

Chapter 4

Core Model

In this chapter of the user's guide, the following topics are discussed:

- a general description of the core model,
- the process of modifying an option file using a text editor or DOS and
- running the core model.

QUICK START

The process of modifying an option file through a text editor or DOS and running the core model using information provided in the NONROAD model is straightforward. In summary, the steps involved in running the core model with existing data are as follows:

1. To modify an option file, either use the graphical user interface as discussed in Chapter 3 or a text editor, such as Microsoft's Notepad or WordPad, to access the *.opt file that you are interested in changing. Alternatively, you can access the file through DOS by typing "*edit (the name of your file).opt*" from the nonroad directory. (For more information, see the section below on Creating an Option File Using a Text Editor or Creating an Option File Through DOS.)
2. Once you have modified an option file to define the scenario that you are interested in modeling, you can run the scenario from the graphical user interface as discussed in Chapter 3, or from DOS by simply typing "*nonroad (the name of your file).opt*" from the nonroad directory. (For more information, see the section below on Running the Core Model.)
3. After the model has finished, a message will appear in a DOS window notifying you of the successful completion of the run. To return to the graphical user interface, or to Windows 3.1 or Windows 95, press <Enter>.

GENERAL DESCRIPTION

The core NONROAD model contains all of the algorithms used by the model for calculating emissions estimates. The core model can be operated as a stand-alone application; however, as a stand-alone application it requires some basic knowledge of the DOS operating system. If you prefer, the option file required to run the model can be easily set up through the graphical user interface and the core model run from there as well.

When the NONROAD model is installed on your computer, a directory for the NONROAD Model is created. To access the core model from the graphical user interface, follow the steps described in Chapter 3. From DOS, change to the nonroad directory by typing "*cd nonroad*" (assuming the National Nonroad Emissions Model is located on the c drive of your computer; if not, go to the appropriate drive before typing "*cd \nonroad*".)

Note: In this chapter of the user's guide, information is provided to assist you in setting up an emissions scenario and running the 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.

Note: The core model is executed with the file *NONROAD.EXE*. For more technical information on the core model, please refer to the technical support documents, or the core model code.

CREATING AN OPTION FILE USING A TEXT EDITOR

Before running the core model, you must define the scenario you are interested in modeling by creating an option file. Chapter 3 describes how to create this option file from the graphical user interface. This section describes the alternative method of creating an option file with a text editor.

In a fashion similar to that followed in Chapter 3, the following example is provided to more easily explain the process of creating and setting up an option file:

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.

This example will be referred to throughout this chapter.

To create an option file using a text editor, follow these steps:

1. Using the left button on the mouse, locate a text editor on your computer. Double click with the left button on the mouse on the text editor icon to execute the editor.
2. From the main menu of the text editor, use the left mouse button to click on **File**, or press <Alt-F>. Next, use the left button on the mouse to click on **Open**, or use the up/down arrow keys to highlight **Open** and press <Enter>.
3. An open file dialog box appears. In the field at the bottom of the dialog box identified as Files of Type:, use the left button of the mouse to click on the small arrow at the right of the field and select **All Files (*.*)**. Next, locate the directory where the NONROAD model is located. Use the mouse to highlight the input data file that you want to use to create your option file and either double click the left button on the mouse or click on **Open**.
4. Figure 4-1 shows the file *TEMPLATE.OPT* in the text editor. For our modeling example, we want to modify this file by

Note: Steps 1 through 4 in this section are for the text editors Notepad and WordPad. If you use a different text editor, the exact key and mouse strokes may differ.

Note: Windows 3.1 and Windows 95 comes with the text editor Notepad. This can be found in the Accessories program group. The use of a normal word processor such as WordPerfect or Microsoft Word is not recommended unless you are experienced in dealing with ASCII files in those applications.

Note: The option file that you create or modify to represent your modeling scenario must be a *.opt file. When the NONROAD Model is installed on your computer, a file called *TEMPLATE.OPT* will also be installed. If this is the first option file that you are creating, use *TEMPLATE.OPT*. If you have previously created an option file, you can select any *.opt file for modification.

specifying the parameters required to estimate summertime emissions in 1998 from four-stroke gasoline engines in Travis County, Texas. To prevent the accidental loss of data resulting from overwriting an existing file, first save the *TEMPLATE.OPT* file as *TRAVIS.OPT*. This is done by clicking with the left button on the mouse on **Files** and **Save As**. In the field identified as File Name, type "*TRAVIS.OPT*" and click on **SAVE**.

```

template.opt - Notepad
File Edit Search Help
Written by Nonroad interface at 9/2/97 1:52:58 PM
This is the options file for the NONROAD program.
The data are separated into "packets" based on common
information. Each packet is specified by an
identifier and a terminator (/END/). The packet keywords
can appear in mixed case and any notes or descriptions
can be placed between the data packets.

Some packets are optional and can be removed by
placing any non-blank character in the first column
of the line containing the packet identifier.

-----
This is the packet that defines some of the data files
read by the model.

The filenames can appear in any order; the keyword in the
first 19 columns identifies the file type.
The file types are:

ALLOC XREF -- SCC to allocation indicator cross reference
ACTIVITY -- activity data by equipment type
TECHNOLOGY -- technology type definitions and tech. fractions
SEASONALITY -- data used for temporal distribution

```

Figure 4-1: *TEMPLATE.OPT* file opened in a text editor.

