

# Motor Vehicle Emission Simulator (MOVES) 2010

## User Guide

# Motor Vehicle Emission Simulator (MOVES) 2010

## User Guide

Assessment and Standards Division  
Office of Transportation and Air Quality  
U.S. Environmental Protection Agency



United States  
Environmental Protection  
Agency

EPA-420-B-09-041  
December 2009





## Table of Contents

	<u>Page</u>
1.0. Getting Started with MOVES .....	4
1.1 About MOVES.....	4
1.2 MOVES Documentation .....	5
1.2.1 About this Manual.....	5
1.2.2 Other Documentation and Online Help .....	5
1.2.3 What's Next? Working with MOVES.....	6
1.3 Installation.....	7
1.3.1 System Requirements.....	7
1.3.2 Starting and Testing Your Installation.....	8
1.4 Executing Example Run Specification .....	9
1.4.1 Step-by-step Example Run .....	9
1.4.2 MOVESExecution Database.....	9
1.4.3 MOVES Output Structure.....	10
2.0 Using MOVES.....	12
2.1 Overview of MOVES User Interface .....	13
2.2 Navigation Panel .....	15
2.2.1 Description.....	16
2.2.2 Scale.....	17
2.2.2.1 Domain/Scale .....	17
2.2.2.2 Calculation Type .....	19
2.2.3 Time Spans.....	20
2.2.3.1 Time Aggregation Level .....	20
2.2.3.2 Years .....	21
2.2.3.3 Months .....	22
2.2.3.4 Days .....	22
2.2.3.5 Hours .....	22
2.2.4 Geographic Bounds.....	22
2.2.5 Vehicles/Equipment.....	29
2.2.6 Road Type.....	30
2.2.7 Pollutants and Processes .....	31
2.2.7.1 Processes .....	33
2.2.7.2 Pollutants.....	33
2.2.8 Manage Input Data Sets.....	34
2.2.9 Creating and Using Internal Control Strategies.....	36
2.2.9.1 Adding Strategy Files .....	37
2.2.9.2 Deleting Strategy Files .....	38
2.2.9.3 Exporting Strategy Files .....	38
2.2.9.4 Naming Strategy Files .....	39
2.2.9.5 Saving and Transferring Strategy Files.....	39
2.2.9.6 Creating AVFT files outside of MOVES.....	39
2.2.9.7 Managing On-Road Vehicle Retrofit Data .....	40
2.2.9.8 Importing Retrofit Data .....	41
2.2.9.9 Rate of Progress Calculation.....	42
2.2.10 Output .....	43
2.2.10.1 Specifying Database, Time Period, and Units in Output.....	43
<b>2.2.10.1.1 Output Database.....</b>	<b>43</b>
<b>2.2.10.1.2 Units .....</b>	<b>44</b>

<b>2.2.10.1.3 Activity</b> .....	44
2.2.10.2 Specifying Emission Distinctions in Output .....	46
<b>2.2.10.2.1 Always</b> .....	46
<b>2.2.10.2.2 For All Vehicle/Equipment Categories</b> .....	47
<b>2.2.10.2.3 On Road/Off Road</b> .....	47
<b>2.2.10.2.4 Estimate Uncertainty</b> .....	48
2.2.11 Advanced Performance Features .....	48
2.3 Main Menu Bar.....	50
2.3.1 File .....	50
2.3.2 Edit.....	50
2.3.3 Pre Processing.....	51
2.3.3.1 Database.....	53
2.3.3.2 RunSpec Summary.....	54
2.3.3.3 Tools .....	55
<b>2.3.3.4 Data Managers (Importers)</b> .....	56
<b>2.3.3.4.1 Meteorology Data Importer</b> .....	62
<b>2.3.3.4.2 Source Type Population Importer</b> .....	62
<b>2.3.3.4.3 Age Distribution Importer</b> .....	62
<b>2.3.3.4.4 Vehicle Type VMT and VMT Fractions</b> .....	62
<b>2.3.3.4.5 Average Speed Distribution Importer</b> .....	64
<b>2.3.3.4.6 Road Type Distribution Importer</b> .....	64
<b>2.3.3.4.7 Ramp Fraction Importer</b> .....	65
<b>2.3.3.4.8 Fuel Formulation Importer</b> .....	65
<b>2.3.3.4.9 Fuel Supply Importer</b> .....	66
<b>2.3.3.4.10 I/M Importer</b> .....	66
<b>2.3.3.4.11 Zone Road Activity Importer</b> .....	68
<b>2.3.3.4.12 Links Importer</b> .....	68
<b>2.3.3.4.13 Link Source Types Importer</b> .....	69
<b>2.3.3.4.14 Operating Mode Distribution Importer</b> .....	70
<b>2.3.3.4.15 Link Drive Schedules Importer</b> .....	70
<b>2.3.3.4.16 Off Network Importer</b> .....	70
2.3.4 Action.....	71
2.3.5 Post Processing .....	72
2.3.5.1 Run MySQL Script on Output Database.....	72
<b>2.3.5.2 Produce Summary Report</b> .....	73
<b>2.3.5.3 Produce State/County Map</b> .....	77
2.3.6 Tools .....	78
2.3.7 Settings.....	82
2.3.7.1 Configure MOVES .....	82
2.3.8 Help.....	83
Appendix A – Using MOVES to Generate Lookup Tables .....	1
Appendix B - Linking MySQL Tables from MS ACCESS or MS Excel (includes ODBC usage).....	1
Appendix C - Using the Batch Command Line Interface .....	1
Appendix D - Creating an On-Road Retrofit Input File .....	1
Appendix E - Best Practices .....	1
Appendix F - Scenario 1: Estimating Changes in CO <sub>2</sub> using the AVFT .....	1
Appendix G - Scenario 2: MOVES Project Level Example .....	1
Appendix H - Scenario 3: County Data Manager Example and Basic MySQL Queries for Analyzing Output .....	1
Appendix I – Stage II Refueling Control Programs.....	1
Appendix J – MOVES “Decoder” .....	1



## **1.0. Getting Started with MOVES**

MOVES is the U.S. Environmental Protection Agency's (EPA) Motor Vehicle Emission Simulator. It helps the user to answer "what if" questions, such as "How would particulate matter emissions decrease in my state on a typical weekday if truck travel was reduced during rush hour?" or "How does the total hydrocarbon emission rate change if my fleet switches to gasoline from diesel fuel?" The purpose of the tool is to provide an accurate estimate of emissions from mobile sources under a wide range of user-defined conditions.

In the modeling process, the user specifies vehicle types, time periods, geographical areas, pollutants, vehicle operating characteristics, and road types to be modeled. The model then performs a series of calculations, which have been carefully developed to accurately reflect vehicle operating processes, such as cold start or extended idle, and provide estimates of bulk emissions or emission rates. Specifying the characteristics of the particular scenario to be modeled is done by creating a Run Specification, or RunSpec.

The MOVES model is different from previous EPA mobile source emissions models in that it was deliberately designed to work with databases. With this design, new data that may become available can be more easily incorporated into the model. In addition, MOVES allows and facilitates the import of data specific to a user's unique needs.

The MOVES model includes a "default" database that summarizes emission relevant information for the entire United States. The data for this database comes from many sources including EPA research studies, Census Bureau vehicle surveys, Federal Highway Administration travel data, and other federal, state, local, industry and academic sources. The MOVES team continually works to improve this database, but, for many uses, up-to-date local inputs will be more appropriate, especially for analyses supporting State Implementation Plans (SIPs) and conformity determinations.

### **1.1 About MOVES**

MOVES2010 is the latest version of the MOVES emissions modeling tool. MOVES2010 builds on the functionality of previous MOVES versions: MOVES2004, MOVESDemo, and DraftMOVES2009. MOVES2010 can be used to estimate national, state, and county level inventories of criteria air pollutants, greenhouse gas emissions, and some mobile source air toxics from highway vehicles. Additionally, MOVES2010 can make projections for energy consumption (total, petroleum-based, and fossil-based).

MOVES2010 is intended for official use. Please see the MOVES2010 policy statement available on the EPA's MOVES web site <http://www.epa.gov/otaq/models/moves/index.htm> for EPA's policy on selecting appropriate models for official mobile source emission estimates.

Future versions of MOVES will be able to estimate pollutants from other mobile sources such as aircraft, locomotives, and commercial marine vessels. In addition, the capability to estimate non-highway mobile source emissions is planned.

MOVES is distributed free of charge by EPA pursuant to the GNU General Public License (GPL). It is written in Java™ and uses the MySQL relational database management system. Sun Microsystems owns, operates, and supports MySQL, and allows distribution of the database system pursuant to the GNU GPL. The principal user inputs and outputs, and the internal working storage locations for MOVES are MySQL databases. The MOVES2010 model includes a "default" input database, which uses national data and allocation factors to approximate results for the 3,222 counties in the United States, District of Columbia, Puerto Rico, and the U.S. Virgin Islands. MOVES2010 is capable of modeling emissions for the calendar years 1990 and 1999-2050.

MOVES is set up to run both a "Master" and one or more "Worker" sub-programs. This allows users to operate MOVES on a single computer system or on a network of computers. See the MOVES2010 Installation Guide and/or the Software Design Reference Manual for more information about specific requirements and computer configurations.

## 1.2 MOVES Documentation

The following discussion highlights the contents of this manual, how best to use the manual, and where additional information may be found.

### 1.2.1 About this Manual

The MOVES2010 User Guide focuses on operating MOVES software to create and execute Run Specifications (RunSpecs).

Throughout this document, information may be highlighted to make it easier to find solutions to problems or to avoid errors using the following prompts:

**⚠ CAUTION!** Cautions must be observed to avoid errors in execution or to ensure the intended execution will occur.

**📖 NOTE** Notes contain important information about the panel being described.

**! TIP** Tips contain hints for input or better operation of the run.

The MOVES2010 User Guide is accessible as a .pdf file under the **Help** menu in the MOVES interface.

### 1.2.2 Other Documentation and Online Help

Additional documentation covering the following topics is available for MOVES2010.

**Installation:** A README text file is included on the MOVES Installation CD. It guides the user through the MOVES system requirements and the process for installing the MOVES program and associated software.

**Software Design:** The MOVES Software Design and Reference Manual (SDRM) will be available on the MOVES website. The SDRM covers the basic concepts and functional design of the software and the underlying MySQL database, including technical specifications for all calculations performed in the model. It is intended to answer more detailed and substantive questions about the MOVES software than those addressed in this guide.

