Chapter 3   ***Graphical User Interface***

In this chapter of the user's guide, the following topics are discussed after the Quick Start description:

- a general description of the graphical user interface,

- the types of model inputs requested by the user interface,

- a list of possible data options,

- the range of acceptable values for each of the possible data options,

- moving around in the user interface,

- creating an option file, and

- running and viewing a model scenario.

QUICK START

The process of setting up an option file that contains a scenario that you are interested in modeling, and running and viewing the modeling results, is quite simple with the graphical user interface. Each of the steps involved is discussed in detail in this chapter. In summary, the steps involved in setting up, running, and viewing a modeling scenario are as follows:

1. With the left button on your mouse, double click on the Graphical User Interface icon created when the National Nonroad Emissions Model was installed on your computer. (For more information, see General Description below .)

2. When you first run the interface, the modeling parameters are the ones contained in the default option file *TEMPLATE.OPT*. Define the parameters that you are interested in modeling by using the left button on the mouse to click on **Scenario** on the main menu. A submenu will appear that contains the various options that you can specify. These options include the name of the output file and report, fuel

specifications, the period of time and geographical region over which you want to estimate emissions, and the source categories that you want to model. (For more information, see the section below on Creating an Option File.)

3. Once you have defined your modeling scenario, you can save the file by clicking on **Files** on the main menu and **Save** (or **Save As**) on the submenu. (For more information, see the section below on Creating an Option File.)

4. To submit your option file to the core model to run, click on **Model** on the main menu and **Run** on the submenu. Once the run has been completed, a message will appear in a DOS screen notifying you that the run was successful. (For more information, see the section below on Running and Viewing a Modeling Scenario.)

5. To view the results of this model run, click on **Model** on the main menu and **Reports** on the submenu. This will move you to the reporting utility. (For more information, see the section below on Running and Viewing a Modeling Scenario.)

6. To exit the interface, use the left button on the mouse to click on **Files** on the main menu and **Quit** on the submenu. (For more information, see the section below Exiting the Graphical User Interface.)

GENERAL DESCRIPTION

The interface provides you with an easy method for designing the nonroad emissions simulations that you are interested in viewing. In addition, the interface provides you with an easy means for accessing the reporting utility to view the results of your modeling run. With simple Windows-type menus and pull-down screens, you can quickly set up, execute, and view a modeling scenario without any computer programming experience.

> **Note**: In this chapter of the user's guide, information is provided to assist you in setting up an emissions scenario to model using the default data provided in the model. If you are interested in modifying the data files to reflect alternatives to the default data, please see Chapter 6 on Advanced Topics.

When the NONROAD emissions model is installed on your computer, an icon for each component of the model is created. While this allows you to access each component of the model individually, through the interface you can operate all of the functions of the National Nonroad Emissions Model -- from the creation of a scenario, to the running of the core model, to the development of reports -- without exiting one program and executing another.

MOVING AROUND IN THE GRAPHICAL USER INTERFACE

To begin, use the left button on your mouse to double click on the Graphical User Interface icon (Figure 3-1).

Note: If your computer operates with Windows 95, the icon can be found in a program group called Nonroad accessed from the Start menu. If you are operating with Windows 3.1, the icon can be found through Program Manager.

Figure 3-1: Graphical User Interface icon.

 When launched, the main menu screen of the interface appears (Figure 3-2). From this screen you can access five menus that make up the graphical interface module, as listed at the top of the window in the menu bar:
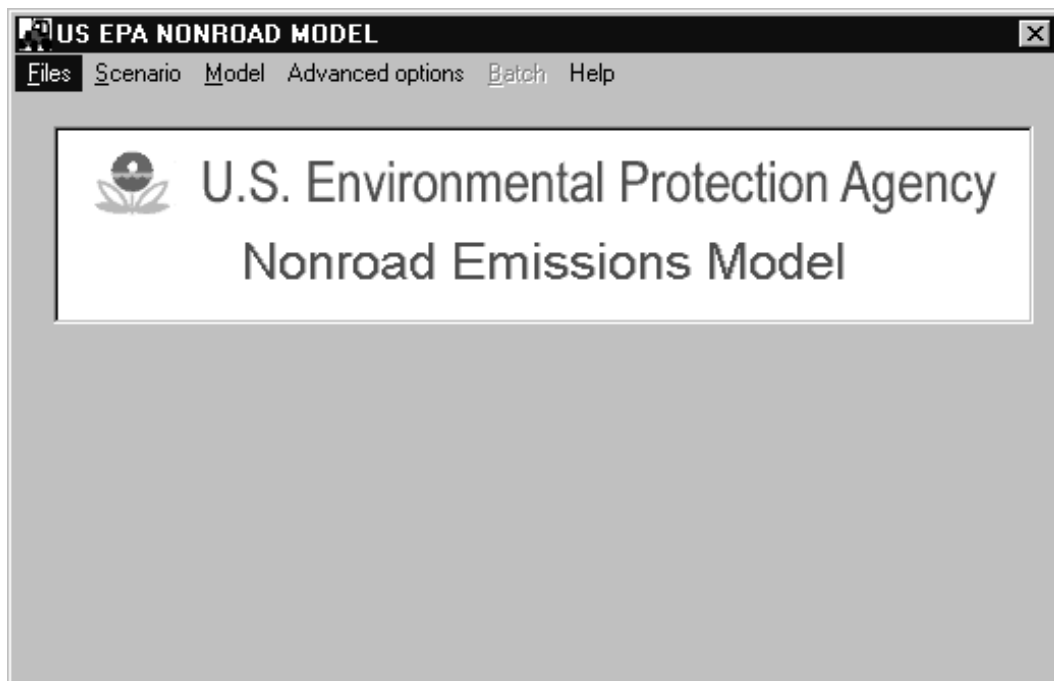
Figure 3-2: Initial graphical user interface screen.

