50,**IRH [Relative humidity (percent)]**
{14} 1000,**PRES [Station pressure (mb)]**
{15} 1000,**WATDENSE [Density of water (g/m^3)]**
{16} 0,**IPCODE [Precipitation code (0=no precip; 1-18=liquid; 19-45=frozen)]**
{17} T,**WSPROF [T - Power law profile; F - Constant wind speed with height]**
{18} F,**MM5 data used check**
{19} F,**Advanced user Deposition flag; if T, then the ability to set deposition velocities is turned on**
{20} 50,50, **Number of x and y grid points for CALPUFF run**
{21} 20,20**, Number of x and y grid points for CALGRID run**
{22} 100,100**, Number of x and y grid points for AERMOD run**
{23} T, **Flag used to set if the Aermod simulation will or will not use polar grids**
{24} **Species names, molecular weight, particle mean diameter, particle standard deviation, default user-defined deposition velocity**
{25} 4,**NumSpecs [Number of species present] Molecular weight does not appear to be used by Calpuff**
{26} PM10,0.0,10.0,1.0,0.0
{26} PM2.5,0.0,2.5,1.0,0.0
{26} PM15,0.0,15.0,1.0,0.0
{26} PM30,0.0,30.0,1.0,0.0
{27} Point Source Parameter Units,8,
{28} Parameter,Release Height,Exit Velocity,Exit Temperature,Stack Diameter,Building Cross Section,Initial Horizontal Plume Size,Initial Vertical Plume Size,Release Rate,
{29} Default Value,0,0,25,1,0,1,1,1,
{30} User Units,m,m/s,C,m,m^2,m,m,g/s,
{31} Model Units,m,m/s,C,m,m^2,m,m,g/s,
{32} Conversion Factor,1,1,1,1,1,1,1,1,
{33} Min Value,0,0,inf,0,0,1,1,0,

{34} Max Value,inf,inf,inf,inf,inf,inf,inf,inf,
{35} Range String,Release Height >= 0.0,Exit Velocity >= 0.0,No Boundaries,Stack Diameter >= 0,Building Cross Section >= 0.0,Initial Horizontal Plume Size 0.0,Initial Vertical Plume Size > 0.0,Release Rate >= 0.0,
{36} Line Source Parameter Units,3,
{37} Parameter,HTL,ELEVL,QEMITL,
{38} Default Value,0,0,0,
{39} User Units,m,m,g/s/m,
{40} Model Units,m,m,g/s/m,
{41} Conversion Factor,1,1,1,
{42} Min Value,0,0,0,
{43} Max Value,inf,inf,inf,
{44} Range String,HTL >= 0.0,ELEVL >= 0.0,QEMITL >= 0.0,
{45} Area Source Parameter Units,7,
{46} Parameter,HT,ELEV,TEMPK,WEFF,REFF,SIGZ,QEMIT,
{47} Default Value,0,0,25,1,1,1,
{48} User Units,m,m,C,m/s,m,m,g/s
{49} Model Units,m,m,K,m/s,m,m,g/s
{50} Conversion Factor,1,1,1&0,1,1,1,1,
{51} Min Value,0,0,inf,1,1,1,0,
{52} Max Value,inf,inf,inf,inf,inf,inf,inf,
{53} Range String,HT >= 0.0,ELEV >= 0.0,No Boundaries,WEFF >= 1.0,REFF >= 1.0,SIGZ >= 1.0,QEMIT >= 0.0,
{54} Meteorology Parameter Units,3,
{55} Parameter,Wind Speed,Wind Direction,Mixing Height,
{56} Default Value,2.2,270,500,
{57} User Units,m/s,No Unit,m,
{58} Model Units,m/s,No Unit,m,
{59} Conversion Factor,1,1,1,
{60} Min Value,0,0,0,
{61} Max Value,inf,360,inf,
{62} Range String,Wind Speed >= 0.0,Wind Direction >= 0 and Wind Direction <= 360,Mixing Height >= 0.0,
{63} Line Source Vehicle Parameter,4,
{64} Parameter,Vehicle SpeedL,Vehicle WeightL,Road LengthL,Distance TraveledL,
{65} Default Value,50,3000,0.0,0.0,
{66} User Units,km/hr,kg,km,km,
{67} Model Units,km/hr,kg,km,km,
{68} Conversion Factor,1,1,1,1,
{69} Min Value,0,0,0,0,
{70} Max Value,inf,inf,inf,inf,
{71} Range String,Vehicle Speed >= 0.0,Vehicle Weight >= 0.0,Road Length >= 0.0,Distance Traveled >= 0.0,
{72} Area Source Vehicle Parameter,4,
{73} Parameter,Vehicle Speed,Vehicle Weight,Polygon Area,Distance Traveled,
{74} Default Value,50,3000,0.0,0.0,
{75} User Units,km/hr,kg,km^2,km,
{76} Model Units,km/hr,kg,km^2,km,
{77} Conversion Factor,1,1,1,1,
{78} Min Value,0,0,0,0,
{79} Max Value,inf,inf,inf,inf,
{80} Range String,Vehicle Speed >= 0.0,Vehicle Weight >= 0.0,Polygon Area > 0.0,Distance Traveled >= 0.0,
{81} 1.0E+6,Contours (micrograms/m^3),**Unit label and conversion factor for the concentration contour labels**
{82} 1.0E+6,Contours (micrograms/m^2/sec),**Unit label and conversion factor for the deposition contour labels**
{83} 1.0E+6,Contours (micrograms-sec/m^3),**Unit label and conversion factor for the exposure contour labels**
{84} 1.0E+6,Contours (micrograms/m^2),**Unit label and conversion factor for the total deposition contours**
{85} 1.0E-15,**Minimum contour level used by Calconc suggest 3 orders of magnitude smaller than the smallest specified contour level**
{86} 11,**Number of contour levels to be calculated**