**MOVES Database Documentation:** More detailed documentation of the MOVES database can be found within the README folder of the database itself. For example, C:\MySQL\data\MOVESDB20091214\readme

### 1.2.3 What's Next? Working with MOVES

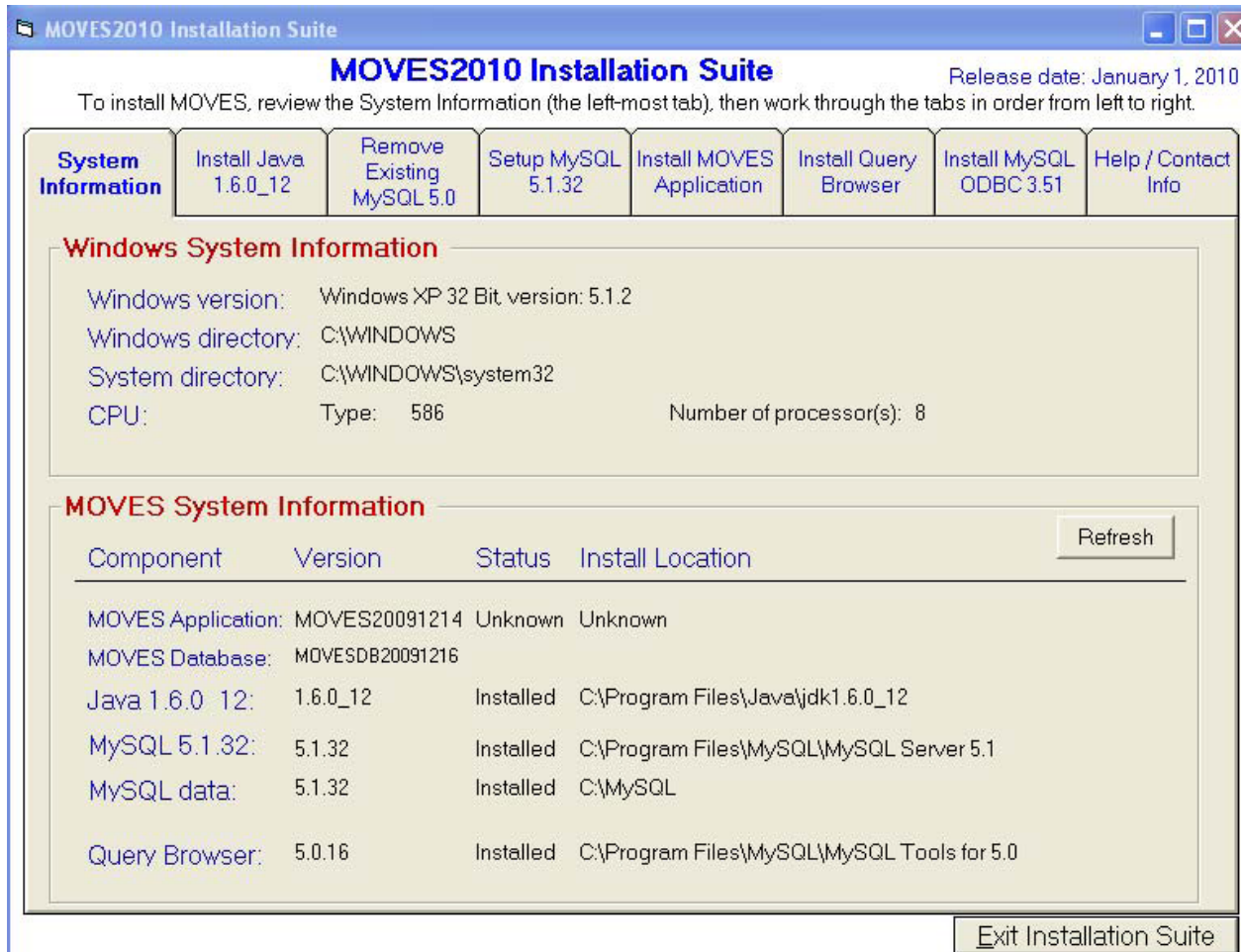
Depending on the status of your installation and how familiar you are with MOVES, you have several options for next steps to get up and running with MOVES2010.

- ⇒ If you have just installed MOVES or are uncertain about your installation, follow instructions in "Testing Your Installation," provided in Section 1.3.2 of this guide.
- ⇒ If MOVES is installed and operational on your computer, you should run the example to become more familiar with the tool before moving on. See "Executing Example MOVES2010 Run Specification" included in Section 1.4.
- ⇒ If you would like to become more familiar with MOVES terminology and design, see the "MOVES Software Design and Reference Manual," available in .pdf format on the MOVES website.
- ⇒ If you have questions about the MOVES Graphical User Interface (GUI) or executing a run, see "Overview of MOVES RunSpec User Interface" in Section 2 of this guide.
- ⇒ If you are ready to begin using MOVES, see "Starting MOVES," also in Section 2 of this guide.
- ⇒ If you would like to work through an example, a set of sample scenarios is included in the Appendices.
- ⇒ If you would like to get started using MOVES for regular work, advice and guidance on using MOVES is available on the MOVES website.

The rest of this guide provides information on using the MOVES2010 model.

## 1.3 Installation

Step-by-step instructions on installation are supplied by EPA in the MOVES2010 Installation Suite. The MOVES2010 Software Installation Suite consists of the required installation programs (MySQL 5.1.32, Java 1.6.0\_12, and MOVES2010, MySQL Query Browser, and MySQL ODBC).



### 1.3.1 System Requirements

MOVES2010 is a 32-bit application and has been developed to work only in Microsoft Windows XP and later environments. Note that installing MOVES with Microsoft Vista or Windows7 may require changes to the User Access Control (UAC) security settings. VISTA 64-bit architecture is supported at this time. Computer(s) used to run either of the MOVES application programs must have at least 256 MB of RAM, although it is recommended that 512MB or more be available to improve execution time. Because execution run time performance is a constraint with MOVES, high speed processor(s) are highly recommended.

**CAUTION!** It would be detrimental to performance to operate more than one copy of the MOVES Worker program on a single computer.



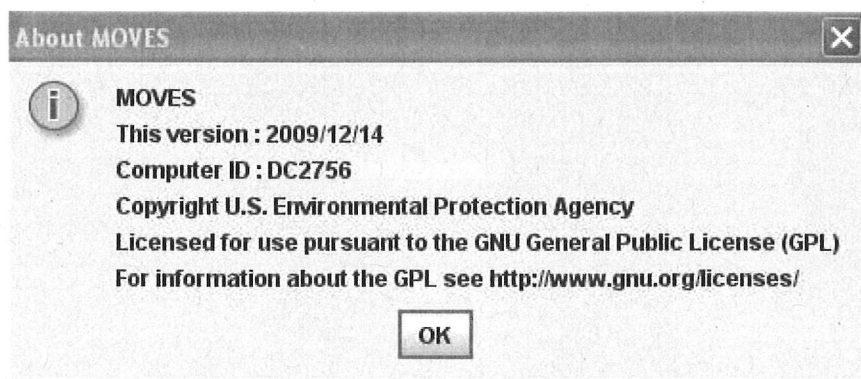
The MOVES program is open source and is written in JAVA and MySQL.

The MOVES Default database distributed with MOVES2010 requires approximately 1.0 GB of disk storage. MOVES Worker and Output databases are often large, so several additional gigabytes of disk space should be available on all machines used to run either the MOVES Master and/or the MOVES Worker program. Users who conduct many MOVES runs will want to use late-model, high-performance microcomputer systems.

### 1.3.2 Starting and Testing Your Installation

After all necessary components of the MOVES2010 Installation Suite have been executed, test the installation with the following steps.

1. Check that the MySQL server is operating. This is done by opening the MS DOS prompt (Start-Programs-Accessories-Command Prompt) and changing the path to "C:\mysql\bin" (c:\> cd \mysql\bin). After changing the path, the user should type 'mysql' at the prompt. The MySQL program should begin. If an error message appears, the MySQL program or server has not been successfully installed. Once installed, the MySQL server will be set up to run automatically as a Windows system service.
2. Double-click the **MOVES Master** program icon.
3. When starting the **MOVES Master** program, the **About MOVES** panel will appear in the center of the GUI screen. Click the "OK" button to continue.



¶ **NOTE** This panel identifies the version of MOVES that is in use and includes the version date, the EPA copyright, and a link for the GNU General Public License (GPL) website. Also included is the Computer ID or name. This panel will appear only at start-up.

4. Execute the Example Run Specification (MOVES2010Example.mrs) as described in the following section.

## 1.4 Executing Example Run Specification

The MOVES2010 Example Run Specification (**MOVES2010Example.mrs**) models the entire U.S. for one year (1999) at the most aggregate level. It estimates total energy consumption and emissions of methane and nitrous oxide for the running, start, and extended idle processes for all vehicle (use) types (gasoline, diesel, and CNG transit buses) and roadway types. Vehicle Miles Traveled (VMT) is also estimated.

### 1.4.1 Step-by-step Example Run

If MySQL and MOVES2010 have been installed, the MySQL server and the **MOVES Master** program have been started, and the **MOVES Worker** program(s) are operational, the example RunSpec can be executed as follows:

1. Select **File, Open** on the Main Menu Bar.
2. Navigate, if necessary, to the top level of the MOVES directory.
3. Select the **MOVES2010Example.mrs** file.
4. Select **Output**, then **General Output** on the Navigation Panel, which is the left-side panel. Select or click on Create Database to establish a database into which results should be placed. Select or enter a database name.

**NOTE** Many MOVES features request that the user create input or output databases. These database names must be consistent with MYSQL naming conventions. In general, this means they may contain letters, numbers, "\_" (underscore) and "\$". They may not contain spaces, special characters, or be "reserved words" (SQL words such as "select", "where", "delete", etc.). See Appendix E, "Best Practices" for additional advice on naming MOVES databases.

5. Select **Action, Execute** on the Main Menu Bar to start the RunSpec. The GUI will ask you if you want to save the RunSpec before executing. You may choose Yes, No or Cancel. The RunSpec will execute if either Yes or No are chosen, but selecting Cancel will not execute and will allow users to make changes to the RunSpec. This RunSpec may take up to 15 minutes to process.
6. The output will be saved in the MySQL relational database named in Step 4. The output can be reviewed using the Post-Processing menu options described in Section 2.3.5, below, or by viewing the database directly using MYSQL commands, either through the command prompt window or through MySQL Query Browser.

### 1.4.2 MOVESExecution Database

After **Execute** has been selected, MOVES begins to compile and populate all the necessary tables in the **MOVESExecution** database. This database is in many respects similar to the default database, but there are several key differences.

The **MOVESExecution** database is the location where conflicts are resolved between the default database and user input databases. First, information is pulled from the **Domain Input Database** on the **Geographic Bounds** panel (see section 2.2.4). MOVES takes note of the tables and key fields within the tables that are imported and uses that to limit what is imported from database tables that are lower in the hierarchy. Next to be added to the **MOVESExecution** database are user input databases listed on the **Manage Input Data Sets** panel (see section 2.2.8), with the last database listed taking priority over those listed above it. Finally, MOVES uses the default database to populate the remaining tables not provided by the user.

Also, the **MOVESExecution** database contains data in tables that are empty in the default database and additional tables that are not present in the default database. These tables are often not complete: to save memory, much of these additional data are cleared after they are used in calculations. Users can save more of this data by checking one or more **Save Data** boxes on the **Advanced Performance Features** panel (see section 2.2.11). Note that the **Save Data** process lengthens run time and the tables it produces are often difficult to interpret correctly.

### 1.4.3 MOVES Output Structure

The output database will typically contain eleven tables. The tables are:

**1. MovesOutput:** This table contains the inventory emission results of the run disaggregated by parameters such as Year, Month, Day, Source Type, County, Fuel, Model-Year, Road-type, SCC, and Emission Process.

**⚠ Caution!** If the calculation type **Emission Rates** is selected on the **Scale** panel, the emissions values will represent an intermediate step in the rates calculations rather than the real emissions for that time and place. To reduce file size, use the **Aggregation and Data Handling** panel on the **Advanced Performance Features** screen to select “**Clear MOVESOutput after rate calculations.**”

**2. MovesActivityOutput:** This table provides information on the vehicle activity generated and used in the run. It is disaggregated by parameters such as Year, Month, Day, County, Fuel, Model-Year, Road-type, and SCC. Activity is not distinguished by pollutant and process. If a certain activity type (see also Section 2.2.10.1.3) is not needed for the emission processes in a run, it is not generated.

**🔗 NOTE** In the **MOVESActivityOutput** table, population always has month, day & hour keys of zero (i.e. “doesn’t matter”) because population is assumed constant in time across an entire year. This scheme reduces table size, but requires care when filtering results and when joining to other tables. Also, if you assign some VMT to a road type, but then do not select that road type in the RunSpec, the population will be lower than it should be. This is because the assigned VMT is not used when calculating population. Users should model all road types (where VMT is allocated) to ensure proper calculations.