- **Files** - In this menu, you can open an existing option file, save a modified option file with the old or a new name, or exit the interface.

- **Scenario** - In this menu, you can define the various parameters that you want in your modeling run.

- **Model** - In this menu, you can run the core model and access the reporting utility.

- **Advanced Options** - In this menu, you can select your own data files to replace the default files provided with the model.

- **Help -** In this menu, general information is provided on the version and status of the model.

Choosing Menu Options

The interface is a fully operational Windows program. The items on the menu bar, Files, Scenario, Model, Advanced Options, and Help, can be selected in one of three ways: either by using the left button on the mouse to click on the menu choice, by using the arrow keys to move left or right, or by typing the underlined letter in the menu choice while holding down the Alt key. An underlined character in each of the menu items is not fully operational at this time.

Note: To "deselect" any menu or submenu item, highlight another menu selection, or press **<ESC>**. To back up one level in the menu tree, press **<ESC>**.

Once selected, the menu will display a list of submenu items. The following shows the list of submenu items for each of the five menu choices:

Files	Open
	Save
	Save As
	Exit
Scenario	Output File
	Options
	Period
	Region
	Sources
Model	Run
	Reports
Advanced Options	Geographical Allocation
	Temporal Allocation
	Growth
	Equipment Population

                                    Phase In
                                    Emission Factors
                                    Deterioration Factors

            Help   Help
                                    About
                                    Current

Similar to the main menu, a submenu item can be selected by either using the left button on the mouse to click the highlighted item, by pressing **<Enter>** when the item is highlighted, or by simply typing the underlined letter of the item (an underlined character in each of the submenu is not fully operational at this time). For example, to define the region that you are interested in modeling you would follow these steps:

1. With the mouse, highlight **Scenario** on the menu bar and click the left mouse button. Alternatively, you can type **<Alt-S>** to display the scenario submenu.

2. Drag the mouse down to highlight **Region** and click the left mouse button. Alternatively, type **<G>** to access the report.

After selecting an option, you can move around the input form by using **<Tab>**, the arrow keys or pointing and clicking with the mouse. When an input error occurs, an appropriate message (dialog box) is displayed. If a dialog box appears, use the left button on the mouse to click on **OK** or press **<Enter>**, make the corrections needed, and continue.

CREATING AN OPTION FILE

The first step involved in operating the National Nonroad Emissions Model is to setup an option file which defines the parameters that you are interested in modeling. Once this file has been set up, it can be submitted directly to the core model from the interface, and then the model output can be imported into the reporting utility after it has been run.

An option file (**.opt*) contains all of the parameters that specify your modeling run. To more easily explain the process of setting up an option file, the following example is provided:

> *You have been given an assignment to estimate the total emissions from all gasoline powered lawn and garden equipment with four-stroke engines in Travis County (Austin), Texas. You are interested in estimating these emissions for the summertime of 1998.*

Throughout this chapter, we will refer to the above example.

Opening and Naming an Option File, and Naming the Output File

When you first run the interface, the default option file *TEMPLATE.OPT* is automatically opened. You can modify this file to define a modeling scenario of interest. Alternatively, if you wish to use or modify any other option file (**.opt*) that you have previously created, follow these steps:

1. From the Main Menu, use the left mouse button to click on **Files**, or press **<Alt-F>**.

2. A pull-down menu appears identifying four choices. Click the left mouse button on **Open**, or use the up/down arrow keys to highlight **Open** and press **<Enter>**. Alternatively, press **<O>**.