5. For a simple modeling run using the data provided with the model, five packets of information can be modified. These five packets are the */RUNFILES/*, */OPTIONS/*, */PERIOD/*, */REGION/*, and */SOURCE CATEGORY/*. To modify any of these five packets, use the left button on the mouse to scroll through the file in the text editor and locate the packet you want to change.

Note: As discussed in more detail in Chapter 6, Advanced Topics, input data files are separated into "packets" based on common information. Each packet is specified by an identifier (e.g. */RUNFILES/* for the runfiles packet) and a terminator (*/END/*).

Modifying /RUNFILES/

In the /RUNFILES/ packet, the data files read by the model are defined. For our simple example, we are only interested in changing the name of the message file and the output data file. To do this, follow these steps:

1. Locate the /RUNFILES/ packet by using the left button on the mouse to scroll through the file *TRAVIS.OPT*. Alternatively, use the up/down arrow keys to scroll through the file.
2. Once you locate /RUNFILES/, use the left button on the mouse to click on the current message file (if you opened *TEMPLATE.OPT*, the message file would be *TEMPLATE.MSG*). Alternatively use the left/right arrow keys to move to the current message file. Type in the name that you want to call the new message file (in our example *TRAVIS.MSG*).
3. Next, use the left button on the mouse to click on the current output data file (if you opened *TEMPLATE.OPT*, the output data file would be *TEMPLATE.OUT*). Alternatively use the left/right arrow keys to move to the current output data file. Type in the name that you want to call the new output file (in our example, *TRAVIS.OUT*). In Figure 4-2, the message and output data files have been changed.

Note: The core model creates two output files. The first is a message file (*.msg) that can be read with a text editor or through DOS and identifies relevant information on the model run such as time and date of the run and any warnings which occurred during the run. The second is a data file (*.out) that can be opened and viewed in the reporting utility.

```

Travis.opt - Notepad
File Edit Search Help

ALLOC XREF -- SCC to allocation indicator cross reference
ACTIVITY -- activity data by equipment type
TECHNOLOGY -- technology type definitions and tech. fractions
SEASONALITY -- data used for temporal distribution
REGIONS -- definitions of inputs data regions
MESSAGE -- output file for including program messages
OUTPUT DATA -- data file used by the reporting utility

-----

/RUNFILES/
ALLOC XREF      : c:\nonroad\data\allocate\allocate.xrf
ACTIVITY       : c:\nonroad\data\activity\activity.dat
TECHNOLOGY     : c:\nonroad\data\tech\tech.dat
SEASONALITY    : c:\nonroad\data\season\season.dat
Regions       : c:\nonroad\data\season\season.dat
Message        : travis.msg
Output data    : travis.out
/END/

-----

This is the packet that defines the equipment population
files read by the model. There can be up to 10 files.
Typically each file represents one state. Data

```

Figure 4-2: TRAVIS.OPT runfiles.

Modifying /OPTIONS/

The options packet, /OPTIONS/, defines some of the parameters that make episode-specific emission factor adjustments. These options include fuel specifications that will be in effect during the modeling period and ambient temperatures that are typical during the same period. In addition, you can define the title that describes your modeling scenario and which will appear on your model reports.

To modify the options packet, follow these steps:

1. Locate the /OPTIONS/ packet by using the left button on the mouse to scroll through TRAVIS.OPT. Alternatively, use the up/down arrow keys to scroll through the file.
2. Once you locate /OPTIONS/, use the left button on the mouse to click on any current title that appears in the line **Title 1**. Type in a name that describes your scenario. If you are interested in adding a subtitle, type in that name on the line **Title 2**. If you do not want a title to appear on your report, leave these fields blank.

3. For the remaining information in the /OPTIONS/ packet, click on the appropriate line and change any existing information that is relevant for your run. Figure 4-3 shows the /OPTIONS/ packet with modifications for our model example.

Modifying /PERIOD/

The packet /PERIOD/ defines the period of time for which emissions are to be estimated. To modify the /PERIOD/ packet, follow these steps:

1. Locate the /PERIOD/ packet by using the left button on the mouse to scroll through *TRAVIS.OPT*. Alternatively, use the up/down arrow keys to scroll through the file.
2. Once you locate /PERIOD/, use the left button on the mouse to click on the line that you want to modify and type the information that is appropriate for your scenario. Figure 4-4 shows the /PERIOD/ packet with the changes required for our example.
3. The valid responses for the /PERIOD/ packet are as follows:

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 (%)	0 to 0.5
Temperature (°F)	-40 to 120
Altitude*	Low or High (Specify high where average altitude > 5000 ft)

*High altitude adjustments are not included in the draft model. This adjustment will be made in future model updates.

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.