**⚠ Caution!** If the calculation type **Emission Rates** is selected on the **Scale** panel, the activity values will represent an intermediate step in the rates calculations

rather than the real activity for that time and place. To reduce file size, use the **Aggregation and Data Handling** panel on the **Advanced Performance Features** screen to select “**Clear MOVESActivityOutput after rate calculations.**”

- 3. MovesRun:** This table contains information about the date and time of the run, information about the run specification, and the name of the units in which MOVES outputs are represented. These include units for energy (i.e., joules), emissions (i.e., tons), VMT (i.e., miles), and time units (i.e., months, days, hours, etc.).
- 4. MovesError:** This table contains any error messages or diagnostic information that might be generated if the MOVES run is unsuccessful.
- 5. MovesWorkersUsed:** This table contains information as to which copy of the MOVES Worker Program processed portions of the run. It is of interest if the run is executed on a multi-computer configuration.
- 6. Activity Type:** This table lists the activity types that can be reported in the movesactivityoutput table and provides their activitytypeid. In MOVES2010 this table lists: distance traveled, source hours, extended idle hours, source hours operating, source hours parked, population, and starts. (See also Section 2.2.10.1.3)
- 7. MovesEventLog:** This table is used by MOVES programmers for storing diagnostic results. It contains no information useful to users.
- 8. RatePerDistance:** When “Emission Rates” is selected on the Scale panel, the RatePerDistance table stores emissions as rates per distance (e.g. grams per mile) with the units depending on those selected in the run specification. This table includes rates for exhaust and evaporative emissions that occur while vehicles are on “real roads”, but does not include rates for starts, extended idle, or evaporative emissions (with the exception of refueling) that occur while a vehicle is parked.
- 9. RatePerProfile:** When “Emission Rates” is selected on the Scale panel, the RatePerProfile table stores vapor venting emissions from parked vehicles as rates per vehicle (e.g. grams per vehicle). Note that the denominator is the total vehicle population—not just the vehicles that are parked. The temperature profile id in this table refers to a particular daily pattern of temperatures, important because vapor venting depends on temperatures in both the current and previous hours.
- 10. RatePerVehicle:** When “Emission Rates” is selected on the Scale panel, the RatePerProfile table stores emissions from starts and extended idle, and some evaporative emissions (permeation and liquid leaks) from parked vehicles as rates per vehicle (e.g. grams per vehicle). Note that the denominator is the total vehicle population—not just the vehicles that are parked.
- 11. MovesTablesUsed:** This table contains a list of the tables used when executing the RunSpec and the source database for each table. The numbering of the tableUseSequence is such that lower numbers take priority. Some tables may be listed

more than once; this occurs when information comes from both the user and default databases.

## 2.0 Using MOVES

Using MOVES2010, the user can model various scenarios in order to determine specific emissions profiles. MOVES2010 requires the user to set up a RunSpec to define the place, time, vehicle, road, fuel, emission producing process, and pollutant parameters. The RunSpec is an XML file that can be edited and executed directly or with the MOVES Graphical User Interface (GUI). This section of the User Guide gives an overview of the screen layout and navigation functions of the GUI, and provides details on how to create and modify a RunSpec.

Appendix C provides instructions for how MOVES can be executed without running its graphical user interface. This may be useful in situations where repeated or unattended runs are needed, or when another computer program is used to execute MOVES. This "batch mode," or "command line" interface presumes that a MOVES RunSpec file has been prepared.

Although a full range of default data is available in MOVES, the allocation of the default data down to the county level is based on a generalized algorithm and will not be the most recent nor best available local data. Therefore, MOVES is designed to allow the user to customize the model. The primary reasons a user would customize MOVES are:

- To better represent conditions and vehicle activity in a local area, especially for use in conformity demonstrations or for SIP purposes; and,
- To alter inputs for "what-if" analysis, such as the use of different I/M or fuel scenarios.

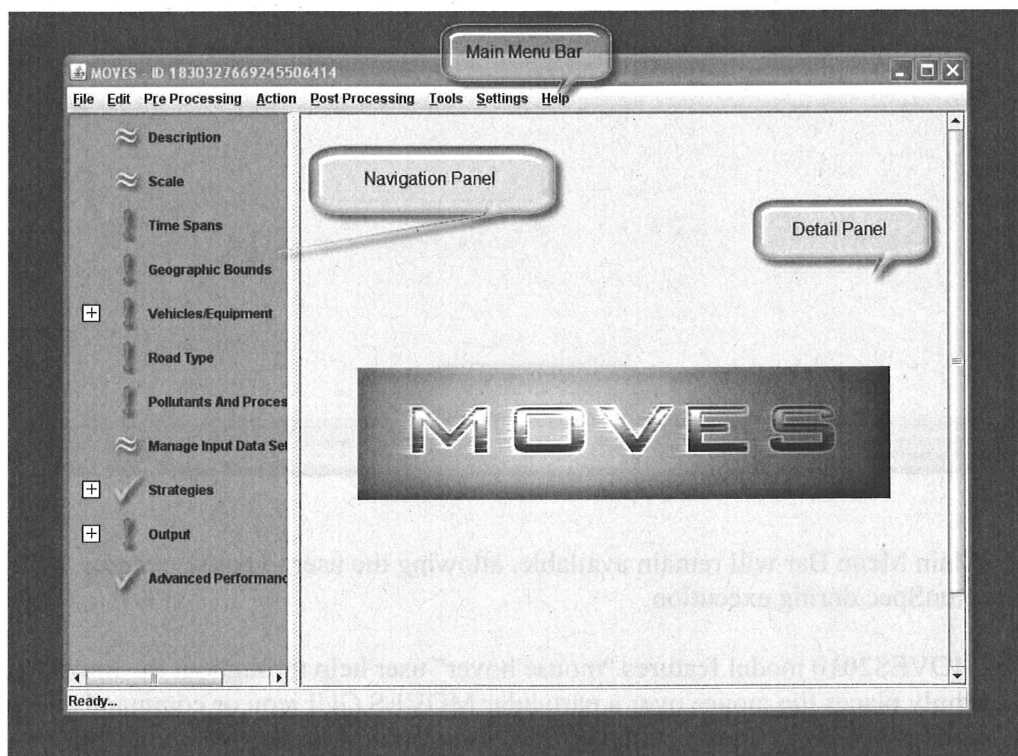
Several components of the model are intended to facilitate customization. These include the **Strategies** procedures described below. MOVES also allows users to provide **User Input Databases** and to use various **Importers** with the **County Data Manager**, and Project Domain Manager described later in this document.

For future versions of MOVES, EPA intends to develop additional data importers or strategies to make it easier to incorporate alternate activity inputs.

**NOTE** Many MOVES features request that the user create input or output databases. These database names must be consistent with MYSQL naming conventions. In general, this means they may contain letters, numbers, "\_"(underscore) and "\$". They may not contain spaces, special characters, or be "reserved words" (SQL words such as "select", "where", "delete", etc.). See Appendix E, "Best Practices" for additional advice on naming MOVES databases.

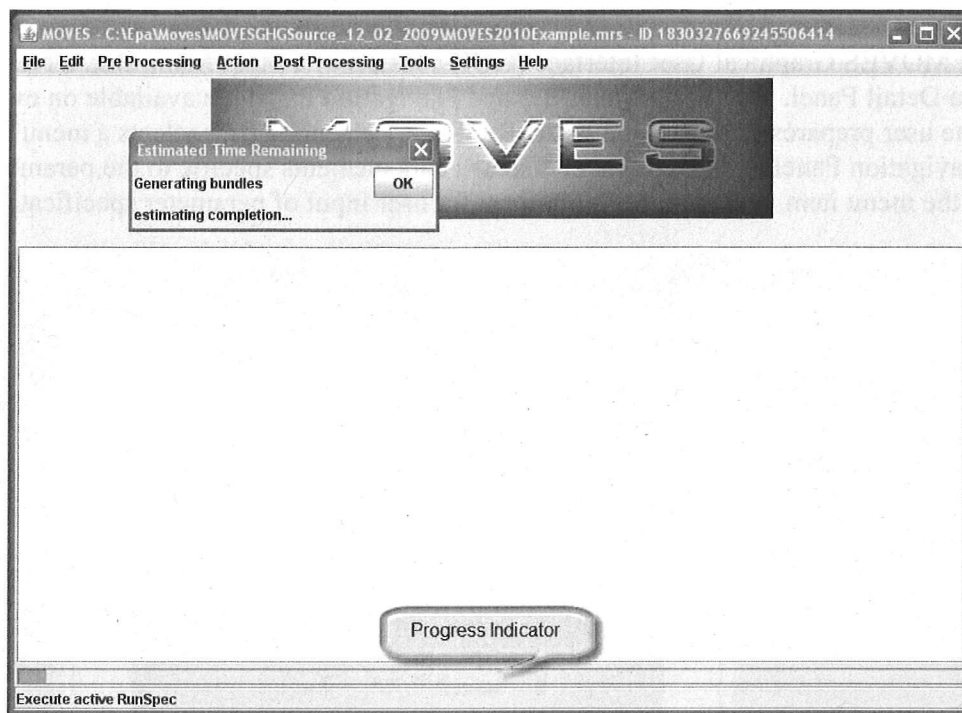
## 2.1 Overview of MOVES User Interface

The MOVES Graphical User Interface (GUI) consists of a Main Menu Bar, a Navigation Panel, and a Detail Panel. The Main Menu Bar and Navigation Panel are available on every screen as the user prepares for execution of the RunSpec. When the user selects a menu item from the Navigation Panel, the Detail Panel displays data elements specific to the parameter(s) defined by the menu item, and provides a platform for user input of parameter specifications.





During execution of a RunSpec, a progress indicator will appear on the screen.



The Main Menu Bar will remain available, allowing the user to pause, resume, or stop (cancel) the RunSpec during execution.

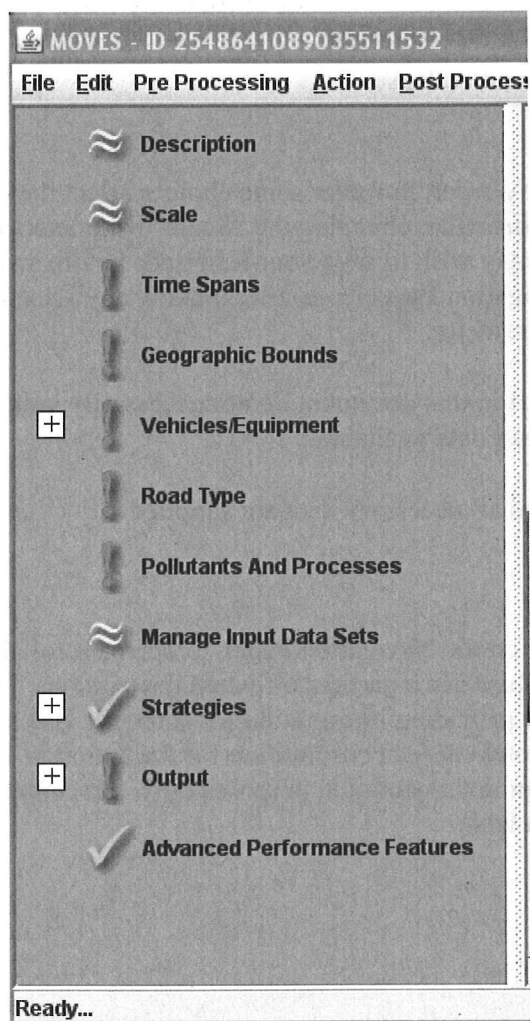
The MOVES2010 model features "mouse hover" user help throughout the model. To use it, the user simply places the mouse over a particular MOVES GUI icon or command and keeps it still. A small help box will appear with a brief text message; for example, the readiness status of a Navigation Panel icon, the keyboard combination required to execute a command from the Main Menu Bar, or a brief description of a Detail Panel icon.



Most panels use buttons and scroll or drop down list boxes, typified by the "**Select All**" "**Delete**" and "**Add**" buttons. These buttons will be enabled or disabled based upon selection in the associated list box. For example, a "**Delete**" button will be disabled until a selection is made in the associated list box.