3. An open file dialog box appears that requests you to select a new option file. Highlight the desired file and either double click on the file name with the left button of the mouse, click on **OK**, or press **<Enter>**. This loads the option file into the interface.

Before beginning to modify the scenario options in the option file, you should save the new file with a different name to prevent yourself from accidentally overwriting data in a previously created option file or *TEMPLATE.OPT*. To do this, select **Files** from the main menu and **Save As** from the submenu.

The first step in defining your modeling run is to give a name to the output file that the model will generate. To do this, use the left mouse button to click on **Scenario** (or press **<Alt-S>**) and then **Output File** (or press **<P>**). An open file dialog box appears that allows you to name the output file. This will

> **Note**: The name of the output file must be no longer than eight characters long as a result of DOS conventions. The extension *.OUT* will automatically be added to this filename.

default to the same name as the option file but with the extension *.OUT*. Alternatively, you can use some other name by typing the name in the field below File Name and clicking on **OK**. For our example, you would name the output file *TRAVIS.OUT*.

Defining a Report Title

The interface allows you to easily title your model run. This title will be used on any reports you might generate for this particular scenario.

To provide a more descriptive name for your model run, follow these steps:

> **Note**: The two title lines of your modeling run must be no longer than 80 characters each. In addition, the titles should not include single or double quotation marks (i.e., " or '), or periods.

1. Use the left mouse button to click on **Scenario** (or press **<Alt-S>**) and then **Options** (or press **<O>**). The Options screen will appear (Figure 3-3). When the Options screen first appears, it will contain the title and any options previously defined for this option file. To change the title, simply click in the field below Title 1 and type in a descriptive name for your scenario. If you want to add a subtitle, you can do so in a similar fashion in the field below Title 2.

2. When you have provided a title for your report, click on **Done** or **<Alt-D>** and then press **<D>** to return to the main menu.

Defining Fuel Specifications, and Ambient Temperatures

Nonroad emissions result from the combustion, evaporation, and spillage of fuel in equipment and vehicles designed primarily for off-road use. The emissions that come from nonroad equipment and vehicles are dependent, in part, on the composition of the fuel used. Similarly, the ambient temperature will also affect the emissions from nonroad sources. As a result, it is important to define the composition of the fuel and the ambient temperature for the period of time that you are interested in modeling.

> **Note**: The minimum temperature that you define for your modeling scenario is the average low for the time period selected. For example, when modeling a typical summer day, the minimum temperature should be the average low temperature in Fahrenheit for the summer season. Moreover, when modeling an entire year, the minimum temperature should be the average daily low for the year and not the lowest temperature ever recorded. Other modeling parameters, such as RVP, sulfur, and oxygen content, depend on local fuel characteristics. Data for an area should be available from the state or local air agency.

Figure 3-3: Options screen.

To do this, follow these steps:

1. Once you have opened an option file, use the left mouse button to click on **Scenario**, or press **<Alt-S>**. (For our example, the file *TRAVIS.OPT* should be loaded. If not, open that file using the steps outlined above.)

2. Under **Scenario,** click on **Options** (or press **<O>**). The Options screen will appear.

3. For our example, we need to identify the gasoline specifications and ambient temperatures that will be in effect in Travis County during the summer of 1998. To do this, use the left button on the mouse to click on the appropriate field and type in any required changes to the scenario options. Next, you must identify whether the

modeling region is at high or low altitude[1]. High should be specified where the average altitude is greater than 5000 feet; this is consistent with the MOBILE program. For Travis County, click on the button for **Low**. When you have completely defined fuel specifications, and ambient temperatures, click on **Done** or press **<Alt-D>** and then **<D>** to return to the main menu.

Note: The acceptable range of values for each of the fuel specification and ambient temperature options are as follows:

Fuel RVP	6 to 14
Oxygen (wt.%)	0 to 5
Fuel Sulfur (wt %)	0 to 0.5
Temperature (°F)	-40 to 120

Defining the Modeling Period

The modeling period is simply the period of time for which you want to estimate emissions. To define the modeling period, follow these steps:

1. After you have defined the parameters in the **Options** submenu, use the left mouse button to click on **Scenario** (or press **<Alt-S>**) and then **Period** (or press **<E>**). The Period screen will appear (Figure 3-4). The interface makes it easy to define the modeling period through a series of select buttons. The following options are available to you:
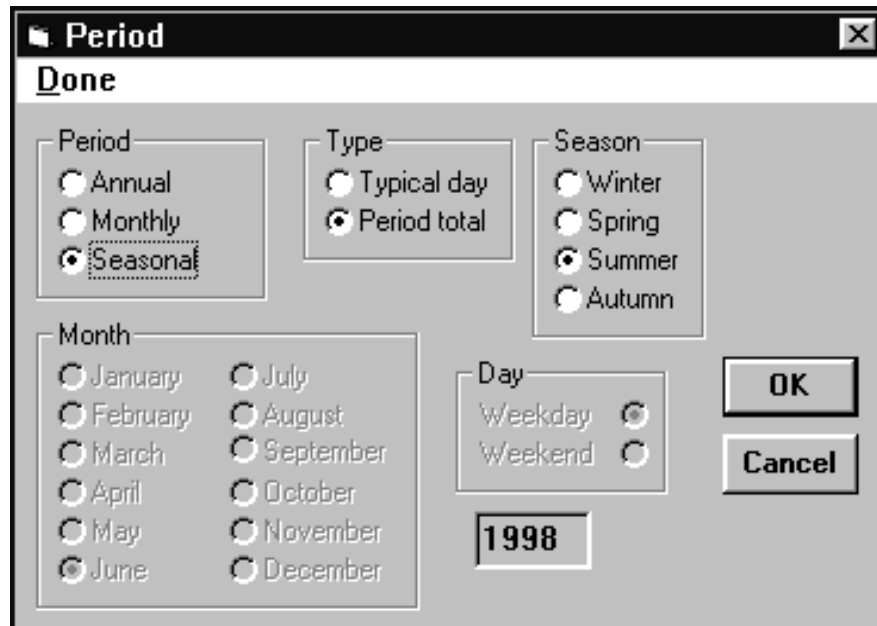

Figure 3-4: Period screen.

---

[1] Altitude adjustment is not in the draft version of the model; it will be incorporated into future model updates. Hence, this selection is currently disabled in the user interface.

- **Period** - You can define the modeling period as annual, monthly, or seasonal.

- **Month** - If you select the modeling period to be monthly, the list of months will become active and you can select the appropriate month of the year.

- **Season** - If you select the modeling period to be seasonal, the list of seasons will become active and you can select the appropriate season of the year.

- **Type** - For any period that you select, you can estimate the emissions as the total for a 24-hour period (typical day) or for the total period.

- **Day** - If you select the type to be a typical day, days will become active and you can select a weekday or weekend day.

- **Year** - You can define any year between 1970 and 2050 that you are interested in modeling.

  Note: While the model allows you to define any model year from 1970 to 2050, it actually forecasts and backcasts based on 1996 data. For this reason, its estimates become less accurate as you move further into the past or the future.

2. Since we are interested in estimating the total emissions during the summertime of 1998 in our example, use the left button on the mouse to click on **Seasonal** (under Period), **Summer** (under Season), **Period Total** (under Type), and type in 1998 in the year field.

3. When you have finished defining the period, click on **Done** or **OK**, or press **<Enter>** to return to the main menu.

Defining the Modeling Region

The interface makes it relatively easy to define the geographical region -- from one county to multiple counties to the entire state -- that you are interested in modeling. To do this, follow these steps:

1. Use the left mouse button to click on **Scenario** (or press **<Alt-S>**) and then **Region** (or press **<R>**). The Region screen will appear (Figure 3-5).

2. The first step in defining the modeling region is to remove any previous regions that have been selected in a previous scenario. To do this,

   Note: An additional option, US TOTAL, which estimates national totals, is available. Refer to Chapter 4, Core Model, Modifying/REGION/.

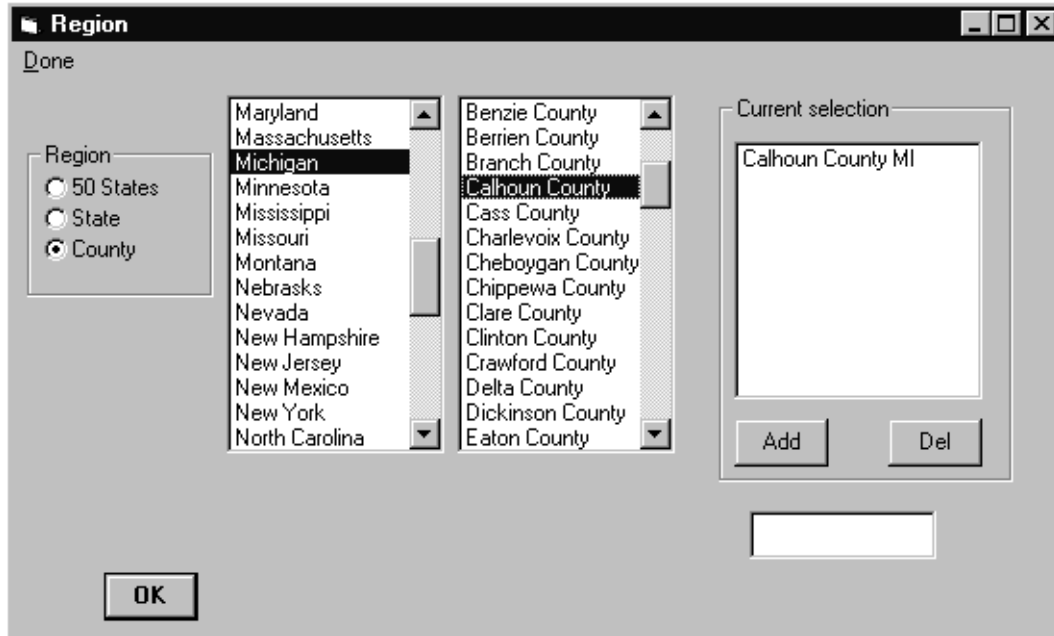highlight any regions in the field under Current selection and click on **Del**.