Period type	Annual, seasonal, or monthly
Summation Type	Typical day or period total
Year of Episode	A four-digit year between 1970 and 2050

Season of year	Winter, spring, summer, autumn
Month of year	Complete month name
Weekday or Weekend	Weekday or Weekend

Note: The season is ignored if the period is not seasonal. Similarly, the month is ignored if the period is not monthly. Also, weekday/weekend is ignored if summation type is period total.

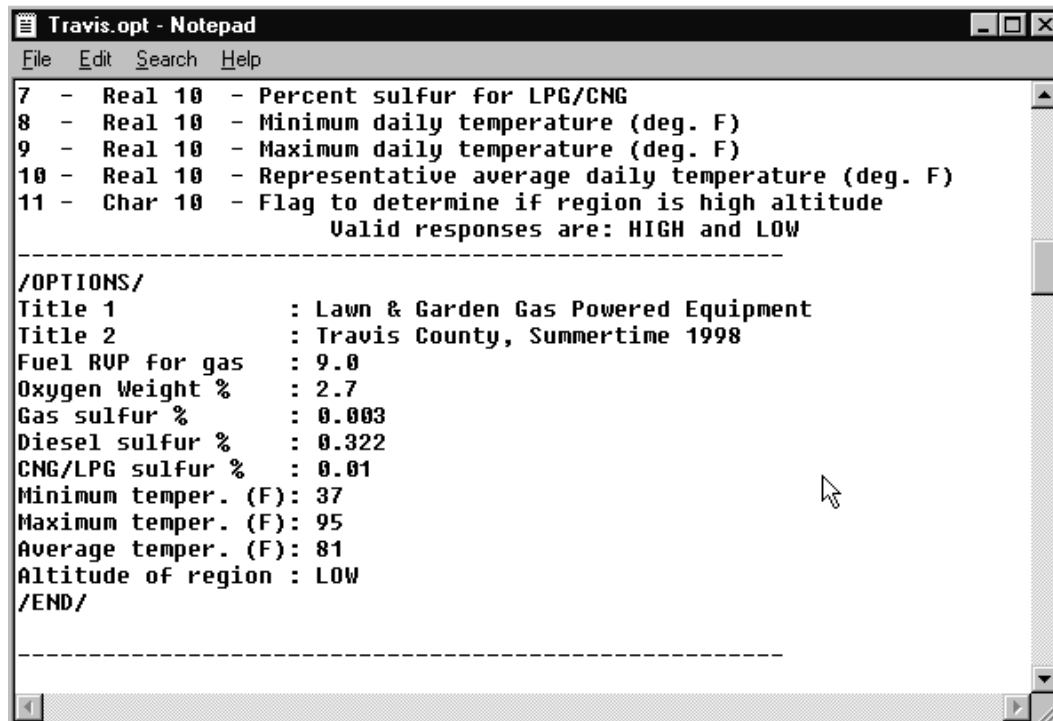


Figure 4-3: Modified/OPTIONS/ packet.


```

Travis.opt - Notepad
File Edit Search Help
records matter. The selection of certain parameters
will cause some of the records that follow to be ignored.
The order of the records is as follows:

1 - Char 10 - Period type for this simulation.
           Valid responses are: ANNUAL, SEASONAL, and MONTHLY
2 - Char 10 - Type of inventory produced.
           Valid responses are: TYPICAL DAY and PERIOD TOTAL
3 - Integer - year of episode (4 digit year)
4 - Char 10 - Month of episode (use complete name of month)
5 - Char 10 - Type of day
           Valid responses are: WEEKDAY and WEEKEND
-----
/PERIOD/
Period type      : seasonal
Summation type   : period total
Year of episode  : 1998
Season of year   : summer
Month of year    :
Weekday or weekend :
/END/

```

Figure 4-4: Modified/PERIOD/ packet.

Modifying /REGION/

The packet /REGION/ defines the geographic region that you are interested in modeling. In the National Nonroad Emissions Model, regions are defined by using two types of parameters. The first specifies the type of region and the second identifies the region codes or FIPS codes to be modeled. You can specify a global FIPS code to indicate an entire group of counties or states. For example, by entering "06000" as the FIPS code, you are specifying all of the counties in the State of California. A FIPS code of "00000" indicates all states in the United States. For our example, the FIPS code for Travis County, Texas is 48453 (48 designates the State of Texas; 453 is the code for Travis County). The first two pages of FIPS codes used in the model are listed in Appendix A; an electronic file containing all FIPS codes is contained on the NONROAD compact disc (file *fipslist.asc*) or can be downloaded from the nonroad modeling web page.

Note: The acceptable responses for the region level are US TOTAL, 50 STATE, STATE, COUNTY, AND SUBCOUNTY.