**Note** When a selection is identified (e.g., by highlighting one state or county in a **Geographic Bounds** list box), the user must click the "**Add**" button to add the selection to the RunSpec. The user can consult the associated icon in the Navigation Panel to determine if the parameter has been sufficiently defined. If the parameter icon changes from exclamation point to check mark, the parameter definition is sufficient to allow the RunSpec to be executed.

## 2.2 Navigation Panel





The Navigation Panel appears on the left side of the MOVES screen and includes parameters the user must consider when setting up a RunSpec.



Selecting an item from the Navigation Panel will open a user interface in the Detail Panel in the center of the MOVES screen. In some cases, the RunSpec parameter listed on the Navigation Panel is split into sub-topics; these items are shown with an icon ( or ) that allows the list of sub-topics to be expanded or collapsed. The user must click on the sub-topic for the user interface to be opened.

The Navigation Panel includes icons that display the readiness status of the various RunSpec parameters. A RunSpec will not be executable if an exclamation mark is shown for any parameter.



Icon	Meaning
	Needs additional user supplied data.
	Sufficiently filled in to run.
	Default data present, but otherwise sufficiently filled in to run.
	Parameter expand/collapse.

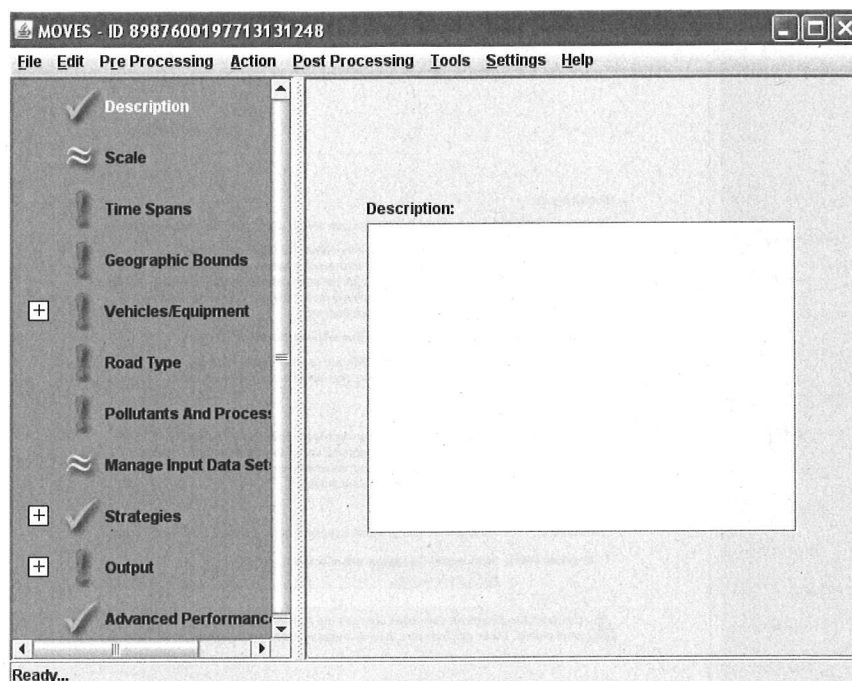
**Note** Parameters may be visited in any order, however some choices affect the choices available in other screens. In particular, changing the “Scale” parameters can impact other parameter choices. You may wish to save your Run Spec before switching Scales and to pay attention to the Navigation Panel icons after making any selection to make sure it did not affect another parameter.

**Tip** The icons shown on the sample GUI in this document do not necessarily indicate which sections/subsections will have default data available.

**Tip** A RunSpec cannot be executed until all necessary data are supplied.

### 2.2.1 Description

Select **Description**, the first item on the Navigation Panel, to open a scrollable text window that allows the user to give the RunSpec a particular textual description. This is useful for keeping track of various RunSpecs or providing information for someone else looking at the file (such as "This run produces annual total energy consumption for the nation in 1999"). The text entered in the Description Panel is for documentation purposes only. Its contents are reported for each run in the MOVESRun table.



Up to 5,000 characters of text may be entered to describe the RunSpec.

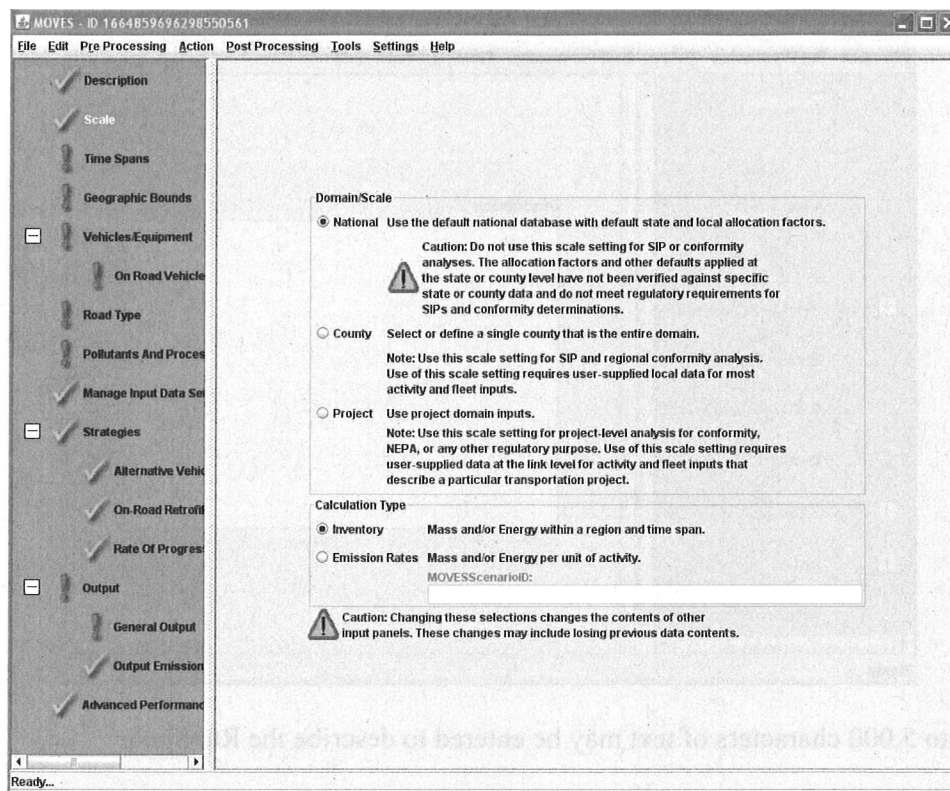
**Tip** The default Description is blank and the MOVES model can be run without a Description. However, it is considered a best practice to carefully describe each and every version of a RunSpec. This will assist in documenting the purpose of that particular MOVES2010 run. See Appendix E, Best Practices, for additional information.

## 2.2.2 Scale

Select **Scale** on the Navigation Panel to choose the level of detail represented by the MOVES input database and whether the model will calculate emissions *sum* or emissions *rate* data.

### 2.2.2.1 Domain/Scale

**National Domain/Scale** is the default selection in MOVES. If **National Domain/Scale** is selected, data collected on a nation-wide level is apportioned or allocated to states or counties. This data will differ from data collected for a specific state or county. For example, consider vehicle miles traveled. If using the **National Domain/Scale**, the primary input is national VMT data which is then allocated to a county or state. The allocation does not take into account factors that may differ between areas of the country (e.g. age distribution). Consequently, a model run for two states or counties that uses the **National Domain/Scale** may not provide an accurate portrayal of specific emission differences between these counties or states that would occur due to dissimilar vehicle age distributions or other factors. For these reasons, the **National Domain/Scale** cannot be used by states for preparing SIP or conformity analyses.



For a finer level of detail about a particular area, the user can choose the **County Domain/Scale**. With this scale choice, the model will replace national default allocations with user-supplied data. Only one year and one county (or custom domain) can be selected in the **Time Span** and **Geographic Bounds** panels, respectively. By supplying local data through the County Data Manager (see Section 2.3.3) the user can provide more accurate information about a particular state or county than would be provided using the **National Domain/Scale**. The **County Domain/Scale** is intended to be used when doing SIP or regional conformity analyses.

**Project Domain/Scale** is the finest level of modeling in MOVES. It allows the user to model the emission effects from a group of specific roadway links and/or a single off-network common area. The definition of a roadway link is a section of any road where a vehicle is moving for more than three seconds. An off-network common area may include project boundaries where vehicle starts and extended idling emissions are produced. The use of the Project Domain/Scale requires the user to completely define the individual project (i.e., specify all individual roadway links and/or the off-network common area). All of the project level data is entered in the Project Domain Manager (see Section 2.3.3.3.2) for data input details). If specified correctly, the MOVES Project Domain/Scale will utilize MOVES emission rates and other factors to correctly calculate emission inventory results for the user's defined project.

**Note** MOVES2010 is not capable of modeling any evaporative emission processes when using **Project Domain/Scale**. EPA plans to add this feature in future versions of MOVES.

Note that switching scales is a major change to the RunSpec. It affects the operation of other Navigation Panel items and may cause previous RunSpec specifications to be changed or lost. You may wish to save your RunSpec before switching scales. Also, the choice the user makes on the **Domain/Scale** panel affects the selections that will be presented in the **Geographic Bounds** detail panel. These differences are covered in greater detail in section 2.2.4 **Geographic Bounds**.

### 2.2.2.2 Calculation Type

In this section of the panel, the user can choose the type of calculation that will occur in the Run. MOVES will calculate the quantity of emissions and/or energy used within a region and time span if the user selects the **Inventory** button. This output is stored in the MOVESOutput and the MOVESActivityOutput tables.

The rate at which emissions occur (the mass and/or energy per unit of activity) is calculated if the user selects the **Emission Rates** button. This output is stored in the RatePerDistance, RatePerProfile and RatePerVehicle Tables.

Calculation Type

☐ Inventory Mass and/or Energy within a region and time span.

☒ Emission Rates Mass and/or Energy per unit of activity.

MOVESScenarioID:

If you select Emission Rates, you must assign a MOVESScenarioID which will be used in the rates tables. The MOVESScenarioID should be a unique identifier for the scenario for which the rates apply. See Appendix A for more information on defining scenarios.

¶ **Note** Calculating **Emission Rates** requires more run time and more output space than inventory calculations, so this choice should be selected with care. See Appendix A for advice on using the **Emission Rates** option to develop a “lookup table” of rates.

¶ **Note** If **Emission Rates** is selected as the **Calculation Type**, then several other panels are affected:

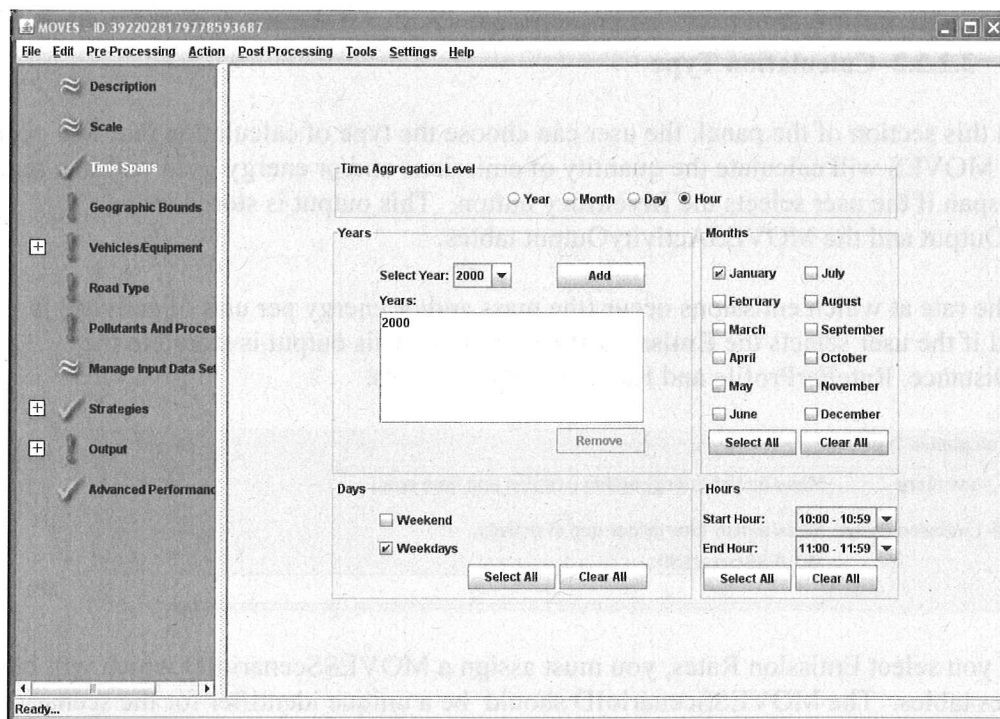
1) If **National Domain/Scale** is selected, only “**Zone & Link**” can be chosen in the “**Region**” section of the **Geographic Bounds** panel, or 1b) if **County or Project Domain/Scale** is selected, only “**Zone & Link**” or “**Custom Domain**” can be chosen;

2) The “**Distance Traveled**” and “**Population**” boxes on the **General Output** panel are automatically checked; and

3) The “**Road Type**” and “**Emission Process**” boxes are automatically selected on the **Output Emission Detail** panel.