{87} 1.0E-10
{87} 1.0E-9
{87} 1.0E-8
{87} 1.0E-7
{87} 1.0E-6
{87} 10.0E-6
{87} 50.0E-6
{87} 100.0E-6
{87} 150.0E-6
{87} 500.0E-6
{87} 100.0E-5
{88} 17, **NUMVEHCODES [Number of vehicle integer codes]**
{89} 1,1176,Dodge Neon, 2002 Civilian vehicle with Eagle GA Touring M+S P185/165R 85T tires
{89} 2,1759,Dodge Caravan,2002 Civilian vehicle with GoodYear Integrity M+S 215/70R15 98S tires
{89} 3,1516,Ford Taurus,2002 Civilian vehicle with Firestone M+S P215/60R16 94T tires
{89} 4,3100,GMC G20 Van, DRI TRAKER vehicle used for measuring dust emissions in real time
{89} 5,5227,GMC C5500, 1999 Civilian vehicle 6 wheels with GoodYear and Michelin tires
{89} 6,2445,M998 HMMWV, Military Vehicle 4 wheels with tires
{89} 7,14318,M923A2 (5-Ton), Military Vehicle 2 front wheels and 8 rear wheels on dual axles
{89} 8,8060,M1078 LMTV, 2.5 Ton Military vehicle with 4 wheels and tires
{89} 9,20000,M977 HEMTT, Military vehicle with 8 wheels and tires
{89} 10,23636,Freightliner, Tractor trailer rig with 22 wheels and tires
{89} 11,8982, M915A4 Truck, Military line-haul wheeled tractor without trailer
{89} 12,10000,M113 APC,Tracked military vehicle armored personnel carrier
{89} 13,12727,M577 Command Post,Tracked military vehicle armored mobile command post
{89} 14,23636,M2 Bradley,Tracked military vehicle armored infantry fighting vehicle
{89} 15,25000,M270 MLRS,Tracked military vehicle armored multiple launch rocket system
{89} 16,50500,M88 Hercules,Tracked military vehicle armored heavy equipment recovery vehicle
{89} 17,60000,M1A1 Abrams,Tracked military vehicle battle tank
{90} **Calpuff polar grid information**
{91} 1
{92} **POLAR_GRD**
{93} 36,10,200.,400.,800.,1500.,2500.,3500.,4500.,5500.,6500.,7500.,
{94} 3,1000,**number of grid spacings used to create area source grids - resolution of area source grid (*** Note Calpuff will only accept up to 4000 discrete receptors ***)**
{95} 2,2, **Wind vector resolution, max windspeed**
{96} 1.0E-20, **Initial concentration value for CalGrid model**
{97} 1,**Model run flag 1=Calpuff run 2=Calgrid run 3=Both models run**
{98} 1,**Run Calgrid in wind blown dust mode 1=Run wind blown 0=Run without windblown dust**
{99} 0.0,**soil moisture (water mass / soil mass)**
{100} http://www.irwin.army.mil/weather/WXdata.txt
{101}
http://raob.fsl.noaa.gov/intl/GetRaobs.cgi?shour=All+Times&ltype=All+Levels&wunits=Knots&access=WBAN+Station+Identifier&view=NO&osort=Station+Series+Sort&oformat=FSL+format+(ASCII+text)