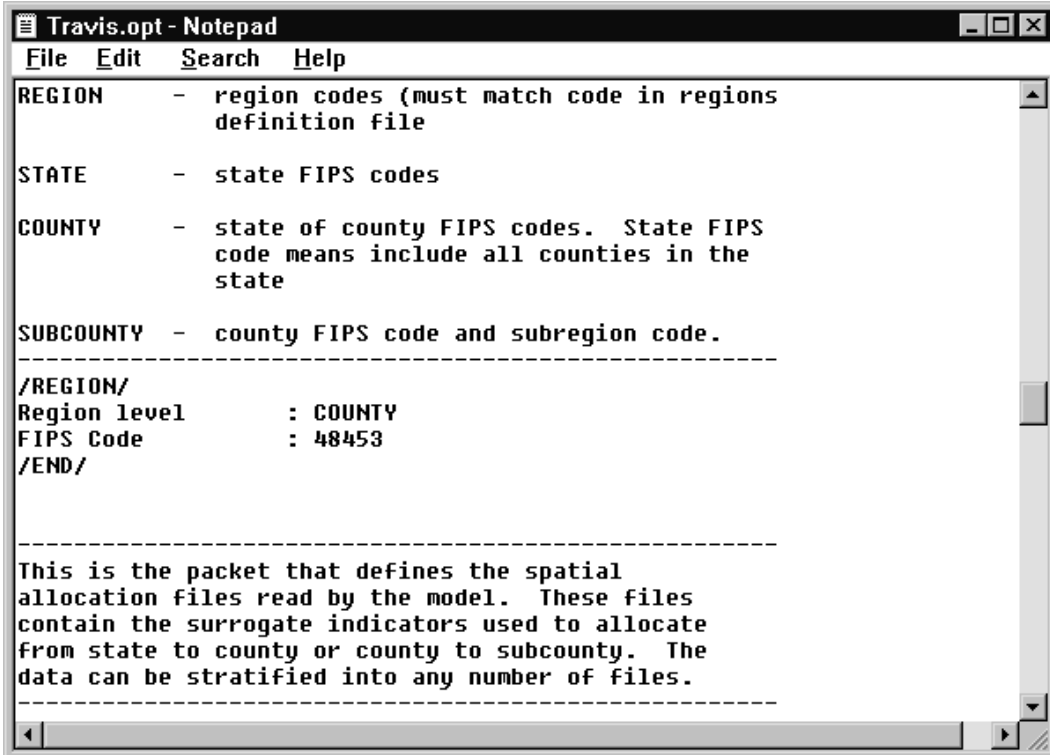
The following table shows how the model's definition of the modeling region depends on the combination of the level and FIPS code(s) specified:

Level	FIPS Code
US TOTAL	The “FIPS code” is ignored and national estimates are calculated. This run is substantially faster than 50 STATE since no state detail or allocation is required.
50 STATE	The “FIPS code” is ignored and state-level estimates are calculated for all 50 states.
STATE	If a list of specific state FIPS codes is supplied [e.g., 06000 (for California) and 48000 (for Texas)], state-wide emissions for the defined states are produced.
COUNTY	If the global code for a state is specified (e.g., 06000), county-level estimates are calculated for all counties within the state. If a list of specific county FIPS codes is supplied, county-level estimates for just those counties will be produced.
SUBCOUNTY	To estimate subcounty emissions, the code must be a combination of a single county FIPS code and a subcounty code. For example, if you were interested in modeling the Northwest portion of Washtenaw County in Michigan, you would combine the FIPS code for the county (26161) with a subcounty code (NW) to form the code 26161NW. Estimates for just the subcounty level will be calculated. If this option is specified, appropriate data for subcounty resolution must also be supplied in the data files. Refer to the section on Modifying Inputs for Subcounty Runs in Chapter 6.

To modify the /REGION/ packet, follow these steps:

1. Locate the /REGION/ packet by using the left button on the mouse to scroll through *TRAVIS.OPT*. Alternatively, use the up/down arrow keys to scroll through the file.
2. Once you locate /REGION/, use the the mouse to click on the appropriate line and remove or change any existing information that is not relevant for your scenario. Figure 4-5 shows the /REGION/ packet with the changes required for our example.
3. If you want to add additional region codes, you can do that by entering the first FIPS code and then pressing <Enter> to create a new line. Using the space bar, move the curser under the first FIPS code and add the second code that you want. The colon

does not need to be repeated in each line. Continue in this fashion until all codes have been indicated.



```
Travis.opt - Notepad
File Edit Search Help
REGION - region codes (must match code in regions
         definition file
STATE - state FIPS codes
COUNTY - state of county FIPS codes. State FIPS
          code means include all counties in the
          state
SUBCOUNTY - county FIPS code and subregion code.
-----
/REGION/
Region level : COUNTY
FIPS Code : 48453
/END/
-----
This is the packet that defines the spatial
allocation files read by the model. These files
contain the surrogate indicators used to allocate
from state to county or county to subcounty. The
data can be stratified into any number of files.
-----
```

Figure 4-5: Modified /REGION/ packet.

Modifying the /POP FILES/ and /ALLOC FILES/ Packet

In the NONROAD model, there are several packets found in an option file that list the names of files that contain data used by the model to estimate emissions. To make these data files more manageable, some of the data is separated into files by state. In most cases, these files do not require any modifications if you are running the model with the default data provided. However, if you modify the /REGION/ packet, two data files may be affected by this modification. These two files are the population data files and the allocation data files and are located in the /POP FILES/ packet and the /ALLOC FILES/ packet, respectively.