Figure 3-5: Region screen.

3. To add a new region, several options are available to you. These options and the methods for selecting them are as follows:

    •   **50 States** - This option allows you to select all states. If this option is selected, you will obtain the state totals for all 50 states individually. To select 50 States, use the left button on the mouse to click on **50 States** (under region) and then click on **Add**. All 50 states will appear in the current selection field.

    •   **State** - This option allows you to select any one, or multiple, of the 50 states. When a state is selected, the model will compute total emissions for the entire state, but will not allocate emissions among counties in the state. To select an individual state, use the left button on the mouse to click on State (under region). To find the state you want, use the scroll bar to move through the list of states. Once

> **Note**: To create a modeling region that is larger than a single state or county, continue to add new regions to the list of currently selected regions by repeating the process for selecting regions. For example, if a metropolitan area covered four counties, simply select the first county and then each additional county one at a time.

found, highlight the state and click on **Add**. The state will appear in the current selection field.

- **County** - This option allows you to select any county within any selected state. To select an individual county, use the left button on the mouse to click on County (under region). Next, select the state you want by using the scroll bar to move through the list of states. When you highlight the state you want, a list of counties for that state will appear. Again, use the scroll bar to move through the list of counties until you find the one that you want. Once found, highlight the county and click on **Add**.

> **Note**: The model only allows you to pick counties from a single state. Emissions from counties in different states must be computed in separate model runs.

> **Note**: When you select county as the region, and highlight the state and click on **Add**, the interface will change the current selection box to replace the state name (e.g., Michigan) with "All of ..." (e.g., "All of Michigan") and will then compute emission estimates for all of the counties chosen in the named state.

4. For our example scenario, use the left button on the mouse to click on County (under Region) and then the scroll bars to locate Texas from the list of states. Once Texas is highlighted, use the scroll bars to locate Travis from the list of counties. Highlight Travis and click on **Add**.

> **Note**: The interface also allows you to define a sub-county region. However, this selection requires that you provide a significant amount of data beyond that furnished with the model. For information on how to input this additional data, please see Chapter 6, Advanced Topics.

5. When you have finished defining the region, click on **Done** to return to the main menu.

Defining Source Categories

The National Nonroad Emissions Model allows you to estimate emissions from many different types of equipment and vehicles. To define the source category that you want to model, follow these steps:

1. Use the left mouse button to click on **Scenario** (or press **<Alt-S>**) and then **Sources** (or press **<S>**). The Sources screen will appear (Figure 3-6).

> **Note**: If you are interested in estimating emissions from more than one segment or fuel type, add the first segment/fuel type and then continue adding segment/fuel types as desired.