A.2.1.2 Cal.par File Description

Line Number	Data Type	Description
1	String	Wind profile description header.
2	String	Wind profile description header.
3	String	Wind profile description header.
4	Floating Point	Rural Power Law exponents for wind profile. The wind profile is generated for simulations using the single observation meteorology option. (defaults Pr1=0.07, Pr2=0.07, Pr3=0.10, Pr4=0.15, Pr5=0.35, Pr6=0.35, Pr7=0.35)

Line Number	Data Type	Description
5	Floating Point	Urban Power Law exponents for wind profile. The wind profile is generated for simulations using the single observation meteorology option. (defaults Pu1=0.15, Pu2=0.15, Pu3=0.20, Pu4=0.25, Pu5=0.30, Pu6=0.30, Pu7=0.30)
6	Floating Point	Potential temperature lapse rate vs. stability for wind profile. The wind profile is generated for simulations using the single observation meteorology option. (defaults Pt1=-0.025, Pt2=-0.20, Pt3=-0.015, Pt4=-0.010, Pt5=0.010, Pt6=0.025, Pt7=0.40)
7	String	Comma-separated list of heights (meters above ground) for CALMET's vertical grid. The CALMET-derived parameters occur at the mid-point between successive heights. The number of heights is determined from the number of values listed minus one (the zero value).
8	Floating Point	Height (meters above ground level [agl]) of lowest observation in upper-air observation profile for "Single Observation" meteorology
8	Floating Point	Height (meters agl) of the top observation in the upper-air observation profile for "Single Observation" meteorology. If ZTOP equals 0.0, then the default value found in the static Calmet.inp file is used.
9	Integer	Number of heights in the upper-air observation profile, for "Single Observation" meteorology.
10	Integer	Default ceiling height (hundreds of feet), for "Single Observation" meteorology.
11	Integer	Default opaque sky cover (tenths), for "Single Observation" meteorology.
12	Floating Point	Default surface air temperature in (deg C), for "Single Observation" meteorology.
13	Integer	Default relative humidity (%), for "Single Observation" meteorology.
14	Floating Point	Default station pressure (mb), for "Single Observation" meteorology.
15	Floating Point	Density of water (kg/m^3). Used in the conversion of AGDISP output to emission rates.
16	Integer	Precipitation code (0 = no precipitation, 1 – 18 = liquid, 19 – 45 = frozen).
17	Character (T*rue* / F*alse*)	Power law profile flag (T = Power law profile, F = Constant wind speed with height). [(default T])
18	Character (T*rue* / F*alse*)	Flag designating whether or not MM5 data are to be used by the modeling system.
19	Character (T*rue* / F*alse*)	Flag designating whether to turn on the advanced user parameters for the calculation of deposition. If this flag = T, then the user has the ability to enter geometric standard deviations and deposition velocities.
20	Integer	Grid spacing and number of receptor locations in the X and Y directions for a CALPUFF simulation.
21	Integer	Grid spacing and number of receptors locations in the X and Y directions for a CALGRID simulation.
22	Integer	Grid spacing and number of receptors locations in the X and Y directions for a AERMOD simulation.
23	Character (T/ F)	Flag used to set if the AERMOD simulation will or will not use polar grids
24	String	Species data header.
25	Integer	Number of species available in DUSTRAN.
26	String	Name of the species, molecular weight, particle mean diameter, particle standard deviation, default deposition velocity
27	Integer	Number of parameters used by a point source
28	String	Label strings for point-source parameters separated by commas.
29	String	Default values for point-source parameters separated by commas.
30	String	User units for point-source parameters separated by commas.
31	String	Model units for point-source parameters separated by commas.
32	String	Conversion values for point-source parameters separated by commas.
33	String	Minimum values for point-source parameters separated by commas.
34	String	Maximum values for point-source parameters separated by commas.