Note: The /POP FILES/ packet defines the equipment population files read by the model. You can identify any number of files for each run. Typically, each file represents one state.

The /ALLOC FILES/ packet defines the spatial allocation files read by the model. These files contain the surrogate indicators used to allocate from the state-level to county-level or county-level to subcounty-level.

When you initially install the NONROAD model, you are prompted to identify the state where you are located. This step sets up the population and allocation factor data files for your state. If during the process of creating an option file you define a region outside of your originally defined state, you must change the state identification code of the population and allocation data files to the new state.

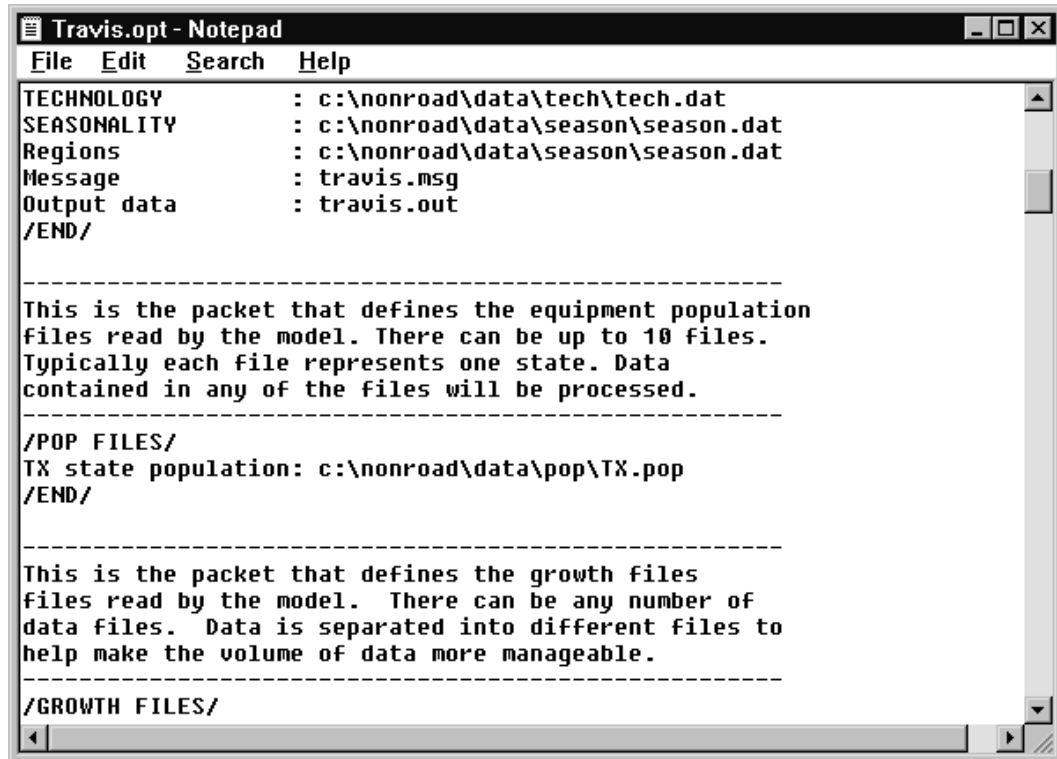
To modify the /POP FILES/ packet, follow these steps.

1. Locate the /POP FILES/ packet by using the left button on the mouse to scroll through *TRAVIS.OPT*. Alternatively, use the up/down arrow keys to scroll through the file.

Note: Population data files are named using the two character state abbreviation.

2. Once you locate /POP FILES/, check to see that the population data files contain the name of the state that you are interested in modeling. If not, use the left button on the mouse to click on the appropriate line and modify the name of the population file to identify the state you want. For our example, the population file should be changed to the file for the state of Texas, “\NONROAD\DATA\POP\TX.POP”, as shown in Figure 4-6. (Since we are only interested in a county in one state in our example, only