### 2.2.3 Time Spans

MOVES can be set to model specific time periods by selecting **Time Spans** on the Navigation Panel. Clicking **Time Spans** will open the panel shown below.



This panel is divided into five sections with boxes, buttons, or drop-down menus in each, allowing the user to select specific aggregation levels, years, months, days, and hours. These panel sections are used to define the time period for which emissions will be modeled in the Run. In the screenshot depicted above, the user has specified that the Run should calculate emissions for the two hours from 10 am to noon on a typical January weekday in the year 2000. The emissions for each hour will be calculated separately.

#### 2.2.3.1 Time Aggregation Level

Click in the **Time Aggregation Level** section to set the level of pre-aggregation that is desired. Only one choice can be selected. The default level is "**Hour**" and implies no pre-aggregation of the MOVES data by time. If the user chooses a longer aggregation level time period such as "**Year**", "**Month**", or "**Day**", the model will pre-aggregate (i.e., compute a weighted average of) all underlying MOVES internal data that are differentiated by hour (e.g., temperatures, VMT distributions, etc) prior to execution of the run. For example, if the user chooses "**Month**", the model will average the temperatures of all selected Days and Hours into an average "**Month**" set (it may be a set because the temperatures will not necessarily be averaged by geography unless that has also been pre-aggregated), and perform the simulation with these average values. Pre-aggregation sacrifices precision to improve model run time.

**Note** Once the user has selected a higher aggregation level than "**Hour**", the model's GUI will automatically fill in the required lower GUI inputs. For example, if the user selects the "**Month**" button in the **Time Aggregation Level** panel, the model will fill in all of the hours and days input. After the initial automatic selection, the user may de-select particular hours, kinds of days, or months. If such a de-selection is done, the user will end up with results that are based on aggregations that do not include de-selected hours, days, or months. For example, if the user selects "**Months**" in the **Time Aggregation Level** panel and de-selects "**Weekends**", the final emission result will represent a monthly aggregation that includes "**Weekdays**" only.

**Note** The proper levels of time aggregation for SIP and conformity analysis will be addressed in guidance documents.

**Caution!** Because of non-linearity in the general MOVES algorithm, differences may arise between aggregated results and non-aggregated results. For example, the emission results from a run where all 24 hours are selected and the **Time Aggregation Level** is set to "**Hour**" may not match the results from a run where the **Time Aggregation Level** is set to "**Day**".

#### 2.2.3.2 Years

Select a calendar year(s) in the **Years** section and click on the "**Add**" button. Select a year by clicking the black triangle to see a drop-down list of calendar years and select an individual year. Click on "**Add**" to have that year appear in the **Year** selection pane. This can be repeated to select as many years as desired if **National Domain/Scale** has been chosen. However, if **County or Project Domain/Scale** was chosen on the **Scale** panel, only 1 year can be selected in the **Year** pane. Use the "**Remove**" button to deselect years.

**Tip** At least one calendar year must appear in the **Years** Selection panel to obtain a valid **Time Spans** input



### 2.2.3.3 Months

Click one or more individual months to model to select the appropriate boxes in the **Months** section. At least one **Month** box must be selected. Click the "**Select All**" button to select all of the months. Similarly, the "**Clear All**" button removes all of the previously chosen months.

¶ **Note** If the **Project Domain/Scale** is selected, the Run Spec may cover only a single month..

### 2.2.3.4 Days

Click one or more kinds of days of the week to model to select the appropriate boxes in the **Days** section. At least one kind of **Day** box must be selected. Click "**Select All**" button to select all of the days. Similarly, the "**Clear All**" button removes all of the previously chosen day types.

¶ **Note** If the **Project Domain/Scale** is selected, the Run Spec may cover only a single type of day.

### 2.2.3.5 Hours

The default time resolution for MOVES at both **National** and **County Domain/Scale** is hourly, with hours expressed in military time ranging from midnight - 12:59 am (expressed as 00:00 - 00:59) to 11 pm - 11:59 pm (23:00 - 23:59). Use the drop down menu to select the appropriate Start and End hours from the list. Click the "**Select All**" button to select all of the hours of the day. Similarly, the "**Clear All**" button removes all of the previously selected hours of the day.

! **Tip** The times selected are relative to the time zone so that 7:00 to 9:59 AM is 7:00 to 9:59 AM in each selected county-state combination.

! **Tip** If only one hour of output is desired, select the same entry for start time and end time (i.e., 0-0:59 and 0-0:59). This will produce one hour of output.

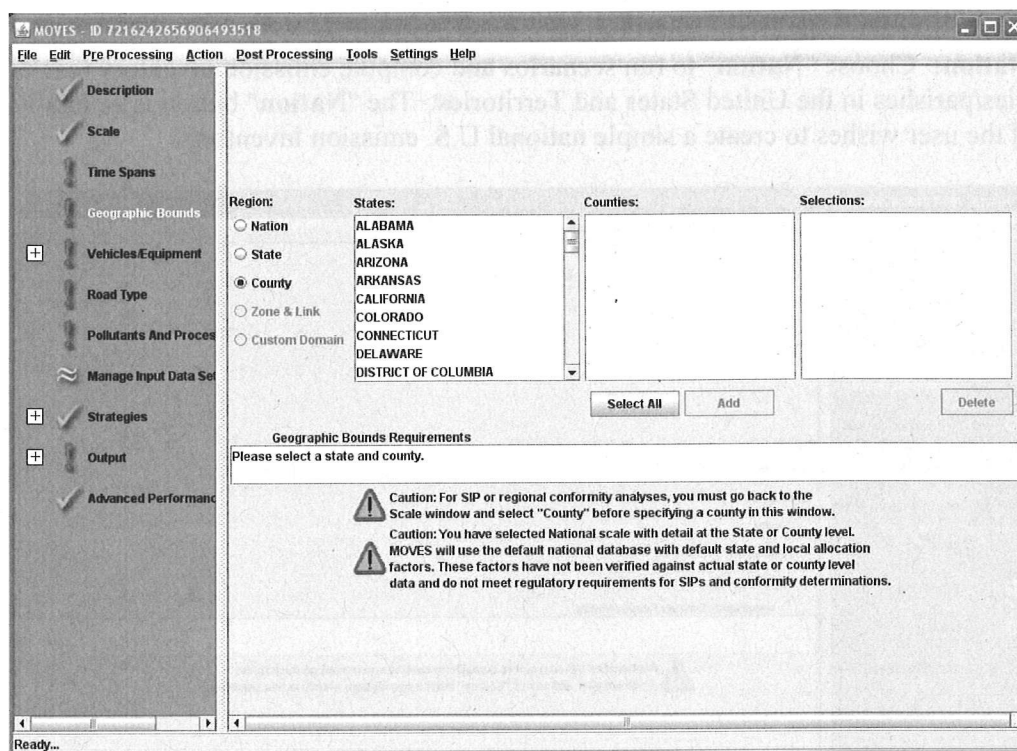
¶ **Note** If the **Project Domain/Scale** is selected, the Run Spec may cover only a single hour..

¶ **Note** The hours selected are run for all chosen days, so it is not possible, for example, to model only from 11:00 PM on Friday to 01:59 AM on Saturday. It is also not possible to run the model for a group of months that span a year end. That is, the user cannot run the model for just December of 2003 through January 2004. However, the model can be run multiple times with different RunSpec time spans to accomplish these tasks, or it can be run for multiple days or years and then manually post-processed to eliminate the un-needed times.

## 2.2.4 Geographic Bounds

Selecting **Geographic Bounds** allows the user to define the region to model. The options available on this panel depend on the choice of **Domain/Scale** and **Calculation Type**

(see Section 2.2.2 of this User Guide). If the user has chosen **Inventory** calculations at the **National** scale, the following screen will appear:



Three buttons, "**Nation**", "**State**" and "**County**" are normally available. Choose the one appropriate for your desired run.

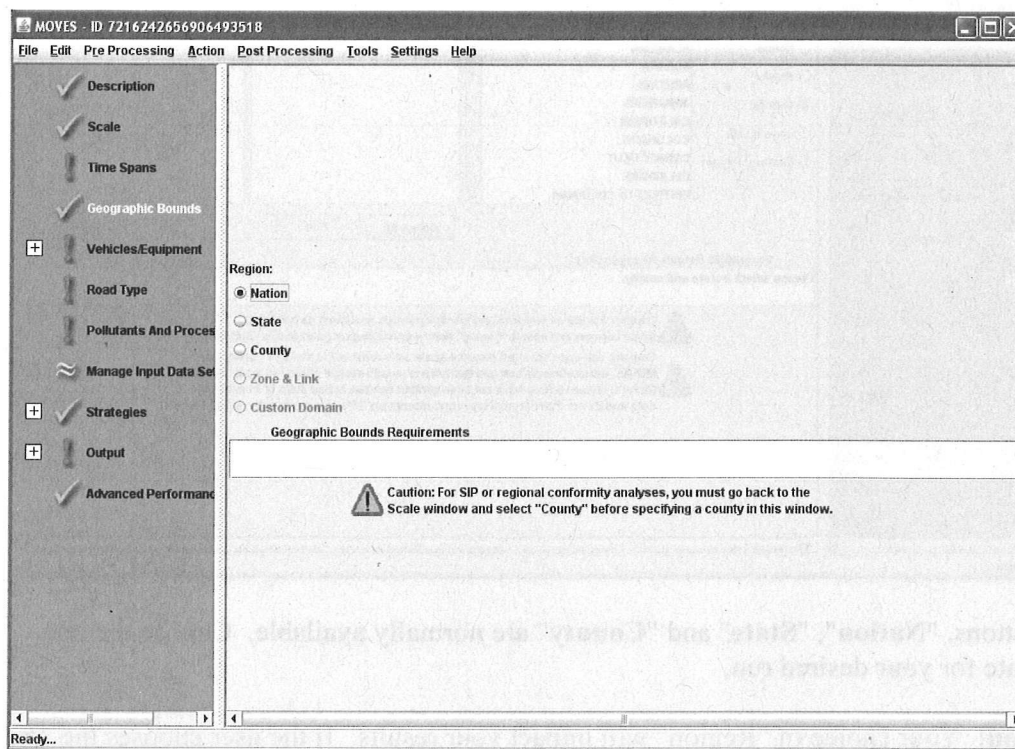
**Caution!** Your choice of "Region" will impact your results. If the user chooses the "**Nation**" or "**State**" input options, the model will pre-aggregate (i.e., compute a weighted average of) all underlying data that is a function of geography (temperatures, road types, etc) and execute the "National" or "State" run as if it were a single county run. Thus, a run at "Nation" or "State" level will be faster than a "County" level run for the same area, but it will be less accurate.