Line Number	Data Type	Description
35	String	Strings giving the range of values that are appropriate for point-source parameters separated by commas.
36	Integer	Number of parameters used by a line source.
37	String	Label strings for line-source parameters separated by commas.
38	String	Default values for line-source parameters separated by commas.
39	String	User units for line-source parameters separated by commas.
40	String	Model units for line-source parameters separated by commas.
41	String	Conversion values for line-source parameters separated by commas.
42	String	Minimum values for line-source parameters separated by commas.
43	String	Maximum values for line-source parameters separated by commas.
44	String	Strings giving the range of values that are appropriate for line-source parameters separated by commas.
45	Integer	Number of parameters used by an area source.
46	String	Label strings for area-source parameters separated by commas.
47	String	Default values for area-source parameters separated by commas.
48	String	User units for area-source parameters separated by commas.
49	String	Model units for area-source parameters separated by commas.
50	String	Conversion values for area-source parameters separated by commas.
51	String	Minimum values for area-source parameters separated by commas.
52	String	Maximum values for area-source parameters separated by commas.
53	String	Strings giving the range of values that are appropriate for area-source parameters separated by commas.
54	Integer	Number of parameters used for meteorology input.
55	String	Label strings for meteorology input parameters separated by commas.
56	String	Default values for meteorology input parameters separated by commas.
57	String	User units for meteorology input parameters separated by commas.
58	String	Model units for meteorology input parameters separated by commas.
59	String	Conversion values for meteorology input parameters separated by commas.
60	String	Minimum values for meteorology input parameters separated by commas.
61	String	Maximum values for meteorology input parameters separated by commas.
62	String	Strings giving the range of values that are appropriate for meteorology input parameters separated by commas.
63	Integer	Number of parameters used for line-source vehicle input.
64	String	Label strings for line-source vehicle input parameters separated by commas.
65	String	Default values for line-source vehicle input parameters separated by commas.
66	String	User units for line-source vehicle input parameters separated by commas.
67	String	Model units for line-source vehicle input parameters separated by commas.
68	String	Conversion values for line-source vehicle input parameters separated by commas.
69	String	Minimum values for line-source vehicle input parameters separated by commas.
70	String	Maximum values for line-source vehicle input parameters separated by commas.
71	String	Strings giving the range of values that are appropriate for line-source vehicle input parameters separated by commas.
72	Integer	Number of parameters used for area-source vehicle input.
73	String	Label strings for area-source vehicle input parameters separated by commas.
74	String	Default values for area-source vehicle input parameters separated by commas.
75	String	User units for area-source vehicle input parameters separated by commas.
76	String	Model units for area-source vehicle input parameters separated by commas.
77	String	Conversion values for area-source vehicle input parameters separated by commas.
78	String	Minimum values for area-source vehicle input parameters separated by commas.
79	String	Maximum values for area-source vehicle input parameters separated by commas.
80	String	Strings giving the range of values that are appropriate for area-source vehicle input parameters separated by commas.