the population file for that state is required. If you were interested in multiple states, the population files for each state would be required.)



```

Travis.opt - Notepad
File Edit Search Help
TECHNOLOGY      : c:\nonroad\data\tech\tech.dat
SEASONALITY    : c:\nonroad\data\season\season.dat
Regions        : c:\nonroad\data\season\season.dat
Message        : travis.msg
Output data    : travis.out
/END/

-----
This is the packet that defines the equipment population
files read by the model. There can be up to 10 files.
Typically each file represents one state. Data
contained in any of the files will be processed.
-----
/POP FILES/
TX state population: c:\nonroad\data\pop\TX.pop
/END/

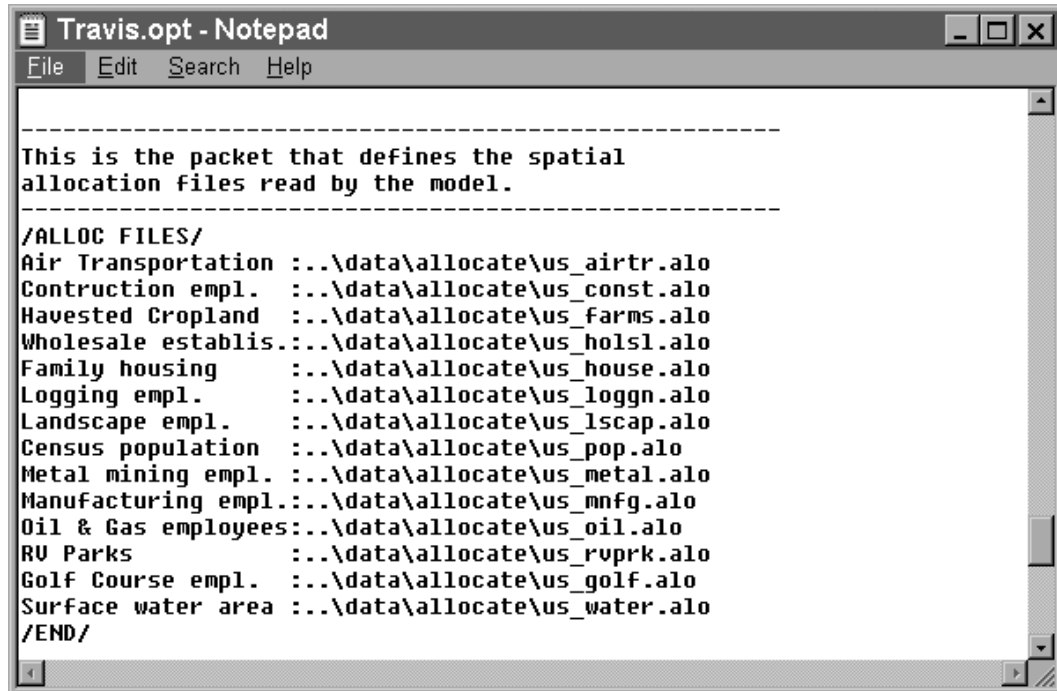
-----
This is the packet that defines the growth files
files read by the model. There can be any number of
data files. Data is separated into different files to
help make the volume of data more manageable.
-----
/GROWTH FILES/

```

Figure 4-6: Modified population data file name.

To modify the /ALLOC FILES/ packet, follow these steps:

1. Locate the /ALLOC FILES/ packet by using the left button on the mouse to scroll through *TRAVIS.OPT*. Alternatively, use the up/down arrow keys to scroll through the file.
2. As you can see in Figure 4-7, in addition to being separated by state, the allocation data files are further stratified by a series of allocation factors. Once you locate /ALLOC FILES/, check to see that the allocation data files contain the name of the state that you are interested in modeling. If not, use the left button on the mouse to click on the appropriate line and modify the name of the allocation data files to identify the state you want. For our example, the files containing the allocation data should be changed to identify the state of Texas.



```

-----
This is the packet that defines the spatial
allocation files read by the model.
-----

/ALLOC FILES/
Air Transportation :..\data\allocate\us_airtr alo
Construction empl. :..\data\allocate\us_const alo
Harvested Cropland :..\data\allocate\us_farms alo
Wholesale establis :..\data\allocate\us_holsl alo
Family housing :..\data\allocate\us_house alo
Logging empl. :..\data\allocate\us_loggn alo
Landscape empl. :..\data\allocate\us_lscap alo
Census population :..\data\allocate\us_pop alo
Metal mining empl. :..\data\allocate\us_metal alo
Manufacturing empl. :..\data\allocate\us_mnfg alo
Oil & Gas employees :..\data\allocate\us_oil alo
RV Parks :..\data\allocate\us_rvprk alo
Golf Course empl. :..\data\allocate\us_golf alo
Surface water area :..\data\allocate\us_water alo
/END/

```

Figure 4-7: /ALLOC FILES/ packet.

Modifying /SOURCE CATEGORY/

The packet /SOURCE CATEGORY/ tells the model which source categories are to be processed. While this is an optional packet, the source category code (SCC) must be defined if you would like to model only a subset of equipment types. Running the model with only the desired equipment types can reduce model run time and help focus reports on data of particular interest.

As discussed in Chapter 3, the SCC is a ten-digit number: the first four digits identify the fuel/engine type; the second six digits identify the segment and equipment. For our example, the SCC for the entire category of four-stroke,

Note: Some packets in the core model, such as /SOURCE CATEGORY/, are optional and can be removed by placing any non-blank character in the first column of the line containing the packet identifier. For example, if you wanted to “comment out” the /SOURCE CATEGORY/ packet, the first line of the packet could look like this:

```
./SOURCE CATEGORY/
```

with the comma (,) as the non-blank character.

gasoline-powered lawn and garden equipment is 2265004000 (2265 designates four-stroke, gasoline-powered equipment while 004000 signifies all lawn and garden equipment). A listing of NONROAD SCC codes is shown in Appendix B.

To modify the /SOURCE CATEGORY/ packet, follow these steps.