For example, if the user chooses "**State**" and "**Michigan**," the model will average the temperatures of all Michigan counties into a single average set (it will be a set because the temperatures will not necessarily be averaged by time) and perform the simulation with these average values. The report "MOVES2004 Validation Results" includes a sensitivity analysis of different pre-aggregation levels. An alternative method of computing a state simulation is to select "**County**" Region and then selecting all the individual counties in the desired state. If the user chooses this option, no pre-aggregation will be done and the model will execute separate simulations for each county in the particular state. This option will likely produce a slightly different emission result than if the geographic specific data were first aggregated and then used because of non-linear calculations in some algorithms.

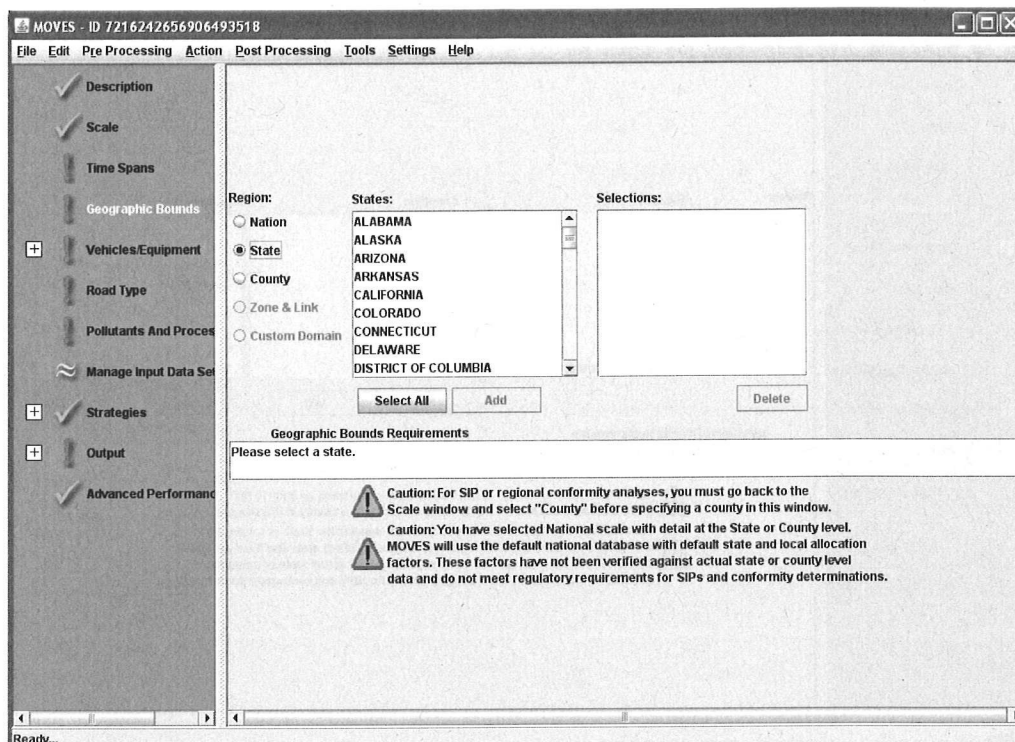


**⚡ Caution!** Be advised that if an entire state or nation is run at the county level, execution times may be quite long. For such runs, we generally recommend multiple-computer processing.

**Nation:** Choose "Nation" to run scenarios and compute emission inventory results for all counties/parishes in the United States and Territories. The "Nation" button is generally chosen if the user wishes to create a simple national U.S. emission inventory.



**State:** Choose "State" to run scenarios and compute emission inventories for particular States or Territories within the United States. This panel has two parts (**States**, and **Selections**) and buttons for choosing the appropriate selections.



To choose the desired state(s):

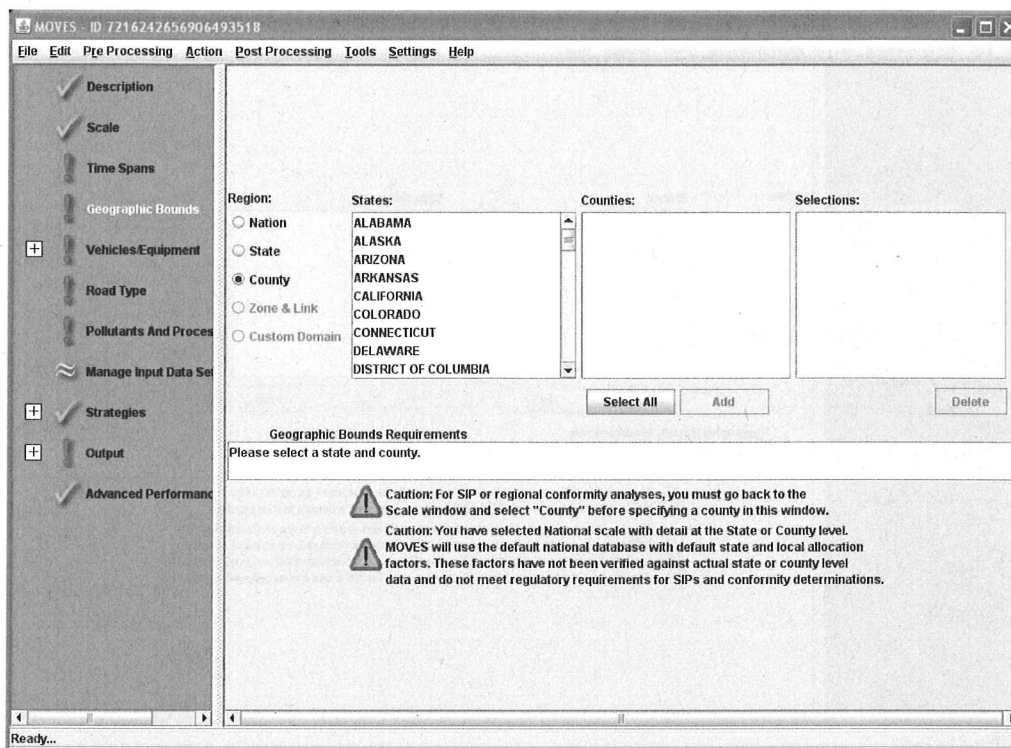
1. Scroll to desired state in "**States**" box and click on it to highlight. Click the "**Add**" button under box. State will appear in the "Selections" box.

2. Click the "**Select All**" button to choose all U.S. states and territories available.

To remove a previously selected state, highlight it in "**Selections**" box and click the "**Delete**" button.

¶ **Note** The "**Add**" and "**Delete**" buttons are inactive if no states are highlighted.

**County:** Choose "**County**" to run scenarios and compute emission inventories for specific counties within a state. If the "**County**" button is selected, a new bounds panel will show in the Detail Panel, as shown below.



The process for selecting a specific county is similar to selecting a state, outlined above.

When using the **County** Domain/Scale and **Inventory** calculation type, the **Region** portion of the **Geographic Bounds** detail panel indicates **County** by default, and the user must specify the state and county in the manner detailed above.

When using the **Project** Domain/Scale and **Inventory** calculation type, the **Region** portion of the **Geographic Bounds** detail panel indicates **County** by default, and the user must specify the state and county in the manner detailed earlier.

When a user selects the **Emission Rates** calculation type on the **Scale** panel, the **Geographic Bounds** panel allows a choice of **Zone & Link** at the **National** scale, and **Zone & Link** or **Custom Domain** at the **County** and **Project** scales.

When either the **County** or **Project** is chosen on the **Scale** panel, the user can also define a "custom county" by selecting the **Custom Domain** option in the **Region** portion of the **Geographic Bounds** panel, as shown in the screen shot below.

MOVES - ID 7216242656906493518

File Edit Pre Processing Action Post Processing Tools Settings Help

☒ Description  
☒ Scale  
☐ Time Spans  
☐ Geographic Bounds  
☒ Vehicles/Equipment  
☐ Road Type  
☐ Pollutants And Processes  
☐ Manage Input Data Set  
☒ Strategies  
☒ Output  
☐ Advanced Performance

Region:

☐ Nation  
☐ State  
☐ County  
☐ Zone & Link  
☒ Custom Domain

Generic County

State ID: 99

County ID: 1 1-999, labels the county within a state.

Description:

GPA Fraction: 0.0 Fraction of county within a fuel Geographic Phase-in Area

Bar. Pressure: 0.0 inHg (avg. for low altitude is 28.9, avg. for high is 24.6)

Vapor Adjust: 0.0 Refueling Vapor Program Adjustment Fraction

Spill Adjust: 0.0 Refueling Spill Program Adjustment Fraction

Domain Input Database

The County domain scale requires a database of detailed data.

Server:

Database:

Enter/Edit Data Refresh

Geographic Bounds Requirements

Please select a domain database.

Ready...

In this case, the user must provide a short county identifier (1-999) that will combine with the state ID (99 for generic) to form the standard 5 digit county identifier used throughout MOVES. In addition, the user must provide a description and GPA (Geographic Phase-in Area) fraction. The Geographic Phase-in Area refers to a group of counties in the western U.S. that had special gasoline sulfur requirements under Tier 2 regulations. See 65 Fed. Reg. 6755-6759 for details on the GPA. The user must also provide the Barometric Pressure (in inches of mercury, inHg) and the Vapor and Spill Adjustment factors, which describe the Stage II Refueling program in the area.

**Note** Barometric Pressure must be indicated by the user for two reasons: 1) it is a required input for some MOVES calculations; and 2) the area is assigned a “Low” or “High” altitude depending on the user’s input (areas with pressure less than or equal to 25.8403 are high altitude; areas with pressure greater than 25.8403 are low altitude).

**Tip** A user might want to use the **Custom Domain** option to represent an area that encompasses a portion of a county, multiple counties, or an area that crosses state boundaries (a nonattainment area may have these characteristics).

When either the **County** or **Project** is chosen on the **Scale** panel, the **Geographic Bounds** panel will contain an area in which to define the **Domain Input Database** for the model, as shown in the following screenshot.

MOVES - ID 7216242656906493518

File Edit Pre Processing Action Post Processing Tools Settings Help

✓ Description  
✓ Scale  
Time Spans  
Geographic Bounds  
+ Vehicles/Equipment  
Road Type  
Pollutants And Proce  
Manage Input Data Ser  
+ Strategies  
+ Output  
✓ Advanced Performance

Region:  
☐ Nation  
☐ State  
☒ County  
☐ Zone & Link  
☐ Custom Domain

States:  
ALABAMA  
ALASKA  
ARIZONA  
ARKANSAS  
CALIFORNIA  
COLORADO  
CONNECTICUT  
DELAWARE  
DISTRICT OF COLUMBIA

Counties:

Selections:

Select All Add Delete

Domain Input Database  
The County domain scale requires a database of detailed data.  
Server:   
Database:  Enter/Edit Data Refresh

Geographic Bounds Requirements  
Please select a state and county.  
Please select a domain database.

Ready...

For both the **County** and **Custom Domain** option, the user must specify the user-supplied database that will contain county-specific data. For Project scale, a database of detailed information on the project is required. See section **2.3.3.3 Data Importer, County Data Manager and Project Domain Manager** for more information. In most cases, this database will be located on the server "localhost" (this is the location where MySQL was originally installed); however, the user can specify a different host server in the **Server** box. After specifying a server, the user must identify a database to receive the data that will be imported. If the user has already created the database, it can be selected by clicking on the drop-down arrow adjacent to the **Database** box and making a selection from the drop-down list. If the database has not been created or if the user wants to make changes to the database, the user can click the **"Enter/Edit Data"** button to open the **County Data Manager**.

**!Tip** The user should be sure to fill out all panels and fields to define the **RunSpec** before using the **County Data Manager** or **Project Domain Manager**. The selections in the Run Spec are used by the **County Data Manager** to define the data needed for the county data to be complete, and to filter the data used when creating export or template files.

**⚠ Caution!** Defaults that are exported from the MOVES database are generally not appropriate for SIP or conformity analyses. See MOVES Policy Guidance document for more information.