2. Similar to defining a region, you must first remove any source segments or equipment that you do not want to model. To do
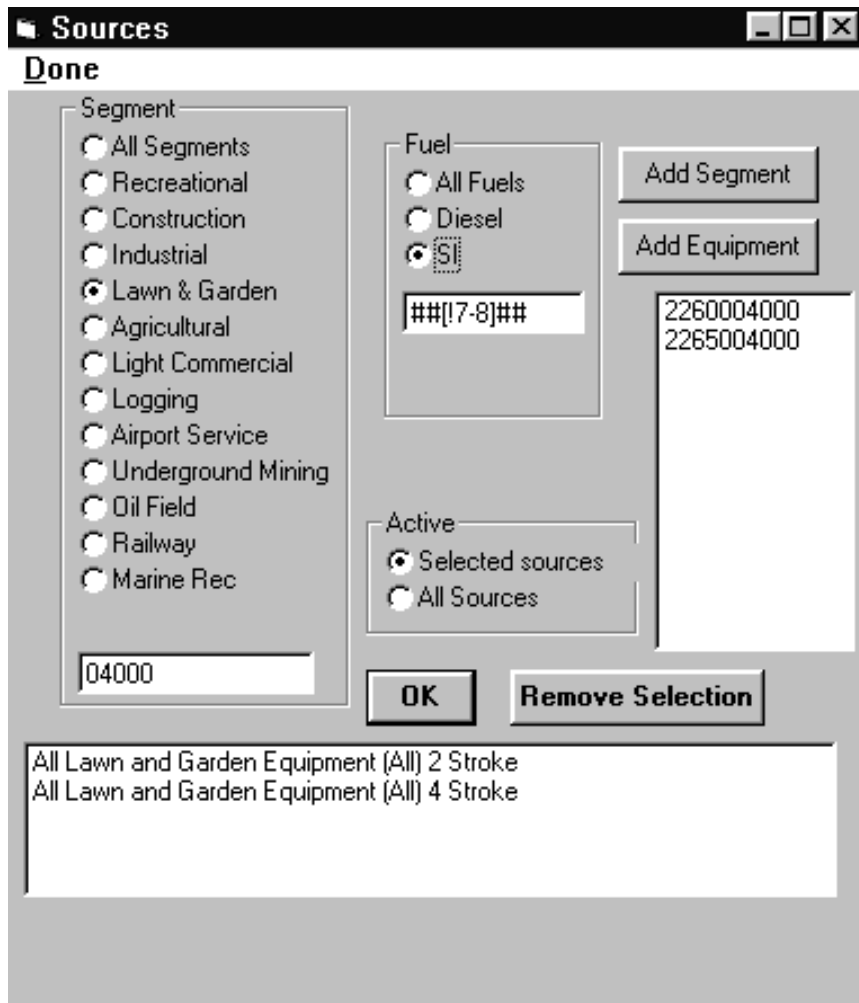

Figure 3-6: Sources screen.

that, highlight the segments and/or equipment in the field at the bottom of the Sources screen and click on **Remove Selection**. To remove multiple sources, hold **<Shift>** or **<Control>** while highlighting, then click on **Remove Selection**.

3. To define a source category for modeling, first select the category segment. If you were interested in estimating emissions for all source categories, click on **All Segments**. If you are only interested in one specific segment, click on the appropriate segment. For our example, we are interested in all lawn and garden equipment. To select this category, use the left button on the mouse to click on **Lawn & Garden**. The last five digits of the SCC are displayed in the field under the segment list.

4. Next, select the fuel type. If you were interested in all fuel types, click on **All Fuels**. If you are interested in all diesel-powered engines, click on **Diesel**. Since we are interested in gasoline-powered lawn and garden equipment with a 4-stroke engine in our example, use the left button on the mouse to click on **SI** (spark ignited). The first five digits of the SCC are displayed in the field under the field list. (A "#" indicates a single digit place holder and a "!" means a logical NOT).

5. Once you have identified the segment and fuel type, click on **Add Segment**. A brief description of the selection should appear in the window at the bottom of the Sources screen.

6. To define only specific equipment within a source segment, identify the segment and fuel type as described above and use the left button on the mouse to click on **Add Equipment**. All of the equipment that falls under the specified segment/fuel type will appear in the window at the bottom of the Sources screen. Hold **<Shift>** or **<Control>** and highlight the equipment that you do not want and click on **Remove Selection** until the equipment you desire remains.

Note: You may notice that as you click on a segment and fuel type, a series of numbers appear in a blank field at the bottom of the segment and fuel boxes. Each source category in the National Nonroad Emissions Model is uniquely identified with a source category code (SCC). The SCC is a ten-digit number: for most categories, the first two represent industry (mobile = 22), the next two represent engine type, the next three represent segment type, and the last three represent equipment type. Refer to Appendix B for a complete list of NONROAD SCCs.

7. When you have finished defining sources, click on **Done** to return to the main menu.

Saving an Option File

Once you have created an option file by defining all of the parameters that you want, you should save this file. To do this, follow these steps:

1. From the Main Menu, use the left mouse button to click on **Files**, or press **<Alt-F>**.

2. Click the left mouse button on **Save**, or use the up/down arrow keys to highlight **Save** and press **<Enter>**. Alternatively, press **<S>**.

RUNNING AND VIEWING A MODELING SCENARIO

Once you have set up an option file, the interface allows you to easily submit the option file to the core model to run. Similarly, the interface allows you to access the reporting utility to view the results of your run and prepare printed reports.

To submit an option file to the core model, follow these steps:

1. From the Main Menu, use the left mouse button to click on **Files**, or press **<Alt-F>**.

2. Click the left mouse button on **Open**, or use the up/down arrow keys to highlight **Open** and press **<Enter>**. Alternatively, press **<O>**.