1. Locate the /SOURCE CATEGORY/ packet by using the left button on the mouse to scroll through *TRAVIS.OPT*.
2. Once you locate /SOURCE CATEGORY/, use the left button on the mouse to click on the appropriate line and remove or change any existing information that is not relevant for your scenario (Figure 4-8).

```

Travis.opt - Notepad
File Edit Search Help
THC EXHAUST      : c:\nonroad\data\emsfac\exhthc.det
/END/

-----

This packet is used to tell the model which source
categories are to be processed.  It is optional.
If used, only those source categories list will
appear in the output data file.  If the packet is
not found, the model will process all source
categories in the population files.

-----

/SOURCE CATEGORY/
Lawn & Garden 4-stk:2265004000
/END/

-----

This is the packet that defines the scrappage curve
used for model year distribution.  The first value
is the fraction of useful life already used and the
second value is the percentage of equipment scrapped.

-----

/SCRAPPAGE/
0          0
0.06      1

```

Figure 4-8: Modified /SOURCE CATEGORY/ packet.

Saving an Option File

After you have completed the modification of an option file to reflect the scenario that you are interested in modeling, save the file by following these steps:

Note: Steps 1 through 3 in this section are for the text editors Notepad and WordPad. If you use a different text editor, the exact key and mouse strokes may differ.

1. From the main menu of the text editor, use the left mouse button to click on **File**, 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>.
3. To exit the text editor, click the left mouse button on **Exit**, or use the up/down arrow keys to highlight **Exit** and press <Enter>. Alternatively, press <X>.

CREATING AN OPTION FILE THROUGH DOS

The process of creating an option file through DOS is quite similar to that using a text editor; however, this method requires some basic knowledge of DOS commands. To create an option file through DOS, follow these steps:

1. Open a DOS window by locating the MS-DOS icon and double clicking with the left button on the mouse.
2. In the DOS window, change to the Nonroad directory by typing “*cd c:\nonroad*” (assuming the National Nonroad Emissions Model is located on the c drive of your computer; if not, type the appropriate drive before “*\nonroad*”). Figure 4-9 shows the nonroad directory located in a DOS window.

Note: If your computer is operating with Windows 3.1, the MS-DOS icon is generally located in the program group “**Main**”. Under Windows 95, the MS-DOS icon can be found by going to the **Start** menu and then **Programs**.

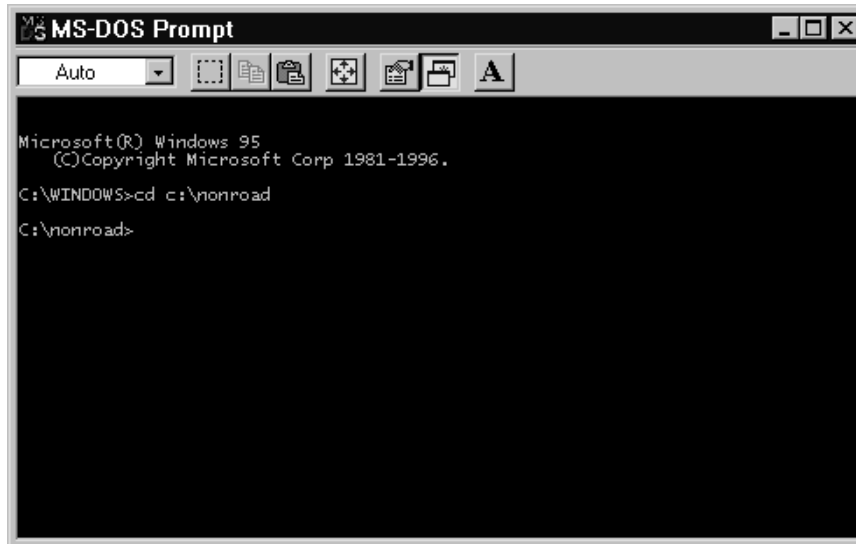
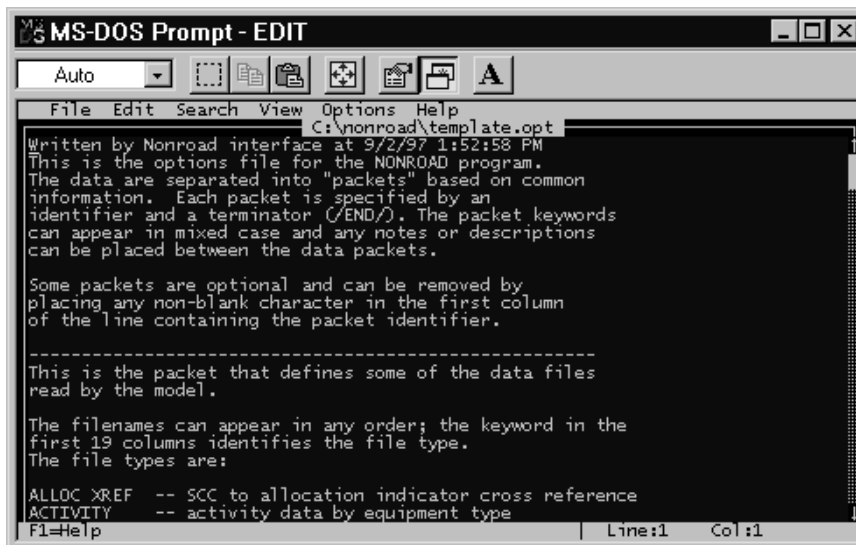


Figure 4-9: Nonroad directory in a DOS window.