**📌 Note** If the user selects **Custom Domain**, the **County Data Manager** will not export any default data; therefore, the user must have all the required information to use this option.

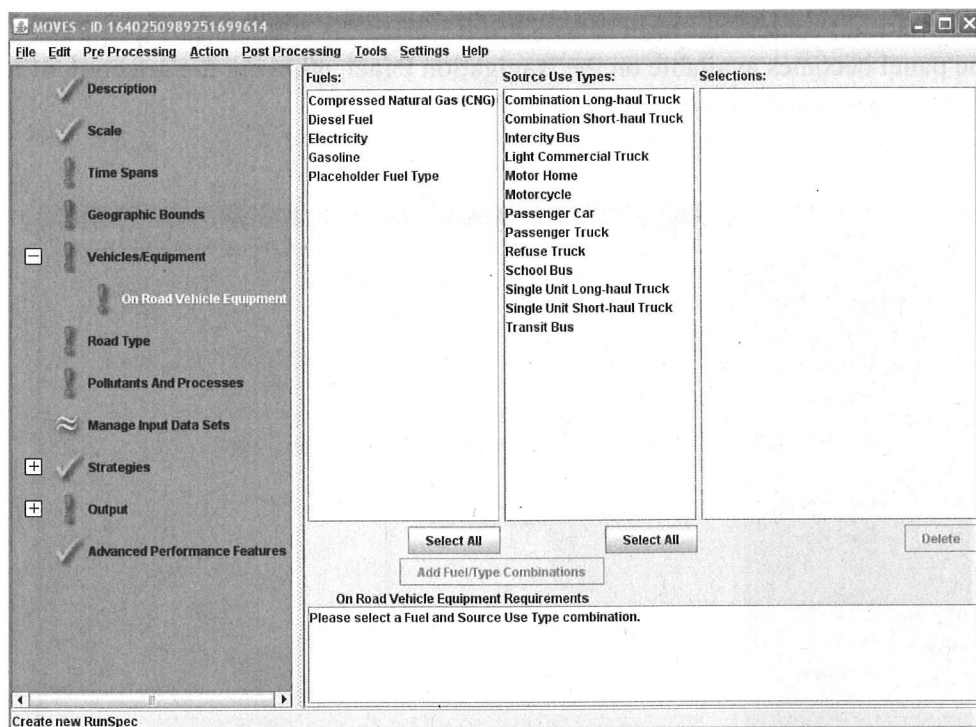
The user can clear the selected database and update the list of available databases by clicking the **"Refresh"** button.

The **Geographic Bounds** panel includes the text box, **Geographic Bounds Requirements**, which defines what is expected of the user on the panel.

### 2.2.5 Vehicles/Equipment

To select which on-road vehicles are to be modeled, click on **Vehicles/Equipment** in the Navigation Panel. This will open the **On Road Vehicle Equipment** Detail Panel. Two lists (**Fuels** and **Source Use Types**) appear to the left of this panel to allow distinct selections for the vehicles/equipment.

**Note** We hope to add additional fuels in the future, and the "Placeholder Fuel Type" is an incomplete step in this direction. However, the Placeholder Fuel is not currently active, and there is no vehicle population assigned to it. Selecting this fuel type should not change model results, but it will generate an error message.



1. Click and highlight the **Fuels** choice or click the **"Select All"** button to choose all the choices.
2. Click and highlight the **Source Use Types** choice or click the **"Select All"** button to choose all the choices.
3. Click **"Add Fuel/Type Combinations"** button to move the selected choices to the **Selections** box.



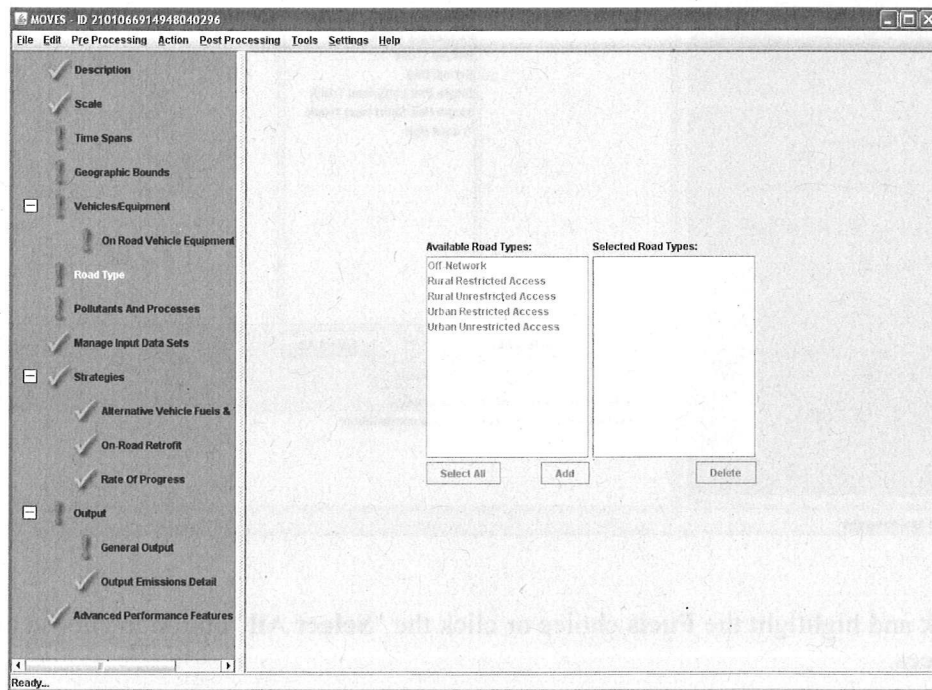
To clear highlighted selections from Selections box click **“Delete”**

**Note** No default selection will appear on this panel, and the model cannot be run without a selection made.

**Note** The **On Road Vehicle Equipment** panel contains a small subpanel at the bottom that reminds the user that they are required to select a Fuel and Source Use type. This panel disappears after the user makes a valid **Fuel** and **Source Use Type** selection. If a selected combination is not in the database, the subpanel will notify the user that the combination is invalid (e.g. “Diesel fuel/Motorcycle combination is not in the database”). These non-existent combinations do not need to be removed from the RunSpec, but no results are reported for such vehicles.

## 2.2.6 Road Type

If any on-road vehicle has been selected (see section 2.3.4 **Vehicles/Equipment**), the **Road Type** panel becomes available on the Navigation Panel, allowing the selection of road type. All **Road Type** panel controls are disabled otherwise.



**Tip** The user must select at least one road type.

1. Click and highlight the desired **Road Types** from the list.

2. Click the **"Select All"** button to select all **Road Types** in list.
3. Click the **"Add"** button to move choice to the **Selected Road Types** box.

Click the **"Delete"** button to clear the highlighted selections.

**Note** The list of **Road Types** is the complete list of roadway types present in the underlying MOVES database. The default database has **Road Types** that represent urban and rural driving on roads with restricted and unrestricted vehicle access. Restricted access roadtypes are usually used to model freeways and interstates; ramps are considered part of restricted access roadtypes. The **Ramp Fraction** tab of the CDM (see section 2.3.3.4) will only become available if an unrestricted road type (i.e. 2 or 4) is selected.

**Note** **"Off Network"** is automatically selected when start or extended idle processes are selected on the **"Pollutants and Processes"** panel. All **Road Types** are automatically selected when any evaporative or refueling processes are selected on the **"Pollutants and Processes"** panel.

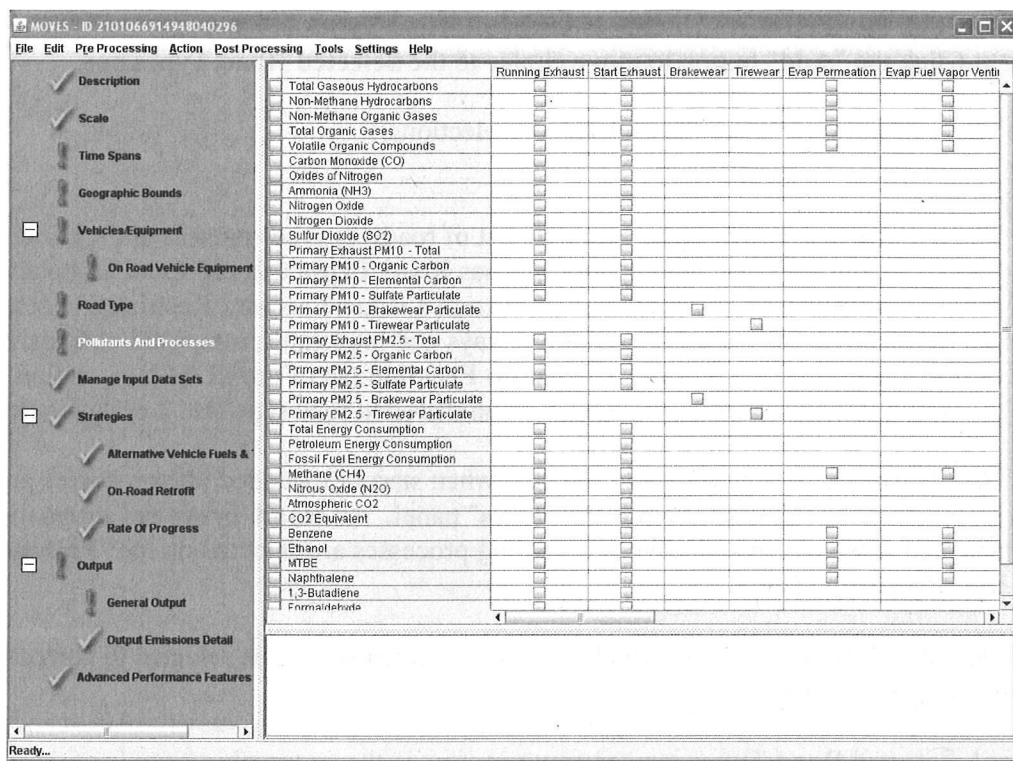
**Tip** All **Road Types** in an area on which VMT is allocated should be selected to correctly calculate population.

**Caution!** Selected **Road Types** may or may not exist in the geographic bounds of the RunSpec. For example, a county may have no urban roads. MOVES will display results only for road types that exist in the selected geographic area.

### 2.2.7 Pollutants and Processes

To select the pollutants and processes to include in the RunSpec, the user should choose **Pollutants and Processes** on the Navigation Panel. A matrix will appear in the Detail Panel, with **"Pollutants"** listed as rows and **"Processes"** listed as columns. When a pollutant/process combination is selected by clicking a check box, it is included in the RunSpec. When the RunSpec is run, the model will compute emissions for that pollutant/process combination.





There is no limit to the number of boxes that may be checked; however, at least one box must be checked to produce a valid MOVES RunSpec input.

**Note** It may be necessary to scroll the **Pollutants and Processes** panel to see all of the Process columns or Pollutant rows.

**Note** Checking the box on the left side of the panel will select all of the processes for that pollutant. The box on the left side of the panel will also show a check if any process is selected. Additionally, unchecking the box on the left side of the panel will deselect all processes for that pollutant.

**Note** In order to replicate MOBIL6.2 runs (e.g. to compare the results of the two models), all processes except extended idle need to be selected.

**Caution!** Users are cautioned that the number of check boxes selected will affect execution time. Large runs calculating many pollutant/process combinations may require processing times of several hours, so users are advised to begin with only a few pollutant/process combinations.

A **Pollutant/Process Requirements** box is included at the bottom of the Detail Panel. Text in this box actively responds to pollutant/process combination selections made in the matrix. In some cases, a specific pollutant/process calculation depends upon another pollutant/process calculation, and the user must select the base pollutant/process combination in

order to be able to select the dependent combination. An error message will appear in the Pollutant/Process Requirements box should the user not select the base combination.