Line Number	Data Type	Description
81	Floating Point, String	Conversion factor for the concentration contour labels, Unit label for the concentration contour labels
82	Floating Point, String	Conversion factor for the deposition contour labels, Unit label for the deposition contour labels.
83	Floating Point, String	Conversion factor for the exposure contour labels, Unit label for the exposure contour labels
84	Floating Point, String	Conversion factor for the total deposition contour labels, Unit label for the total deposition contour labels
85	Floating Point	Minimum concentration level to be used in the processing of the contours from the generated results.
86	Integer	The number of contour levels to be processed from the generated results.
87	Floating Point	The individual contour levels to be used in the processing of the generated results.
88	Integer	Number of vehicles available in the modeling system.
89	Integer, Floating Point, String, String	Vehicle ID number, Vehicle weight, Vehicle name, Vehicle description
90	String	Polar grid header line.
91	Integer	Number of polar grids to be used in the simulation run.
92	String	Name of the polar grid to be used in the simulation run.
93	Integer, Integer, Floating Point	Number of radial distances used in the generation of the point-source polar grids, Number of arc distances (NARCS) used in the generation of the point-source polar grids, Comma-separated list of arc distances used in the generation of the point-source polar grids (there must be NARCS present in the file)
94	Integer, Integer	Parameters used to specify "sub-grid" Cartesian receptor grids. The first is the number of "main" Cartesian grid points extending beyond the area-source boundaries, and the second number is the spacing (m) between the "sub-grid" receptor points.
95	Integer, Floating Point	Two parameters for setting the resolution of the wind vector field displayed on the map and for setting the maximum wind speed used in the creation of the wind vector field. The first parameter indicates to display every "nth" vector, and the second parameter is the magnitude of the vector spanning the distance between the vector output grid points.
96	Floating Point	Initial concentration value to be used by the CALGRID model.
97	Integer	Flag used to determine which dispersion model to execute for the simulation run. (1=CALPUFF run 2=CALGRID run 3=Both models run). Currently, model selection is being handled by the interface (i.e., Source Emissions = CALPUFF; Windblown Dust = CALGRID) and not through this flag.
98	Integer	Flag used to determine whether the CALGRID run is to be run in the windblown mode. (1=Run wind-blown model 0=Run CALGRID without windblown dust)
99	Floating Point	Soil moisture for each quadrant. (water mass/soil mass)
100	String	URL address of the website used to retrieve surface meteorological data for NTC.
101	String	URL address of the website used to retrieve upper air station data for NTC.

A.2.2 Calmet.inp

The CALMET input file (Calmet.inp) is located in the "StaticData" directory for each site within DUSTRAN and is the template input file for CALMET. Its contents are merged with select-user input from the DUSTRAN interface for each simulation. Many of the parameters contained within Calmet.inp control certain aspects of the CALMET-derived gridded wind field and boundary-layer parameters. These parameters are not available from the DUSTRAN interface and can only be edited via the static file.

The static Calmet.inp file normally contains the most ideal parameter settings for a given site. However, there may be times when the user wants to examine the effects of modifying certain parameter settings. Extreme caution should be used when changing any parameter in the static Calmet.inp file, as it may cause unrealistic results. Each parameter in the Calmet.inp is documented in detail in the CALMET user's guide (Scire et. al. 2000b). The following discussion highlights those parameters that are generally considered the most important relative to affecting the gridded fields. Generally, these parameters are from the section "INPUT GROUP 5 – Wind Field Options and Parameters," and control the three-dimensional gridded wind field that is derived from surface and upper-air observations.