3. An open file dialog box appears that requests you to select an option file. Select the option file that you want to model and either double click on the file name with the left button of the mouse, click on **OK**, or press **<Enter>**. This loads the option file into the interface. If the option file is already loaded, you can skip steps 1 through 3.

4. Use the left button on the mouse to click on **Model**, or press **<Alt-M>**. A submenu screen appears (Figure 3-7) with two selections: Run (with the option file that you have selected) and Reports.

5. To run *TRAVIS.OPT*, use the left button on the mouse to click on Run. A DOS window will appear. After the run has been completed, a message will appear notifying you that the run was successful.

    If the DOS window disappears after the core model completes and you wish to view the DOS screen, then you need to change the default 'close on exit' in your Windows application. If you are running Windows 3.1, this can be accomplished by using the PIF Editor (from Windows Main group) to edit the NONROAD.PIF file and disable the 'Close Window on Exit' box. If you are running Windows 95, then right click on

the *nonroad.exe* filename, select Properties, select the Program tab, and disable the 'Close on exit' box. You will then need to manually close the DOS window when the core model completes.

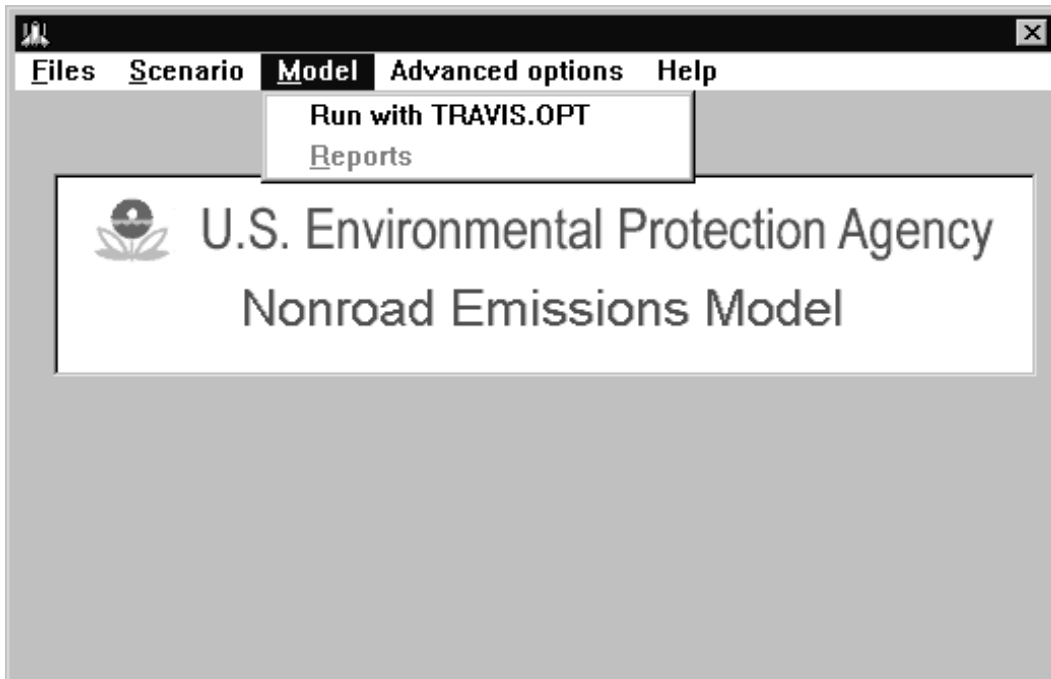6. To return to the interface, press **<Enter>**.

Figure 3-7: Model submenu screen.

7. If you are interested in modeling several scenarios prior to viewing and printing a report, repeat the above steps for each of your option files.

To access the reporting utility, follow these steps:

1. From the Main Menu, use the left mouse button to click on **Model**, or press **<Alt-M>**.

2. Click the left mouse button on **Report**, or use the up/down arrow keys to highlight **Report** and press **<Enter>**. Alternatively, press **<R>**. Chapter 5 describes the Reporting Utility.

Note: The core model runs in DOS under a windows environment which allows you to multitask activities while the model is running. However, multitasking may lengthen the amount of time it takes for the model to run, particularly if you are using a 486 computer with less than 16 megabytes of RAM. If this describes your computer, you should consider closing all other applications prior to running the model.

ENVIRON

ADVANCED OPTIONS

The Advanced Options menu allows you to specify your own data files to replace the default files provided with the NONROAD model. Each submenu selection allows you to specify a different file to use. In some cases, more than one file may be applicable to a selection. The submenu choices are:

> Geographic Allocation
> Temporal Allocation
> Growth
> Equipment Population
> Phase In
> Emission Factors
> Deterioration Factors

The default Geographic Allocation and Equipment Population files have been provided by state to reduce the size of the files. These file names will need to be changed if your modeling region is outside the state you specified during the NONROAD model installation.