#### 2.2.7.1 Processes

In MOVES2010, "**Process**" refers to the mechanism by which emissions are created. Engine operation creates **Running Emissions Exhaust**, **Start Emissions Exhaust** (the addition to running emissions caused by the engine start), and **Extended Idle Emissions Exhaust** (i.e., "hotelling" emissions from a combination long-haul truck). MOVES also distinguishes **Crankcase Running Exhaust**, **Crankcase Start Exhaust**, and **Crankcase Extended Idle Exhaust** to describe the exhaust gases that escape around the piston rings and enter the crankcase during normal operation.

MOVES models evaporative emissions, situations in which unburned fuel escapes the vehicle's fuel system, through the "**Evap Fuel Vapor Venting**", "**Evap Permeation**", and "**Evap Fuel Leaks**" processes. In addition, vehicle refueling can cause "**Refueling Spillage Loss**" and "**Refueling Displacement Vapor Loss**". "**Brakewear**" and "**Tirewear**" describe the non-exhaust particulate emissions that result from brake use and tire wear.

#### 2.2.7.2 Pollutants

MOVES2010 allows the user to calculate emissions of criteria pollutants, greenhouse gases, and air toxics associated with motor vehicle operation. MOVES also calculates energy consumption. For many pollutants, the emissions calculation is based on the prior calculation of another pollutant emission. As noted earlier, the **Pollutant/Process Requirements** box will display an error message if the user selects a dependent pollutant but not the base pollutant. Pollutants and base pollutants are listed in table 2.2.7.2 below.

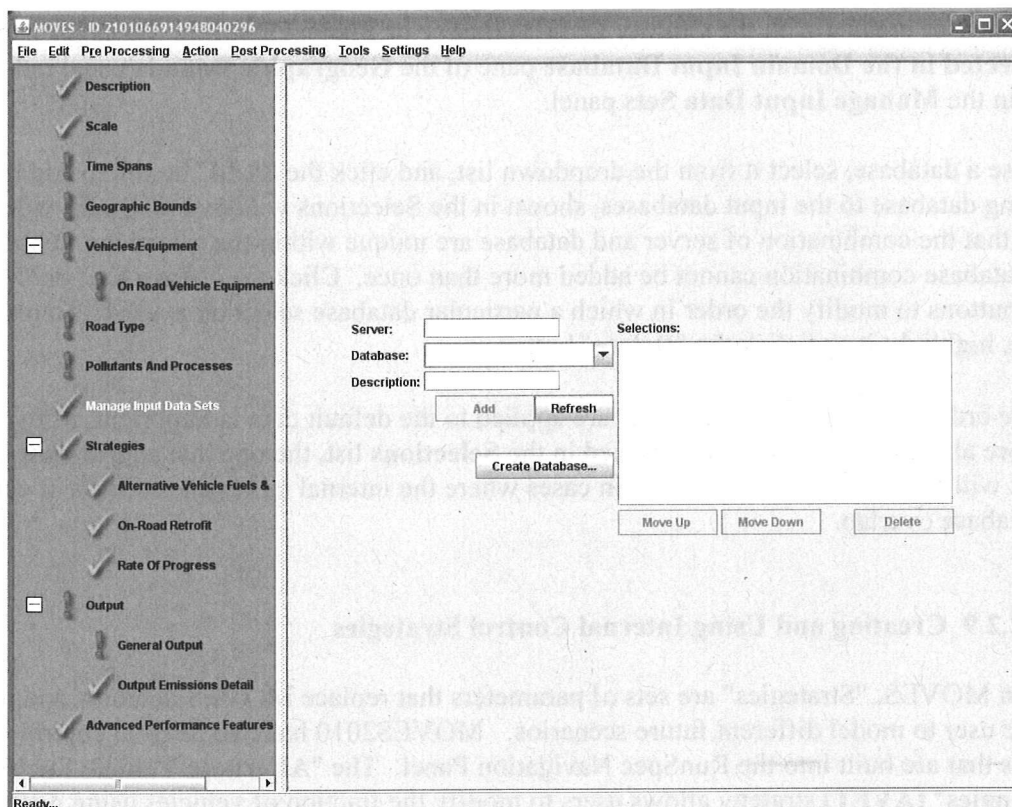
Pollutant Name	Abbreviation	Includes	Depends on
Total Gaseous Hydrocarbons	THC		
Non-methane Hydrocarbons	NMHC		THC and CH <sub>4</sub>
Non-methane Organic Gases	NMOG		NMHC
Total Organic Gases	TOG		NMOG and CH <sub>4</sub>
Volatile Organic Compounds	VOC		NMHC
Carbon Monoxide	CO		
Oxides of Nitrogen	NO <sub>x</sub>	NO, NO <sub>2</sub>	
Ammonia	NH <sub>3</sub>		
Nitrogen Oxide	NO		NO <sub>x</sub>
Nitrogen Dioxide	NO <sub>2</sub>		NO <sub>x</sub>
Sulfur Dioxide	SO <sub>2</sub>		
Primary PM10 Total	PM10	PM10 OC PM10 EC PM10 SO <sub>4</sub>	
PEPM10 Organic Carbon	PM10 OC		PM2.5 OC
PEPM10 Elemental Carbon	PM10 EC		PM2.5 EC
PEPM10 Sulfate Particulate	PM10 SO <sub>4</sub>		Total Energy Consumption
PEPM10 Brakewear Particulate			
PEPM10 Tirewear Particulate			
Primary PM2.5 Total		PM2.5 OC PM2.5 EC PEPM2.5 SO <sub>4</sub>	
PEPM2.5 Organic Carbon	PM2.5 OC		
PEPM2.5 Elemental Carbon	PM2.5 EC		
PEPM2.5 Sulfate Particulate	PM2.5 SO <sub>4</sub>		Total Energy Consumption
PEPM2.5 Brakewear Particulate			
PEPM2.5 Tirewear Particulate			
Total Energy Consumption	TEC		
Petroleum Energy Consumption	PEC		Total Energy Consumption
Fossil Fuel Energy Consumption	FFEC		Total Energy Consumption
Methane (CH <sub>4</sub> )	CH <sub>4</sub>		
Nitrous Oxide (N <sub>2</sub> O)	N <sub>2</sub> O		
Atmospheric Carbon Dioxide (CO <sub>2</sub> )	CO <sub>2</sub>		Total Energy Consumption
CO <sub>2</sub> equivalent	CO <sub>2</sub> e	CO <sub>2</sub> , CH <sub>4</sub> , N <sub>2</sub> O	
Benzene	C <sub>6</sub> H <sub>6</sub>		VOC
Ethanol	C <sub>2</sub> H <sub>5</sub> O		VOC
Methyl tertiary butyl ether (MTBE)	C <sub>5</sub> H <sub>12</sub> O		VOC
Naphthalene	C <sub>10</sub> H <sub>8</sub>		PM10 Total THC
1,3 Butadiene	C <sub>4</sub> H <sub>6</sub>		VOC
Formaldehyde	CH <sub>2</sub> O		VOC
Acetaldehyde	C <sub>2</sub> H <sub>4</sub> O		VOC
Acrolein	C <sub>3</sub> H <sub>4</sub> O		VOC

Table 2.2.7.2

### 2.2.8 Manage Input Data Sets

Select **Manage Input Data Sets** on the Navigation Panel to specify databases containing user-supplied data to be read by the model during execution. Databases entered using this feature will overlay existing databases tables (or portions of a table, if a table containing only a subset of records is provided) in the MOVES default database tables, if they exist. No default selections exist for this panel and the model can be run without any selections on this panel.

**Note** Each database added through this feature can consist of one or more MOVES input database tables.



**Note** These databases must adhere to the MOVES schema and use the database management system (DBMS) used by MOVES. In practice this means that individual tables in the user-supplied database must have identical names as the MOVES default input database and the individual fields in the tables must have identical lengths and types as MOVES default tables. For details regarding the MOVES database schema see the Software Design Reference Manual.

**Tip** The data in user input databases take priority over data from the MOVES default database in construction of the **MOVESExecution** database.

**Note** Entries in the **Manage Input Data Sets** panel will not overwrite or alter the values in the MOVES default database.

To create a new database with empty tables of the proper schema on an existing local or remote server, type the server and a new database name and click the "**Create Database**" button. Leaving the server box blank creates the new database on the localhost server.

You can use database tools outside of MOVES, such as MySQL, to populate new database tables with alternate values. Within MOVES, the **Data Importer**, **County Data Manager**, and **Project Domain Manager** also aid in creating and populating User Input databases. For more information about each of these tools, see sections **2.3.3.3 Data Importer, County Data Manager and Project Domain Manager**.

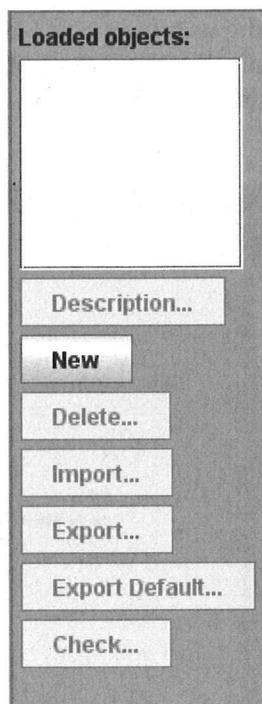
**!Tip** Databases created with the **County Data Manager** or **Project Domain Manager** are **selected in the Domain Input Database** pane of the **Geographic Bounds** panel rather than the **Manage Input Data Sets** panel.

To choose a database, select it from the dropdown list, and click the **"Add"** button to add the new or existing database to the input databases, shown in the Selections window. MOVES will first validate that the combination of server and database are unique within the selections. The same server-database combination cannot be added more than once. Click the **"Move Up"** or **"Move Down"** buttons to modify the order in which a particular database selection is used. To remove a database, highlight it and click the **"Delete"** button.

**!Tip** The order in which these databases are applied to the default data is important. If two or more alternate databases are specified in the **Selections** list, the one that appears last in the list will replace any previous ones in cases where the internal tables and records of each database overlap.

### 2.2.9 Creating and Using Internal Control Strategies

In MOVES, "Strategies" are sets of parameters that replace MOVES defaults, and thus allow the user to model different future scenarios. MOVES2010 has two internal control strategies that are built into the RunSpec Navigation Panel. The **"Alternate Vehicle Fuels & Technologies" (AVFT)** strategy allows users to modify the fraction of vehicles using different fuels and technologies in each model year. The **"On-Road Retrofit"** strategy allows the user to enter information about diesel trucks and buses that have been retrofit with emission control equipment. The use of control strategies is optional; if the user does not select any strategy, the RunSpec will use data from the MOVES default database.



The selection of either strategy opens a split panel on the screen. The left side of the panel is common for both **Strategies** menu options. It is called the **Strategies Management** panel (shown in the screen shot above), and contains buttons that enable the user to add, import, export, delete, and check data. Detailed information specific to the strategy appears to the right, on the **Strategies Detail** panel.

The buttons on the **Strategies Management** panel are used to manipulate the strategy data sets, or "objects", needed for each RunSpec, as explained below. The panels and tools for the AVFT and On-Road Retrofit Strategies are similar, but not identical.

**⚠ Caution!** A RunSpec may have only one AVFT and one **On-Road Retrofit** strategy. The "New" button cannot be selected without first clicking "**Delete**" to remove any existing loaded object. If a strategy data set is already loaded and the user imports a new file, the import action will overwrite the existing data set.

#### 2.2.9.1 Adding Strategy Files

Users can add new strategy data to a RunSpec by clicking the "**New**" button or the "**Import**" button.

Clicking the "**New**" button creates a new strategy data set, which is added to the RunSpec. The new strategy data appears on the list of "**Loaded Objects**" as "(default)". However, in MOVES2010, the "**Loaded Objects**" window does not function as intended and should be ignored.

Clicking the "**Import**" button allows the user to import an appropriate input file. The AVFT Strategy allows the user to import XML, tab-delimited text, and Microsoft Excel files.