CALMET Parameter	**Allowed Values**	**Recommended Value**	**Description**
IWFCOD	0 (No) or 1 (Yes)	1 (Yes)	Wind Field Model Option—This parameter allows the CALMET model to be run as a diagnostic model and include the effects of terrain, or only perform objective analysis on the observations without regards to the underlying terrain. It is recommend the diagnostic model be used (IWFCOD = 1). The objective analysis might be considered if there are a high number of meteorological stations in an otherwise relatively flat domain.
IKINE	0 (No) or 1 (Yes)	0 (No)	Compute Kinematic effects—This parameter is intended to provide a characterization of terrain effects on vertical wind speed. CALMET then adjusts the horizontal wind speed components for mass continuity. This parameter has been found to cause unrealistic results in the horizontal wind field and so use of this parameter is discouraged.
IFRADJ	0 (No) or 1 (Yes)	1 (Yes)	Compute Froude number adjustment—This parameter is used to calculate the effects of terrain blocking on the horizontal wind field. A Froude number is calculated at each grid node in CALMET and if the value is below a certain criteria, the flow is considered blocked and is modified tangent to the terrain. The use of this parameter is recommended, particularly for domains in complex terrain.
ISLOPE	0 (No) or 1 (Yes)	1 (Yes)	Compute slope flow effects—This parameter calculates the wind flows created by sloping terrain, such as cold-air drainage flows. The use of this parameter is recommended, particularly for domains in complex terrain under calm, stable conditions.
IEXTRP	1 (-1) or 2 (-2) or 3 (-3) or 4 (-4)	-2 or -4	Extrapolate surface wind observations to upper levels—Surface winds can be extrapolated into the vertical to augment upper-air data when creating the three-dimensional gridded wind field in CALMET. This parameter allows the user to select the method of extrapolation: 1 (no extrapolation), 2 (power-law), 3 (user-factors), 4 (similarity theory). Out of the four methods, the power-law or similarity theory methods of extrapolation are recommended. Only experienced users should employ the user factors, as it requires a thorough understanding of the site to implement correctly. The negative sign preceding the value of IEXTRP has special meaning—it directs CALMET to ignore the first-level winds in the upper-air data when creating the surface layer wind field (i.e., use surface station data only).
ICALM	0 (No) or 1 (Yes)	0 (No)	Extrapolate surface winds even if calm— Under calm conditions, surface winds may not provide a realistic representation of the winds aloft, and extrapolation of the

CALMET Parameter	Allowed Values	Recommended Value	Description
			surface winds may not be desirable. It is recommended that the surface winds not be extrapolated under calm conditions.
RMAX1 RMAX2	User-specified (km)	RMAX1 (50 to 100 km) RMAX2 (100 to 500 km)	Maximum radius of influence over land—This parameter is the radius of influence at the surface (RMAX1) and aloft (RMAX2) that observed winds have on derived winds at each grid node. CALMET uses a $1/r^2$ interpolation procedure when merging observations with the derived flow. These radii control the maximum extent to include observations at each node.
LVARY	T (True) or F (False)	F	Use varying radius of influence—This parameter increases the radius of influence parameters (RMAX1 and RMAX2) until an observation station is located and can be used in the $1/r^2$ interpolation procedure. This parameter in not recommended in general, especially in areas with complex terrain where the slope flow formulation should be relied on to estimate the wind filed.
R1 R2	User-specified (km)	R1 (0.1 to 5 km) R2 (0.1 to 10 km)	Radius for Equal Weighting of Observations with Node-derived Values—This parameter controls the radius in which observations are weighted equally with node-derived values at the surface (R1) and aloft (R2). For complex terrain with calm conditions, these radii should be set very small (0.1). For less complex terrain, or normal-to-high wind conditions, setting these to 1/20 of the domain width (i.e., 5 km for a 100-km domain) is recommended.
IOBR	0 (No) or 1 (Yes)	0 or 1	Use O'Brien procedure to adjust vertical velocity—This adjustment is used during the final step in the wind field formulation in CALMET and sets the vertical wind velocities to zero at the top layer of the model domain. This generally has the effect of dampening the vertical velocities at all layers of the model. Using this adjustment is recommended unless there is a known reason that vertical velocities should not be zero in the top layer of the domain (e.g., land-sea breeze present).
TERRAD	User-specified (km)	1 to 20 km	Radius of influence of terrain features—This value determines the relative impact terrain will have on the wind field. Under calm conditions and complex terrain, this parameter should be large ($^1/_5$ of the domain width). If the terrain is not expected to have an effect, or modeling the terrain effects is not desirable, this value can be set lower.