EXAMPLES OF CHANGING GEOGRAPHIC ALLOCATION

The Geographic Allocation selection allows you to identify the allocation look-up table (default is "*allocate.xrf*") and allocation data files that you want to use. To change the look-up table or allocation data files, follow these steps:

1. Once you have opened an option file using the interface, use the left mouse button to click on **Advanced Topics**, or press **<Alt-A>**. Next, use the left mouse button to click on **Geographic Allocation** (or press **<G>**). The Geographic Allocations screen will appear (Figure 3-8).
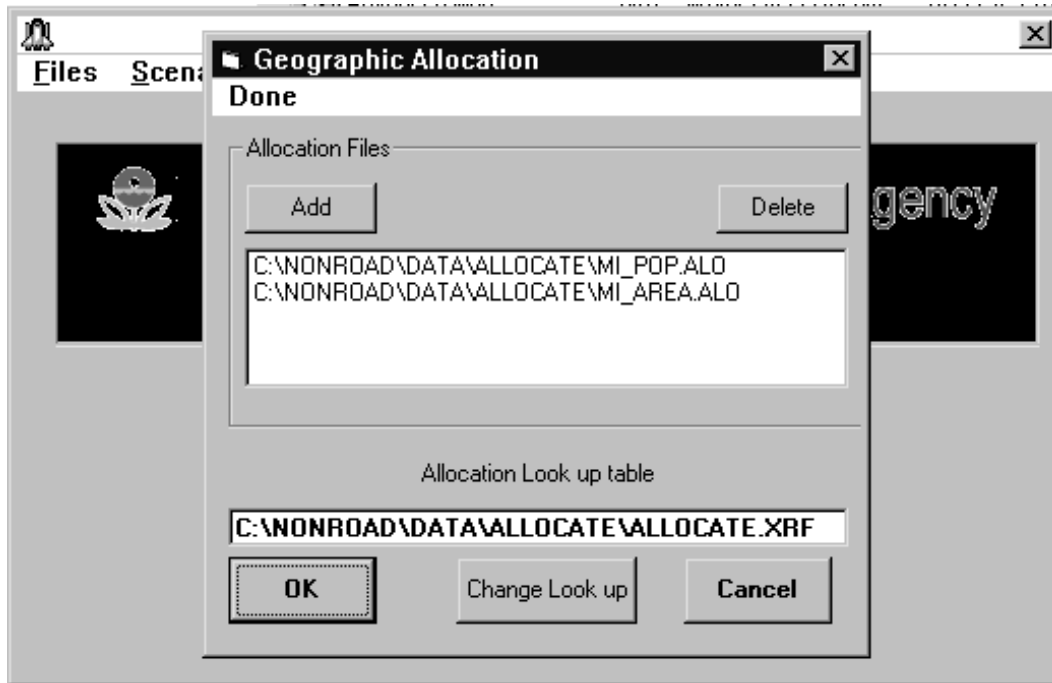
Figure 3-8: Geographic Allocation screen.

2. The first step in defining the geographic allocation data files is to remove any previous data files. To do this, highlight the current files and click on **Del**.

3. To add an allocation data file, use the left button on the mouse to click on **Add**. An open file dialog box appears that requests you to select a new allocation file. Locate the directory where the allocation files are saved (when installed: \nonroad\data\alloc), highlight the data file that you want, and either double click on the file name with the left button of the mouse, click on **OK**, or press **<ENTER>**.

4. To change the default allocation look-up table, use the left button on the mouse to click on **Change Look up**. Again, an open file dialog box appears that requests you to select a new allocation file (this time with the extension *.xrf*). Locate the directory where the allocation files are saved, highlight the data file that you want, and either double

> **Note**: The allocation file contains the actual estimates of indicator values (e.g., the actual number of employees in the construction industry by county). The allocation look-up table is an assignment of equipment type to an allocation indicator (e.g., construction equipment assigned to employees in the construction industry).

click on the file name with the left button of the mouse, click on **OK**, or press **<ENTER>**.

5. When you have identified the allocation files and table that you want, click on **Done** to return to the main menu.

EXITING THE GRAPHICAL USER INTERFACE

To exit the interface, follow these steps:

1. From the Main Menu, use the left mouse button to click on **Files**, or press **<Alt-F>**.

2. Click the left mouse button on **Quit**, or use the up/down arrow keys to highlight **Quit** and press **<Enter>**. Alternatively, press **<Q>**.