3. To locate the *.opt files in the nonroad directory, type “dir *.opt”. A list of all files with the .opt extension will appear. You can select any of these files to modify.
4. To modify an existing option file, type “edit (the name of the file you want to modify).opt”. For example, if you were interested in modifying *TEMPLATE.OPT*, you would type “edit template.opt”. This command will take you into a DOS text editor as seen in Figure 4-10.

Figure 4-10: *TEMPLATE.OPT* in a DOS editor.

5. To change the /RUNFILES/, /OPTIONS/, /PERIOD/, /REGION/, and /SOURCE CATEGORY/ packets, follow the steps described in the previous section on Creating an Option File Using a Text Editor.
6. When you have completed your modifications, use the left button on the mouse to click on **Files** or press <Alt-F>. Next, click on **Save** or press <S>. To exit the DOS editor, click on **Exit** or press <X>. To return to Windows 3.1 or Windows 95 from DOS, type “*exit*”.

RUNNING THE CORE MODEL

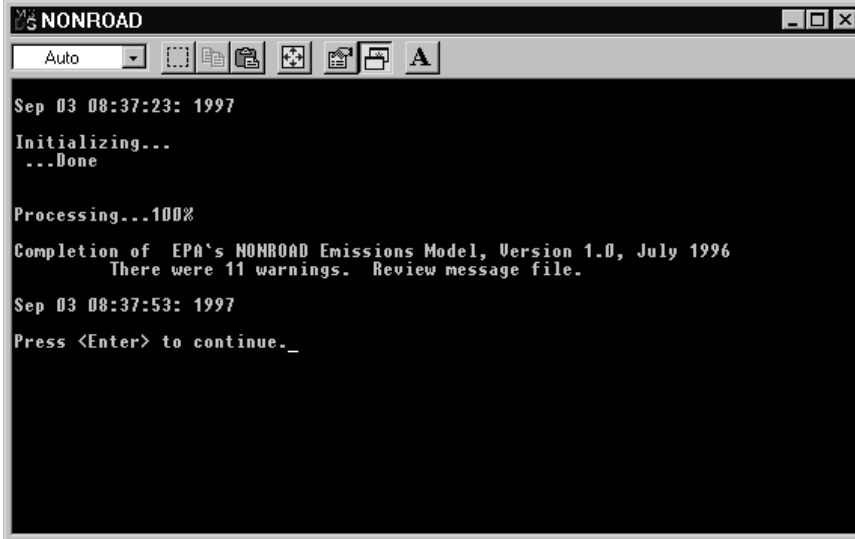
After you have created an option file that defines the scenarios you are interested in modeling, the process of running the core model is quite simple. In Chapter 3, the steps necessary to run the core model through the graphical user interface were described. In this chapter, the steps required to run the core model through a DOS window are defined.

To run the core model through a DOS window, follow these steps:

1. Open a DOS window by locating the MS-DOS icon and double clicking with the left button on the mouse.
2. In the DOS window, change to the Nonroad directory by typing “*cd c:\nonroad*” (assuming the NONROAD model is located on the c drive of your computer; if not, type the appropriate drive before “*\nonroad*”).
3. To run the core model, type “*nonroad (the name of your file).opt*”. For example, if you were interested in running the example option file *TRAVIS.OPT*, type “*nonroad travis.opt*”.
4. The model will now run with your option file. The first step that the model goes through is an initialization. During this step, the core model is reading in the option and data files and checking to ensure that all data files required to complete the run are available. After the model has initialized, it will begin the process of calculating emissions estimates based on the parameters that you set. A line on the screen will inform you of the progress of your run. When the run has

Note: Most of the errors that may occur during the running of a scenario will occur during the initialization stage. If an error occurs, a message will appear on the screen notifying you of the error.

finished, a message will appear indicating that the run was successfully completed (Figure 4-11).



```

NONROAD
Auto
Sep 03 08:37:23: 1997
Initializing...
...Done
Processing...100%
Completion of EPA's NONROAD Emissions Model, Version 1.0, July 1996
There were 11 warnings. Review message file.
Sep 03 08:37:53: 1997
Press <Enter> to continue._

```

Figure 4-11: DOS window showing successful model run.

5. To view the message file associated with this run, type “*edit (the name of the your file).msg*”. Alternatively, you can open the message file through a Windows text editor.

Error Stop Feature

When an error occurs in the core model, an informative message is displayed in the window and written to the output message file, and the program stops. In addition to error checking, a number of conditions are checked that will produce a warning message. Warning messages are meant to inform you that a condition exists which you may want to review and correct. An example is “WARNING: Monthly seasonality factors do not add to 1. Renormalizing...” If you have included new seasonality data and thought it should sum to one, then this message would indicate either a data format problem or a typo. There are conditions which could produce many warning messages and still run to completion. In order to avoid this situation, the model will stop if 50 of the same kind of warning message is generated.