A.2.3 Calpuff.inp

The CALPUFF input file (Calpuff.inp) is located in the "StaticData" directory for each site within DUSTRAN and is the template input file for CALPUFF. Its contents are merged with select-user input from the DUSTRAN interface for each simulation. Many of the parameters contained within Calpuff.inp control certain aspects of the plume concentration and deposition calculations, including the dispersion processes. These parameters are not available from the DUSTRAN interface and can only be edited via the static file.

The static Calpuff.inp file normally contains the most ideal parameter settings for a given site. However, there may be times when the user wants to examine the effects of modifying certain parameter settings.

Extreme caution should be used when changing any parameter in the static Calpuff.inp file, as it may cause unrealistic results. Each parameter in the Calpuff.inp is documented in detail in the CALPUFF user's guide (Scire et al. 2000a). The following discussion highlights those parameters that are generally considered the most important relative to affecting CALPUFF results. Generally, these parameters are from the section "INPUT GROUP: 2 – Technical Options," and control the plume dispersion calculations.

CALPUFF Parameter	Allowed Values	Recommended Value	Description
MGAUSS	0 1	1	Vertical distribution used in the near field—The initial plume distribution in the vertical can be uniform (0) or have a Gaussian distribution (1). A uniform distribution most likely is not representative of an actual release, so a Gaussian distribution is recommended.
MCTADJ	0 1 2 3	0	No terrain adjustment (0) is recommended. The other options are an Industrial Source Complex model type adjustment (1), a simple CALPUFF-type adjustment (2), or a partial plume path adjustment (3). The model default is the partial plume path adjustment.
MCTSG	0 1	0	Subgrid-scale complex terrain flag—this parameter should be turned off (0).
MSLUG	0 1	1	Near-field puffs modeled as elongated—The slug model, which is used to elongate puffs near the source, is recommended (1) and appears to provide more accurate results near the source than no elongation (0).
MTRANS, MTIP, MSHEAR, and MSPLIT	0 1	0	There are several puff manipulation parameters including plume rise (MTRANS), stack-tip downwash (MTIP), vertical wind shear above stack height (MSHEAR), and puff splitting (MSPLIT) that can be employed in CALPUFF.
MCHEM, MAQCHEM	0 1	0	In DUSTRAN, particle concentrations are assumed to decrease rapidly from dispersion and deposition processes. Chemical mechanism (MCHEM) and aqueous phase transformation (MAQCHEM) are considered to be negligible, and the flags should be set to zero (not used).
MWET, MDRY	0 1	1	Deposition from wet and/or dry processes can be treated. It is recommended that deposition be activated.
MDISP	1 2 3 4 5	2	Method used to compute dispersion coefficients—CALPUFF allows the user a variety of ways to calculate dispersion coefficients. The authors recommend calculating the dispersion coefficients from micrometeorological values (2). This method uses similarity theory values, such as the Monin-Obukhov length and friction velocity, to estimate the dispersion parameters. Other options include calculating the dispersion coefficients from measured values of turbulence (1), and using different forms of the Pasquill-Gifford dispersion curves (3 through 5). If the meteorological data are not sufficient to allow the use of similarity theory, then one of the Pasquill-Gifford formulations is recommended.
MPARTL	0 1	0	Partial plume penetration of elevated inversion—This parameter will allow the plume to partially penetrate a temperature inversion if the calculated stack plume rise is higher than the temperature inversion (1). This parameter is not recommended for DUSTRAN (0) because it results in additional plume dilution.

A.2.4 Calgrid.inp

The CALGRID input file (Calgrid.inp) is located in the "StaticData" directory for each site within DUSTRAN and is the template input file for CALGRID. Its contents are merged with select-user input from the DUSTRAN interface for each simulation. Many of the parameters contained within Calgrid.inp control certain aspects of the plume concentration and deposition calculations, including the dispersion processes. These parameters are not available from the DUSTRAN interface and can only be edited via the static file.

The static Calgrid.inp file normally contains the most ideal parameter settings for a given site. However, there may be times when the user wants to examine the effects of modifying certain parameter settings. Extreme caution should be used when changing any parameter in the static Calgrid.inp file, as it may cause unrealistic results. Each parameter in the Calgrid.inp is documented in detail in the CALGRID user's guide (Scire et al. 1989). One general run control parameter in CALGRID that is not set by DUSTRAN is the number of time steps per hour (NTSUBTS). The model recommended default for this parameter is 3. In high wind conditions, CALGRID may produce the following error: "array bounds exceeded." This error can be eliminated by increasing the value of NTSUBTS. It has been found that values of 20 and 30 are successful in preventing this error under high-wind-speed conditions.

A.3 DUSTRAN Initialization File (DUSTRAN.ini)

An initialization file is used to store the configuration settings for the DUSTRAN modeling system. The file, called DUSTRAN.ini, is read upon launching the DUSTRAN application. The file is formatted text and primarily holds a list of variables that store available sites and directory path information. The following is an example of the DUSTRAN.ini file followed by a description of the parameters found in the file.

A.3.1 Example DUSTRAN.ini File

```
[DUSTRAN]
Site_List= Yakima
GIS_exe=C:\Program Files\ArcGIS\Bin\MapWindow.exe
Install_dir=C:\DUSTRAN
GIS_dir=C:\DUSTRAN\Yakima\GISData
First_Run=False
InterfaceMode=Multiple
SystemName=DUSTRAN
AgDispName=SPRAYTRAN
Read_Me=False
Current_site=Yakima
APR_dir=
[Yakima]
CharCategories=Soil,Vegetation
Soil=srztext
SoilSel=yes
Vegetation=owe14d
VegetationSel=yes
Srztext=C:\DUSTRAN\Yakima\TerData\srztext.csv
Owe14d=C:\DUSTRAN\Yakima\TerData\owe14d.csv
```

A.3.2 DUSTRAN.ini File and Parameter Description

Parameter Name	Parameter Description
Site_List	Comma-separated string containing the names of the sites currently registered with the modeling system.
GIS_exe	Directory path of the MapWindow executable.
Install_dir	Directory path in which the DUSTRAN modeling system has been installed.
GIS_dir	Directory path of the folder in which the .mwprj file for the current site is stored.
First_Run	Flag used to determine if the modeling system has been executed since being installed. If this is the first startup of the modeling system, the read me file will be displayed automatically.
InterfaceMode	Flag used to determine if the modeling system will be able to access multiple sites or only a single site.
SystemName	Name of the modeling system. This is the name that will be displayed on the toolbar button and the main interface form.
AgDispName	Flag used to determine if the modeling system is to be used to make DUSTRAN or SPRAYTRAN modeling runs.
Read_Me	Flag used to determine if the read me file is displayed automatically upon the start of the modeling system.
Current_site	Name of the last site opened in the modeling system.
APR_dir	Directory path of the folder used to store Avenue script files. (Currently not used)
CharCategories	Data characteristic categories that have been entered for the simulation sites. By default, each site will have the Soil and Vegetation categories present. Other categories can be entered for a site using the appropriate utility in the DUSTRAN interface.
Soil	Comma-separated string containing the file names of the soil characteristic files entered for the site. File names are listed in order of priority.
SoilSel	Comma-separated string containing the flags denoting whether a soil characteristic file is to be used in the simulation for the site.
Vegetation	Comma-separated string containing the file names of the vegetation characteristic files entered for the site. File names are listed in order of priority.
VegetationSel	Comma-separated string containing the flags denoting whether a vegetation characteristic file is to be used in the simulation for the site.
Srztext	Full file path to the location of the soil characteristic file for the site.
Owe14d	Full file path to the location of the vegetation characteristic file for the site.

Pacific Northwest
NATIONAL LABORATORY

U.S. DEPARTMENT OF
ENERGY

Proudly Operated by **Battelle** *Since 1965*

902 Battelle Boulevard
P.O. Box 999
Richland, WA 99352
1-888-375-PNNL (7665)